STAAD.*Pro* 2007

TECHNICAL REFERENCE MANUAL

DAA037780-1/0001

Research Engineers International
A Bentley Solutions Center

www.reiworld.com
www.bentley.com/staad

STAAD.*Pro* 2007 is a suite of proprietary computer programs of Research Engineers, a Bentley Solutions Center. Although every effort has been made to ensure the correctness of these programs, REI will not accept responsibility for any mistake, error or misrepresentation in or as a result of the usage of these programs.

RELEASE 2007
Document Revision 2.0

Published April, 2007
Revised June, 2008

About STAAD.*Pro*

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

1. Graphical model generation utilities as well as text editor based commands for creating the mathematical model. Beam and column members are represented using lines. Walls, slabs and panel type entities are represented using triangular and quadrilateral finite elements. Solid blocks are represented using brick elements. These utilities allow the user to create the geometry, assign properties, orient cross sections as desired, assign materials like steel, concrete, timber, aluminum, specify supports, apply loads explicitly as well as have the program generate loads, design parameters etc.
2. Analysis engines for performing linear elastic and pdelta analysis, finite element analysis, frequency extraction, and dynamic response (spectrum, time history, steady state, etc.).
3. Design engines for code checking and optimization of steel, aluminum and timber members. Reinforcement calculations for concrete beams, columns, slabs and shear walls. Design of shear and moment connections for steel members.
4. Result viewing, result verification and report generation tools for examining displacement diagrams, bending moment and shear force diagrams, beam, plate and solid stress contours, etc.
5. Peripheral tools for activities like import and export of data from and to other widely accepted formats, links with other popular softwares for niche areas like reinforced and prestressed concrete slab design, footing design, steel connection design, etc.
6. A library of exposed functions called OpenSTAAD which allows users to access STAAD.Pro's internal functions and routines as well as its graphical commands to tap into STAAD's database and link input and output data to third-party software written using languages like C, C++, VB, VBA, FORTRAN, Java, Delphi, etc. Thus, OpenSTAAD allows users to link in-house or third-party applications with STAAD.Pro.

About the STAADN.*Pro* Documentation

The documentation for STAAD.Pro consists of a set of manuals as described below. These manuals are normally provided only in the electronic format, with perhaps some exceptions such as the Getting Started Manual which may be supplied as a printed book to first time and new-version buyers.

All the manuals can be accessed from the Help facilities of STAAD.Pro. Users who wish to obtain a printed copy of the books may contact Research Engineers. REI also supplies the manuals in the PDF format at no cost for those who wish to print them on their own. See the back cover of this book for addresses and phone numbers.

Getting Started and Tutorials : This manual contains information on the contents of the STAAD.Pro package, computer system requirements, installation process, copy protection issues and a description on how to run the programs in the package. Tutorials that provide detailed and step-by-step explanation on using the programs are also provided.

Examples Manual

This book offers examples of various problems that can be solved using the STAAD engine. The examples represent various structural analyses and design problems commonly encountered by structural engineers.

Graphical Environment

This document contains a detailed description of the Graphical User Interface (GUI) of STAAD.Pro. The topics covered include model generation, structural analysis and design, result verification, and report generation.

Technical Reference Manual

This manual deals with the theory behind the engineering calculations made by the STAAD engine. It also includes an explanation of the commands available in the STAAD command file.

International Design Codes

This document contains information on the various Concrete, Steel, and Aluminum design codes, of several countries, that are implemented in STAAD.

The documentation for the STAAD.Pro Extension component(s) is available separately.

Table of Contents

STAAD.*Pro* Technical Reference Manual

Section 1 General Description 1 -

1.1 Introduction 1 - 1
1.2 Input Generation 2
1.3 Types of Structures 2
1.4 Unit Systems 3
1.5 Structure Geometry and Coordinate Systems 4
1.5.1 Global Coordinate System 5
1.5.2 Local Coordinate System 7
1.5.3 Relationship Between Global & Local Coordinates 11
1.6 Finite Element Information 18
1.6.1 Plate/Shell Element 18
1.6.2 Solid Element 31
1.6.3 Surface Element 34
1.7 Member Properties 36
1.7.1 Prismatic Properties 38
1.7.2 Built-In Steel Section Library 40
1.7.3 User Provided Steel Table 41
1.7.4 Tapered Sections 41
1.7.5 Assign Command 41
1.7.6 Steel Joist and Joist Girders 42
1.7.7 Composite Beams and Composite Decks 46
1.7.8 Curved Members 47
1.8 Member/Element Release 47
1.9 Truss/Tension/Compression - Only Members 48
1.10 Tension/Compression - Only Springs 48
1.11 Cable Members 49
1.11.1 Linearized Cable Members 49
1.11.2 Non Linear Cable & Truss Members 1 - 52

1.12 Member Offsets 1 - 53
1.13 Material Constants 54
1.14 Supports 55
1.15 Master/Slave Joints 56
1.16 Loads 56
1.16.1 Joint Load 56
1.16.2 Member Load 57
1.16.3 Area Load / Oneway Load / Floor Load 58
1.16.4 Fixed End Member Load 60
1.16.5 Prestress and Poststress Member Load 60
1.16.6 Temperature/Strain Load 63
1.16.7 Support Displacement Load 63
1.16.8 Loading on Elements 63
1.17 Load Generator 65
1.17.1 Moving Load Generator 65
1.17.2 Seismic Load Generator based on UBC, IBC and other codes 66
1.17.3 Wind Load Generator 67
1.17.4 Snow Load 68
1.18 Analysis Facilities 68
1.18.1 Stiffness Analysis 69
1.18.2 Second Order Analysis 74
1.18.2.1 P-Delta Analysis - Overview 74
1.18.2.1.1 P-Delta Analysis – Large Delta & Small Delta 75
1.18.2.1.2 P-Delta Kg Analysis 77
1.18.2.1.3 P-Delta K+Kg Dynamic Analysis 78
1.18.2.1.4 AISC 360-05 DIRECT Analysis 79
1.18.2.2 Buckling Analysis 80
1.18.2.2.1 Buckling Analysis – Basic Solver 81
1.18.2.2.2 Buckling Analysis – Advanced Solver 82
1.18.2.3 Non Linear Analysis 83
1.18.2.4 Imperfection Analysis 84
1.18.2.5 Multi-Linear Analysis 1 - 84

1.18.2.6 Tension / Compression Only Analysis 1 - 85
1.18.2.7 Nonlinear Cable/Truss Analysis 85
1.18.3 Dynamic Analysis 88
1.18.3.1 Solution of the Eigenproblem 88
1.18.3.2 Mass Modeling 88
1.18.3.3 Damping Modeling 90
1.18.3.4 Response Spectrum 90
1.18.3.5 Response Time History 91
1.18.3.6 Steady State and Harmonic Response 93
1.18.3.7 Pushover Analysis 95
1.19 Member End Forces 96
1.19.1 Secondary Analysis 101
1.19.2 Member Forces at Intermediate Sections 101
1.19.3 Member Displacements at Intermediate Sections 101
1.19.4 Member Stresses at Specified Sections 102
1.19.5 Force Envelopes 102
1.20 Multiple Analyses 103
1.21 Steel/Concrete/Timber Design 104
1.22 Footing Design 104
1.23 Printing Facilities 104
1.24 Plotting Facilities 105
1.25 Miscellaneous Facilities 105
1.26 Post Processing Facilities 1 - 106

Section 2 American Steel Design 2 -

2.1 Design Operations 2 - 1
2.2 Member Properties 2
2.2.1 Built - in Steel Section Library 2
2.3 Allowables per AISC Code 7
2.3.1 Tension Stress 7
2.3.2 Shear Stress 7
2.3.3 Stress Due To Compression 7
2.3.4 Bending Stress 7
2.3.5 Combined Compression and Bending 9
2.3.6 Singly Symmetric Sections 2 - 9

2.3.7 Torsion per Publication T114 2 - 9
2.3.8 Design of Web Tapered Sections 11
2.3.9 Slender compression elements 11
2.4 Design Parameters 11
2.5 Code Checking 18
2.6 Member Selection 18
2.6.1 Member Selection by Optimization 19
2.6.2 Deflection Check With Steel Design 20
2.7 Truss Members 20
2.8 Unsymmetric Sections 20
2.9 Composite Beam Design as per AISC-ASD 20
2.10 Plate Girders 22
2.11 Tabulated Results of Steel Design 23
2.12 Weld Design 26
2.13 Steel Design per AASHTO Specifications 29
2.13.1 AASHTO ASD 29
2.13.2 AASHTO LRFD 37
2.14 Steel Design per AISC/LRFD Specification 44
2.14.1 General Comments 45
2.14.2 LRFD Fundamentals 46
2.14.3 Analysis Requirements 47
2.14.4 Section Classification 47
2.14.5 Axial Tension 48
2.14.6 Axial Compression 48
2.14.7 Flexural Design Strength 49
2.14.8 Combined Axial Force And Bending 49
2.14.9 Design for Shear 50
2.14.10 Design Parameters 50
2.14.11 Code Checking and Member Selection 53
2.14.12 Tabulated Results of Steel Design 55
2.14.13 Composite Beam Design per the American LRFD 3rd edition code 55
2.15 Design per American Cold Formed Steel Code 63
2.16 Castellated Beams 2 – 72

2.17 Steel Design per the AISC 360-05 Design Specifications 2 – 86
2.17.1 General Comments 86
2.17.2 Section Classification 87
2.17.3 Axial Tension 88
2.17.4 Axial Compression 88
2.17.5 Flexural Design Strength 89
2.17.6 Design for Shear 90
2.17.7 Design for Combined Forces and Torsion 90
2.17.8 Design Parameters 91
2.17.9 Code Checking and Member Selection 94
2.17.10 Tabulated Results of Steel Design 2 – 94

Section 3 American Concrete Design 3 -

3.1 Design Operations 3 - 1
3.2 Section Types for Concrete Design 2
3.3 Member Dimensions 2
3.4 Design Parameters 3
3.5 Slenderness Effects and Analysis Consideration 6
3.6 Beam Design 7
3.6.1 Design for Flexure 7
3.6.2 Design for Shear 8
3.6.3 Design for Anchorage 8
3.6.4 Description of Output for Beam Design 9
3.6.5 Cracked Moment of Inertia – ACI Beam Design 12
3.7 Column Design 13
3.8 Designing elements, shear walls, slabs 18
3.8.1 Element Design 18
3.8.2 Shear Wall Design 20
3.8.3 Slabs and RC Designer 28
3.8.4 Design of I-shaped beams per ACI-318 3 – 29

Section 4 Timber Design 4 -

4.1 Timber Design 4 - 1
4.2 Design Operations 13
4.3 Input Specification 16
4.4 Code Checking 17
4.5 Orientation of Lamination 18
4.6 Member Selection 4 - 18

Section 5 Commands and Input Instructions 5 -

5.1 Command Language Conventions 5 - 4
5.1.1 Elements of the Commands 5
5.1.2 Command Formats 7
5.1.3 Listing of Entities by Specification of Global Ranges 10
5.2 Problem Initiation and Title 12
5.3 Unit Specification 14
5.4 Input/Output Width Specification 16
5.5 Set Command Specification 17
5.6 Separator Command 21
5.7 Page New Command 22
5.8 Page Length/Eject Command 23
5.9 Ignore Specifications 24
5.10 No Design Specification 25
5.11 Joint Coordinates Specification 26
5.12 Member Incidences Specification 31
5.13 Elements and Surfaces 35
5.13.1 Plate and Shell Element Incidence Specification 36
5.13.2 Solid Element Incidences Specification 38
5.13.3 Surface Entities Specification 40
5.14 Plate Element Mesh Generation 44
5.14.1 Element Mesh Generation 45
5.14.2 Persistency of Parametric Mesh Models in the STAAD Input File 51
5.15 Redefinition of Joint and Member Numbers 5 - 54

5.16 Entities as single objects 5 - 56
5.16.1 Listing of entities by Specification of GROUPS 57
5.16.2 Physical Members 60
5.17 Rotation of Structure Geometry 63
5.18 Inactive/Delete Specification 64
5.19 User Steel Table Specification 66
5.20 Member Property Specification 76
5.20.1 Type Specs and Addl. Specs for assigning properties from Steel Tables 80
5.20.2 Prismatic Property Specification 85
5.20.2.1 Prismatic Tapered Tube Property Specification 87
5.20.3 Tapered Member Specification 89
5.20.4 Property Specification from User Provided Table 90
5.20.5 Assign Profile Specification 91
5.20.6 Examples of Member Property Specification 92
5.20.7 Composite Decks 94
5.20.8 Curved Member Specification 98
5.20.9 Applying Fireproofing on members 110
5.20.10 Member Property Reduction Factors 115
5.21 Element/Surface Property Specification 117
5.21.1 Element Property Specification 118
5.21.2 Surface Property Specification 119
5.22 Member/Element Releases 120
5.22.1 Member Release Specification 121
5.22.2 Element Release Specification 124
5.22.3 Element Ignore Stiffness 126
5.23 Member Truss/Cable/Tension/Compression Specification 127
5.23.1 Member Truss Specification 128
5.23.2 Member Cable Specification 130
5.23.3 Member Tension/Compression Specification 132
5.24 Element Plane Stress and Inplane Rotation Specifications 137
5.25 Member Offset Specification 5 - 139

5.26 Specifying and Assigning Material Constants 5 - 141
5.26.1 The Define Material Command 143
5.26.2 Specifying CONSTANTS for members, plate elements and solid elements 145
5.26.3 Surface Constants Specification 153
5.26.4 Modal Damping Information 155
5.26.5 Composite Damping for Springs 158
5.26.6 Member Imperfection Information 159
5.27 Support Specifications 161
5.27.1 Global Support Specification 162
5.27.2 Inclined Support Specification 166
5.27.3 Automatic Spring Support Generator for Foundations 169
5.27.4 Multi-linear Spring Support Specification 175
5.27.5 Spring Tension/Compression Specification 178
5.28 Master/Slave Specification 183
5.29 Draw Specifications 186
5.30 Miscellaneous Settings for Dynamic Analysis 187
5.30.1 Cut-Off Frequency, Mode Shapes or Time 188
5.30.2 Mode Selection 190
5.31 Definition of Load Systems 192
5.31.1 Definition of Moving Load System 193
5.31.2 Definitions for Static Force Procedures for Seismic Analysis 198
5.31.2.1 UBC 1997 Load Definition 199
5.31.2.2 UBC 1994 or 1985 Load Definition 204
5.31.2.3 Colombian Seismic Load 209
5.31.2.4 Japanese Seismic Load 212
5.31.2.5 Definition of Lateral Seismic Load per IS:1893 215
5.31.2.6 IBC 2000/2003 Load Definition 221
5.31.2.7 CFE (Comision Federal De Electricidad) Seismic Load 228
5.31.2.8 NTC (Normas Técnicas Complementarias) Seismic Load 232
5.31.2.9 RPA (Algerian) Seismic Load 237
5.31.2.10 Canadian Seismic Code (NRC) 1995 5 – 241

5.31.2.11 Canadian Seismic Code (NRC) 2005 – Vol I 5 - 247
5.31.2.12 Turkish Seismic Code 255
5.31.2.13 IBC 2006 Seismic Load Definition 262
5.31.3 Definition of Wind Load 269
5.31.4 Definition of Time History Load 274
5.31.5 Definition of Snow Load 280
5.31.6 Reference Load Types - Definition 282
5.31.7 Definition of Direct Analysis Members 284
5.32 Loading Specifications 286
5.32.1 Joint Load Specification 288
5.32.2 Member Load Specification 289
5.32.3 Element Load Specifications 292
5.32.3.1 Element Load Specification - Plates 293
5.32.3.2 Element Load Specification - Solids 299
5.32.3.3 Element Load Specification - Joints 301
5.32.3.4 Surface Loads Specification 304
5.32.4 Area Load/One Way Load/Floor Load Specification 307
5.32.5 Prestress Load Specification 321
5.32.6 Temperature Load Specification 328
5.32.7 Fixed-End Load Specification 330
5.32.8 Support Joint Displacement Specification 331
5.32.9 Selfweight Load Specification 334
5.32.10 Dynamic Loading Specification 335
5.32.10.1 Response Spectrum Analysis 336
5.32.10.1.1 Response Spectrum Specification - Generic method 337
5.32.10.1.2 Response Spectrum Specification in Conjunction with the Indian IS: 1893 (Part 1)-2002 Code for Dynamic Analysis 343
5.32.10.1.3 Response Spectrum Specification per Eurocode 8 1996 348
5.32.10.1.4 Response Spectrum Specification per Eurocode 8 2004 354
5.32.10.1.5 Response Spectrum Specification in accordance with IBC 2006 5 - 361

5.32.10.2 Application of Time Varying Load for Response History Analysis 5 - 366
5.32.11 Repeat Load Specification 369
5.32.12 Generation of Loads 372
5.32.13 Generation of Snow Loads 386
5.33 Reference Load Cases – Application 388
5.34 Frequency Calculation 389
5.34.1 Rayleigh Frequency Calculation 390
5.34.2 Modal Calculation Command 392
5.35 Load Combination Specification 393
5.36 Calculation of Problem Statistics 398
5.37 Analysis Specification 399
5.37.1 Linear Elastic Analysis 400
5.37.2 PDELTA Analysis options 403
5.37.3 CABLE Analysis (Non Linear) 409
5.37.4 BUCKLING Analysis 411
5.37.5 DIRECT Analysis 414
5.37.6 Steady State & Harmonic Analysis 416
5.37.6.1 Purpose 417
5.37.6.2 Define Harmonic Output Frequencies 420
5.37.6.3 Define Load Case Number 421
5.37.6.4 Steady Ground Motion Loading 422
5.37.6.5 Steady Force Loading 424
5.37.6.6 Harmonic Ground Motion Loading 427
5.37.6.7 Harmonic Force Loading 430
5.37.6.8 Print Steady State/Harmonic Results 434
5.37.6.9 Last Line of Steady State/Harmonic Analysis 437
5.37.7 Pushover Analysis 438
5.38 Change Specification 439
5.39 Load List Specification 442
5.40 Load Envelope 444
5.41 Section Specification 446
5.42 Print Specifications (includes CG and Story Drift) 448
5.43 Stress/Force output printing for Surface Entities 456
5.44 Printing Section Displacements 458
5.45 Print the Force Envelope 5 - 460

5.46 Post Analysis Printer Plot Specifications 5 - 462
5.47 Size Specification 463
5.48 Steel and Aluminum Design Specifications 465
5.48.1 Parameter Specifications 466
5.48.2 Code Checking Specification 469
5.48.3 Member Selection Specification 470
5.48.4 Member Selection by Optimization 472
5.48.5 Weld Selection Specification 473
5.49 Group Specification 474
5.50 Steel and Aluminum Take Off Specification 477
5.51 Timber Design Specifications 479
5.51.1 Timber Design Parameter Specifications 480
5.51.2 Code Checking Specification 481
5.51.3 Member Selection Specification 482
5.52 Concrete Design Specifications 483
5.52.1 Design Initiation 484
5.52.2 Concrete Design-Parameter Specification 485
5.52.3 Concrete Design Command 487
5.52.4 Concrete Take Off Command 488
5.52.5 Concrete Design Terminator 489
5.53 Footing Design Specifications 490
5.54 Shear Wall Design 491
5.54.1 Definition of Wall Panels for Shear Wall Design 493
5.54.2 Shear Wall Design Initiation 494
5.55 End Run Specification 5 - 496

Index

Notes

General Description

Section 1

1.1 Introduction

The **STAAD.*Pro* 2007** Graphical User Interface (GUI) is normally used to create all input specifications and all output reports and displays (See the Graphical Environment manual). These structural modeling and analysis input specifications are stored in a text file with extension ".STD". When the GUI does a File Open to start a session with an existing model, it gets all of its information from the STD file. A user may edit/create this STD file and have the GUI and the analysis engine both reflect the changes.

The STD file is processed by the STAAD analysis "engine" to produce results that are stored in several files with extensions such as ANL, BMD, TMH, etc. The ANL text file contains the printable output as created by the specifications in this manual. The other files contain the results (displacements, member/element forces, mode shapes, section forces/moments/displacements, etc.) that are used by the GUI in post processing mode.

This section of the manual contains a general description of the analysis and design facilities available in the STAAD engine. Specific information on steel, concrete, and timber design is available in Sections 2, 3, and 4 of this manual, respectively. Detailed STAAD engine STD file command formats and other specific user information is presented in Section 5.

The objective of this section is to familiarize the user with the basic principles involved in the implementation of the various analysis/design facilities offered by the STAAD engine. As a general rule, the sequence in which the facilities are discussed follows the recommended sequence of their usage in the STD input file.

1.2 Input Generation

The GUI (or user) communicates with the STAAD analysis engine through the STD input file. That input file is a text file consisting of a series of commands which are executed sequentially. The commands contain either instructions or data pertaining to analysis and/or design. The elements and conventions of the STAAD command language are described in Section 5 of this manual.

The STAAD input file can be created through a text editor or the GUI Modeling facility. In general, any text editor may be utilized to edit/create the STD input file. The GUI Modeling facility creates the input file through an interactive menu-driven graphics oriented procedure.

1.3 Types of Structures

A STRUCTURE can be defined as an assemblage of elements. STAAD is capable of analyzing and designing structures consisting of both frame, plate/shell and solid elements. Almost any type of structure can be analyzed by STAAD.

For input, see section 5.2

A **SPACE** structure, which is a three dimensional framed structure with loads applied in any plane, is the most general.

A **PLANE** structure is bound by a global X-Y coordinate system with loads in the same plane.

A **TRUSS** structure consists of truss members which can have only axial member forces and no bending in the members.

A **FLOOR** structure is a two or three dimensional structure having no horizontal (global X or Z) movement of the structure [FX, FZ & MY are restrained at every joint]. The floor framing (in global X-Z plane) of a building is an ideal example of a FLOOR structure. Columns can also be modeled with the floor in a FLOOR structure as long as the structure has no horizontal loading. If there is any horizontal load, it must be analyzed as a SPACE structure.

Specification of the correct structure type reduces the number of equations to be solved during the analysis. This results in a faster and more economic solution for the user. The degrees of freedom associated with frame elements of different types of structures is illustrated in Figure 1.1.

Structure Types

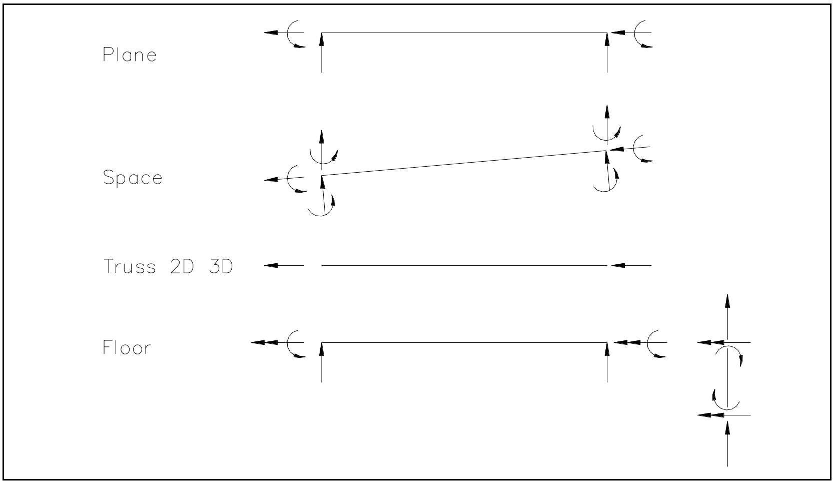


Figure 1.1

1.4 Unit Systems

For input, see section 5.3

The user is allowed to input data and request output in almost all commonly used engineering unit systems including MKS, SI and FPS. In the input file, the user may change units as many times as required. Mix and match between length and force units from different unit systems is also allowed. The input-unit for angles (or rotations) is degrees. However, in JOINT DISPLACEMENT output, the rotations are provided in radians. For all output, the units are clearly specified by the program.

1.5 Structure Geometry and Coordinate Systems

A structure is an assembly of individual components such as beams, columns, slabs, plates etc.. In STAAD, frame elements and plate elements may be used to model the structural components. Typically, modeling of the structure geometry consists of two steps:

A. Identification and description of joints or nodes.

B. Modeling of members or elements through specification of connectivity (incidences) between joints.

For input, see sections 5.11 to 5.17

In general, the term MEMBER will be used to refer to frame elements and the term ELEMENT will be used to refer to plate/shell and solid elements. Connectivity for MEMBERs may be provided through the MEMBER INCIDENCE command while connectivity for ELEMENTs may be provided through the ELEMENT INCIDENCE command.

STAAD uses two types of coordinate systems to define the structure geometry and loading patterns. The GLOBAL coordinate system is an arbitrary coordinate system in space which is utilized to specify the overall geometry & loading pattern of the structure. A LOCAL coordinate system is associated with each member (or element) and is utilized in MEMBER END FORCE output or local load specification.

1.5.1 Global Coordinate System

The following coordinate systems are available for specification of the structure geometry.

A. Conventional Cartesian Coordinate System: This coordinate system (Fig. 1.2) is a rectangular coordinate system (X, Y, Z) which follows the orthogonal right hand rule. This coordinate system may be used to define the joint locations and loading directions. The translational degrees of freedom are denoted by u_1, u_2, u_3 and the rotational degrees of freedom are denoted by u_4, u_5 & u_6.

B. Cylindrical Coordinate System: In this coordinate system, (Fig. 1.3) the X and Y coordinates of the conventional cartesian system are replaced by R (radius) and Ø (angle in degrees). The Z coordinate is identical to the Z coordinate of the cartesian system and its positive direction is determined by the right hand rule.

C. Reverse Cylindrical Coordinate System: This is a cylindrical type coordinate system (Fig. 1.4) where the R- Ø plane corresponds to the X-Z plane of the cartesian system. The right hand rule is followed to determine the positive direction of the Y axis.

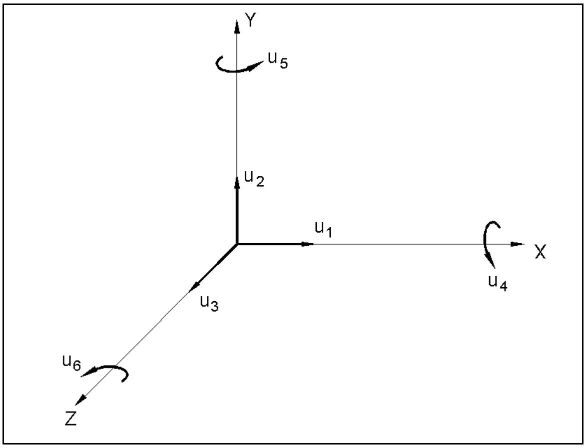


Figure 1.2 : Cartesian (Rectangular) Coordinate System

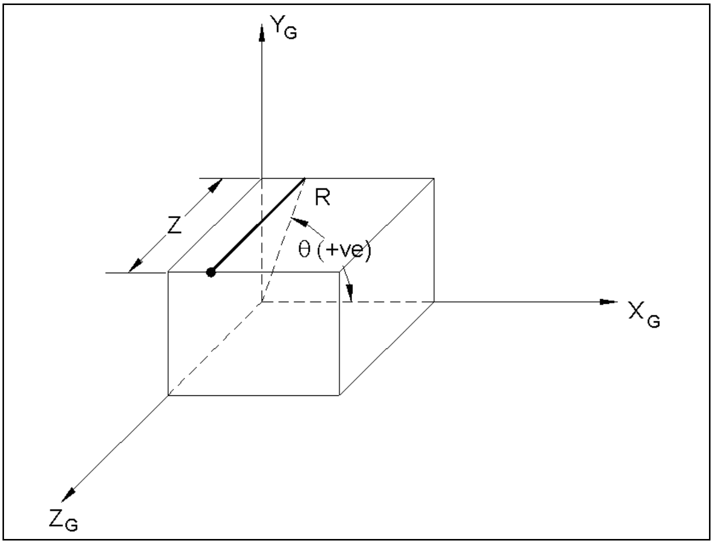


Figure 1.3 : Cylindrical Coordinate System

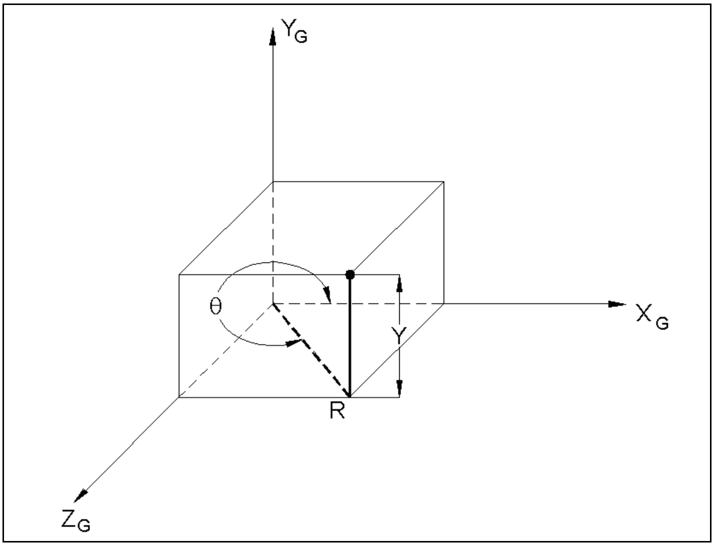


Figure 1.4 : Reverse Cylindrical Coordinate System

1.5.2 Local Coordinate System

A local coordinate system is associated with each member. Each axis of the local orthogonal coordinate system is also based on the right hand rule. Fig. 1.5 shows a beam member with start joint 'i' and end joint 'j'. The positive direction of the local x-axis is determined by joining 'i' to 'j' and projecting it in the same direction. The right hand rule may be applied to obtain the positive directions of the local y and z axes. The local y and z-axes coincide with the axes of the two principal moments of inertia. The local coordinate system is always rectangular.

A wide range of cross-sectional shapes may be specified for analysis. These include rolled steel shapes, user specified prismatic shapes etc.. Fig. 1.6 shows local axis system(s) for these shapes.

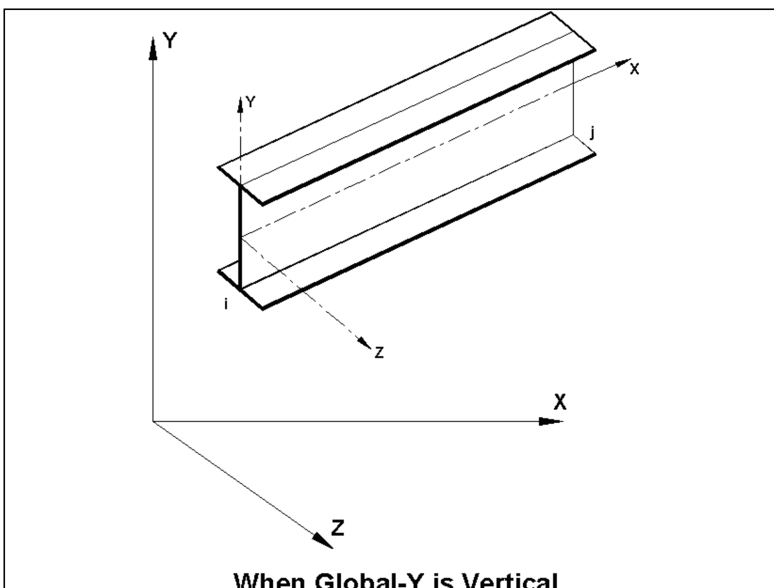


Figure 1.5a

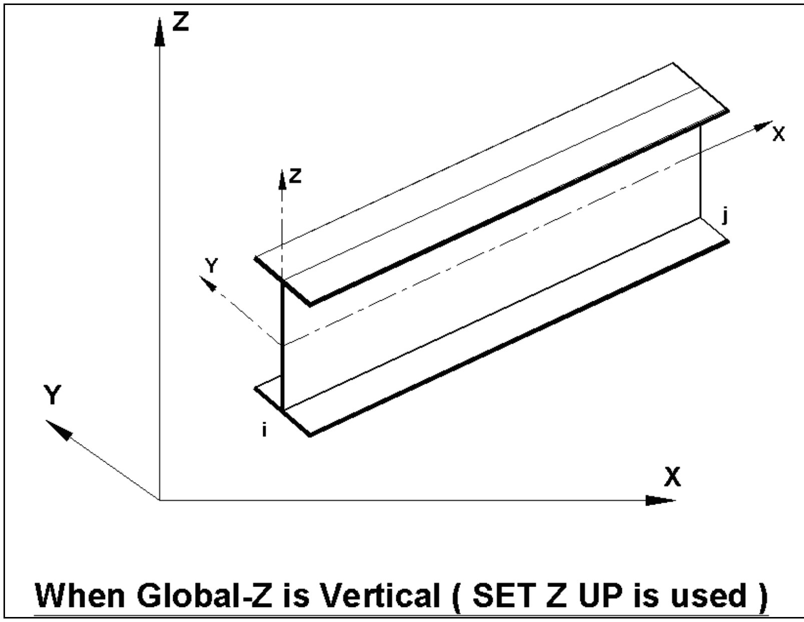
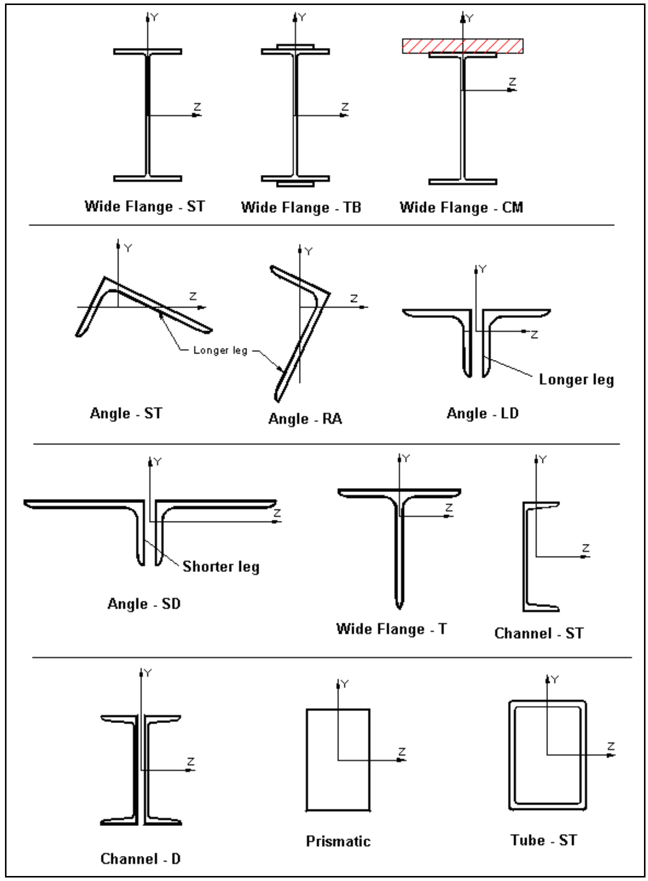


Figure 1.5b



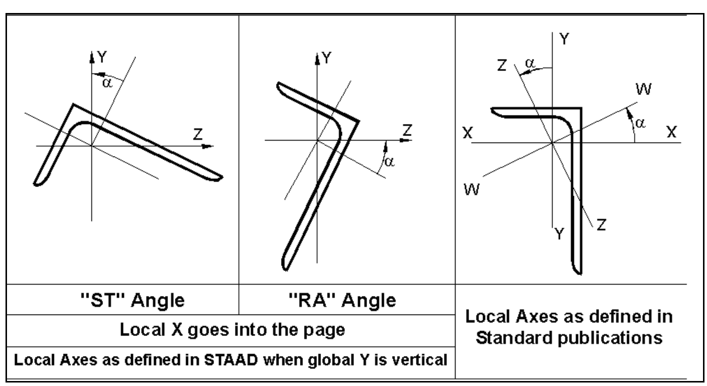


Figure 1.6a - Local axis system for various cross sections when global Y axis is vertical.

NOTE: The local x-axis of the above sections is going into the paper

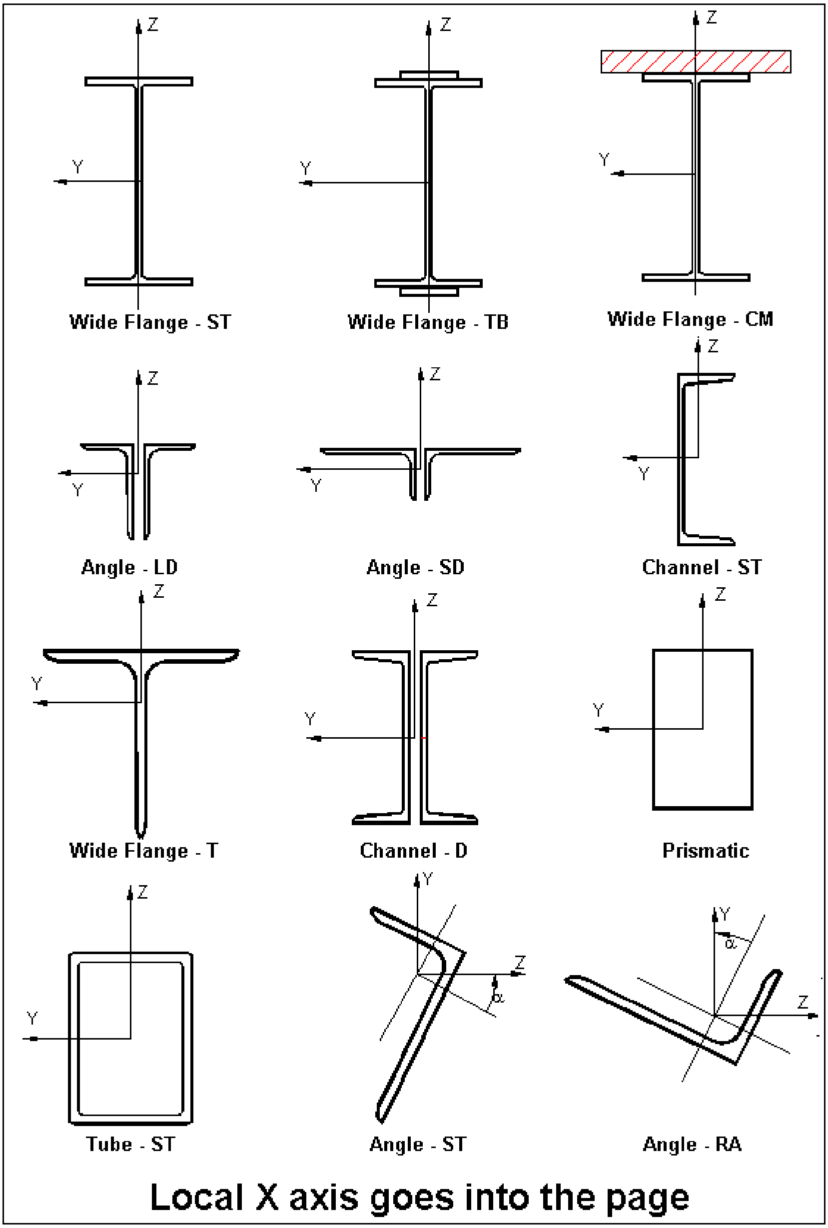


Figure 1.6b - Local axis system for various cross sections when global Z axis is vertical (SET Z UP is specified).

1.5.3 Relationship Between Global & Local Coordinates

Since the input for member loads can be provided in the local and global coordinate system and the output for member-end-forces is printed in the local coordinate system, it is important to know the relationship between the local and global coordinate systems. This relationship is defined by an angle measured in the following specified way. This angle will be defined as the beta (β) angle. For offset members the beta angle/reference point specifications are based on the offset position of the local axis, not the joint positions.

Beta Angle

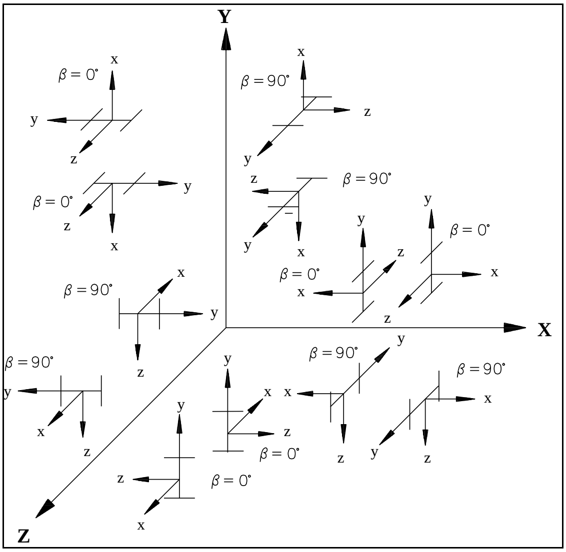
For input, see section 5.26

When the local x-axis is parallel to the global Vertical axis, as in the case of a column in a structure, the beta angle is the angle through which the local z-axis (or local Y for SET Z UP) has been rotated about the local x-axis from a position of being parallel and in the same positive direction of the global Z-axis (global Y axis for SET Z UP).

When the local x-axis is not parallel to the global Vertical axis, the beta angle is the angle through which the local coordinate system has been rotated about the local x-axis from a position of having the local z-axis (or local Y for SET Z UP) parallel to the global X-Z plane (or global X-Y plane for SET Z UP)and the local y-axis (or local z for SET Z UP) in the same positive direction as the global vertical axis. Figure 1.7 details the positions for beta equals 0 degrees or 90 degrees. When providing member loads in the local member axis, it is helpful to refer to this figure for a quick determination of the local axis system.

Reference Point

An alternative to providing the member orientation is to input the coordinates (or a joint number) which will be a reference point located in the member x-y plane (x-z plane for SET Z UP) but not on the axis of the member. From the location of the reference point, the program automatically calculates the orientation of the member x-y plane (x-z plane for SET Z UP).



Relationship between Global and Local axes

Figure 1.7

Reference Vector

This is yet another way to specify the member orientation. In the reference point method described above, the X,Y,Z coordinates of the point are in the global axis system. In a reference vector, the X,Y,Z coordinates are specified with respect to the local axis system of the member corresponding to the BETA 0 condition.

A direction vector is created by the program as explained in section 5.26.2 of this manual. The program then calculates the Beta Angle using this vector.

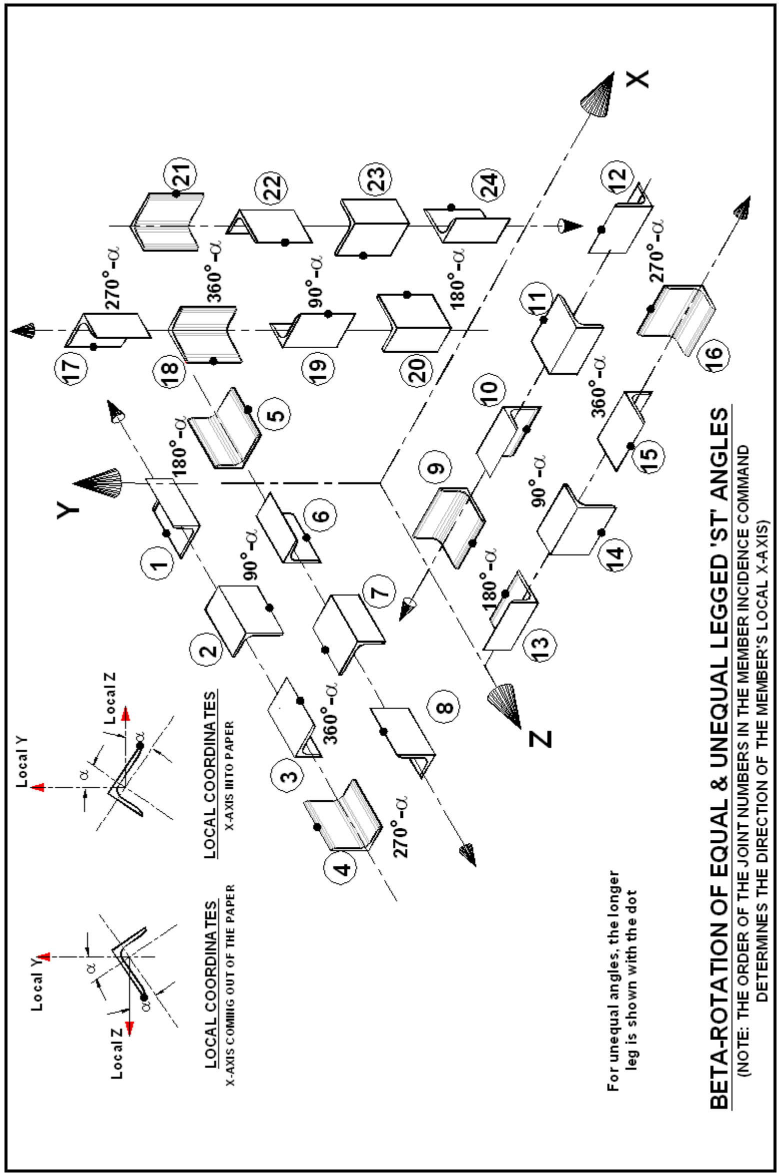

BETA-ROTATION OF EQUAL & UNEQUAL LEGGED 'ST' ANGLES
(NOTE: THE ORDER OF THE JOINT NUMBERS IN THE MEMBER INCIDENCE COMMAND
DETERMINES THE DIRECTION OF THE MEMBER'S LOCAL X-AXIS)
For unequal angles, the longer
leg is shown with the dot
LOCAL COORDINATES
X-AXIS COMING OUT OF THE PAPER
LOCAL COORDINATES
X-AXIS INTO PAPER
Local Y
Local Z
Local Y
Local Z
X
Y
Z

Figure 1.8

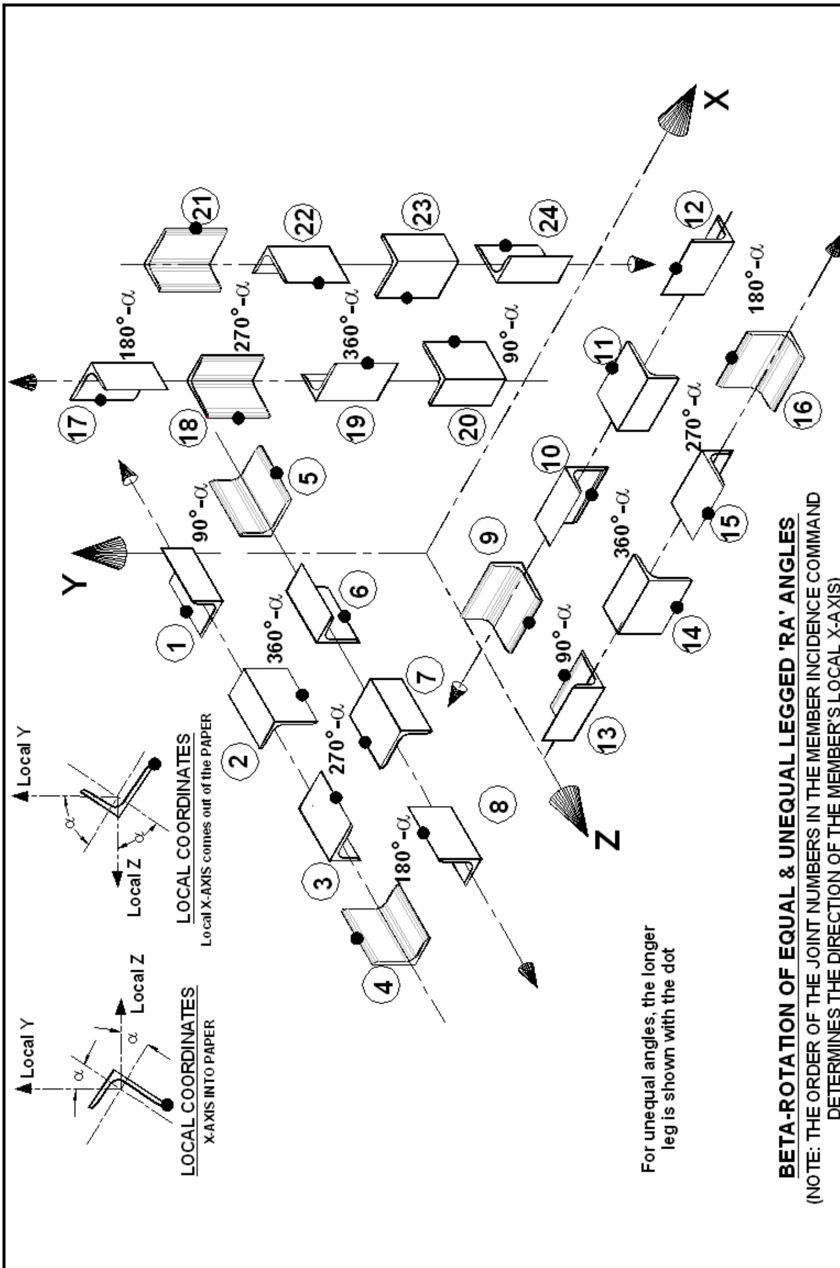

BETA-ROTATION OF EQUAL & UNEQUAL LEGGED 'RA' ANGLES
(NOTE: THE ORDER OF THE JOINT NUMBERS IN THE MEMBER INCIDENCE COMMAND DETERMINES THE DIRECTION OF THE MEMBER'S LOCAL X-AXIS)
For unequal angles, the longer leg is shown with the dot
LOCAL COORDINATES
X-AXIS INTO PAPER
LOCAL COORDINATES
Local X-AXIS comes out of the PAPER
Local Y
Local Z
X
Y
Z

Figure 1.9

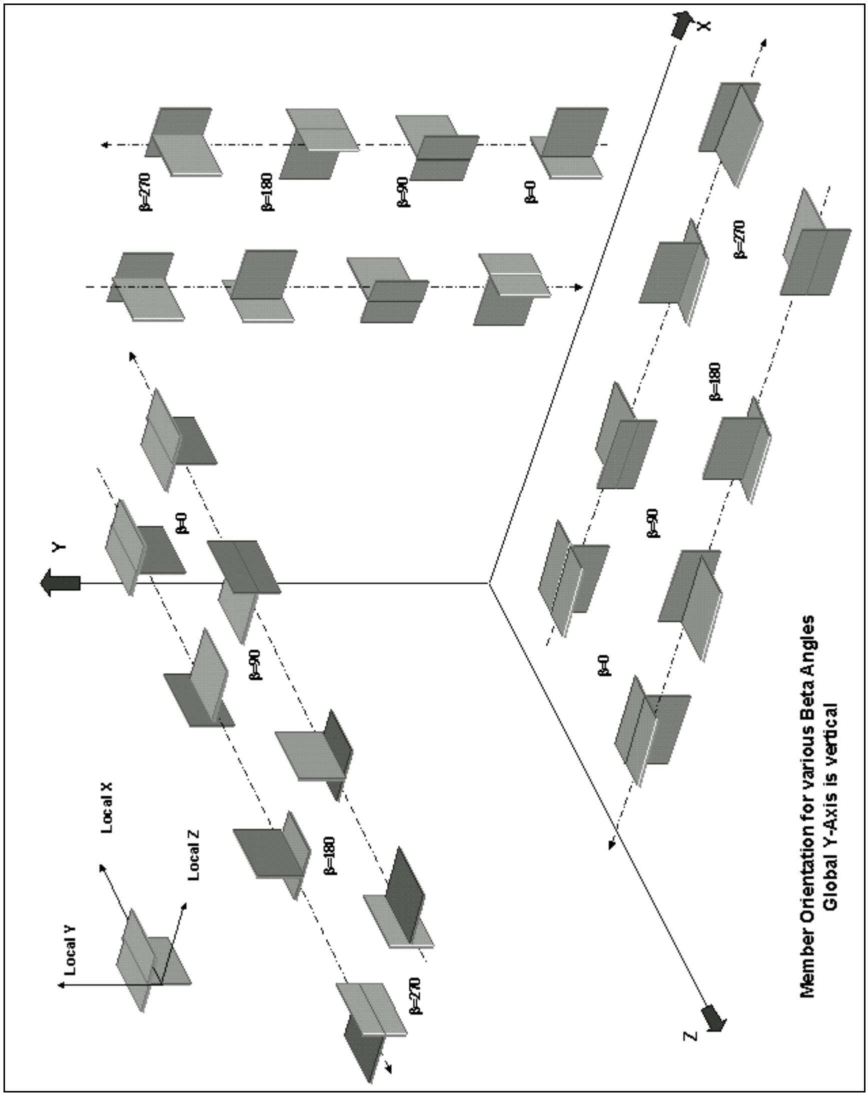

Member Orientation for various Beta Angles
Global Y-Axis is vertical
Local X
Local Y
Local Z
X
Y
Z
β=0
β=90
β=180
β=270

Figure 1.10

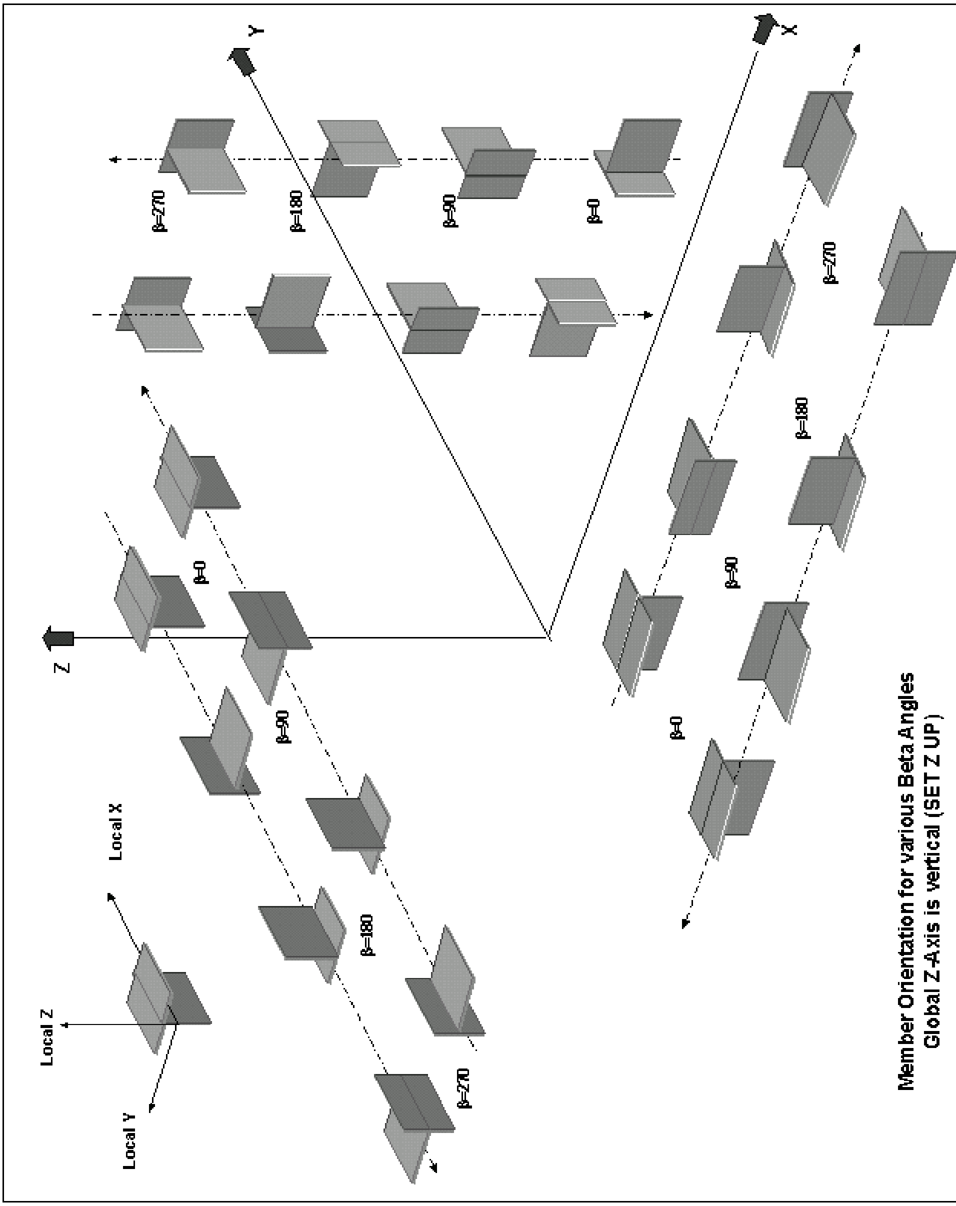

Member Orientation for various Beta Angles
Global Z-Axis is vertical (SET Z UP)
Local X
Local Y
Local Z
X
Y
Z
β=0
β=90
β=180
β=270

Figure 1.11

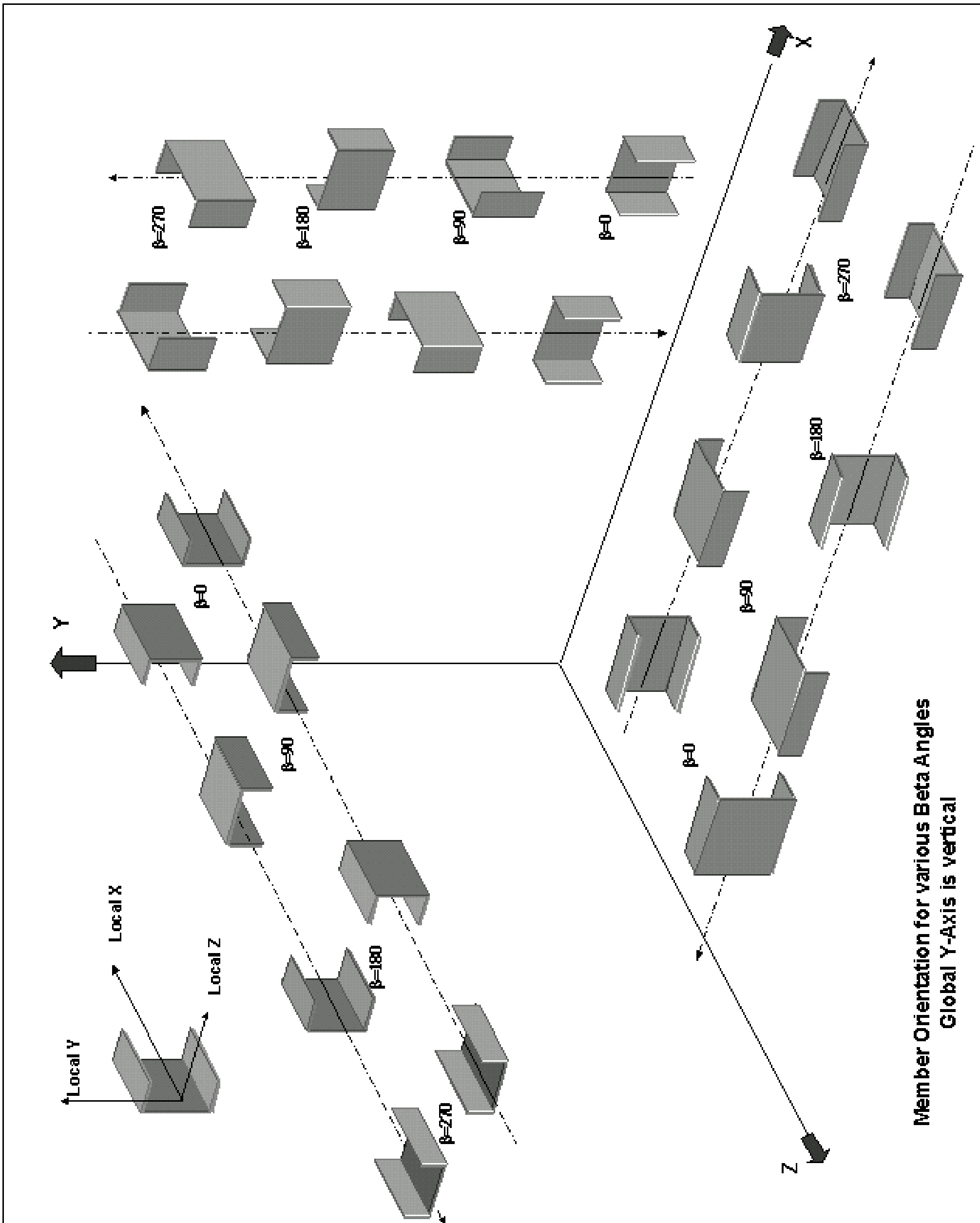

Member Orientation for various Beta Angles
Global Y-Axis is vertical
Local X
Local Y
Local Z
X
Y
Z
β=0
β=90
β=180
β=270

Figure 1.12

1.6 Finite Element Information

For input, see sections 5.11, 5.13, 5.14, 5.21, 5.24, and 5.32.3

STAAD is equipped with a plate/shell finite element, solid finite element and an entity called the surface element. The features of each is explained below.

1.6.1 Plate/Shell Element

The Plate/Shell finite element is based on the hybrid element formulation. The element can be 3-noded (triangular) or 4-noded (quadrilateral). If all the four nodes of a quadrilateral element do not lie on one plane, it is advisable to model them as triangular elements. The thickness of the element may be different from one node to another.

"Surface structures" such as walls, slabs, plates and shells may be modeled using finite elements. For convenience in generation of a finer mesh of plate/shell elements within a large area, a MESH GENERATION facility is available. The facility is described in detail in Section 5.14.

The user may also use the element for PLANE STRESS action only (i.e. membrane/in-plane stiffness only). The ELEMENT PLANE STRESS command should be used for this purpose.

Geometry Modeling Considerations

The following geometry related modeling rules should be remembered while using the plate/shell element

1) The program automatically generates a fictitious fifth node "O" (center node - see Fig. 1.8) at the element center.
2) While assigning nodes to an element in the input data, it is essential that the nodes be specified either clockwise or counter clockwise (Fig. 1.9). For better efficiency, similar elements should be numbered sequentially
3) Element aspect ratio should not be excessive. They should be on the order of 1:1, and preferably less than 4:1.
4) Individual elements should not be distorted. Angles between two adjacent element sides should not be much larger than 90 and never larger than 180.

Load Specification for Plate Elements

Following load specifications are available:

1) Joint loads at element nodes in global directions.
2) Concentrated loads at any user specified point within the element in global or local directions.
3) Uniform pressure on element surface in global or local directions
4) Partial uniform pressure on user specified portion of element surface in global or local directions
5) Linearly varying pressure on element surface in local directions.
6) Temperature load due to uniform increase or decrease of temperature.
7) Temperature load due to difference in temperature between top and bottom surfaces of the element.

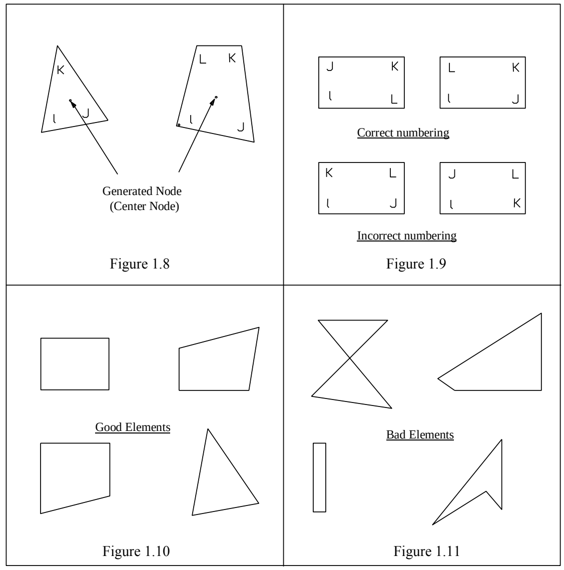


Figure 1.13

Theoretical Basis

The STAAD plate finite element is based on hybrid finite element formulations. A complete quadratic stress distribution is assumed. For plane stress action, the assumed stress distribution is as follows.

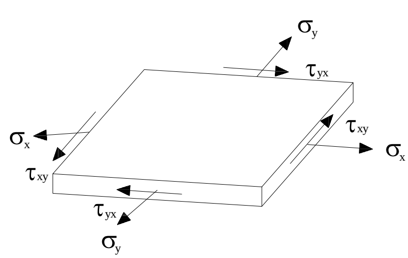


Figure 1.14

Complete quadratic assumed stress distribution:

$$\begin{pmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & x^2 & 2xy & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & y^2 & 0 & 2xy \\ 0 & -y & 0 & 0 & 0 & -x & 1 & -2xy & -y^2 & -x^2 \end{bmatrix} \begin{pmatrix} a_1\\ a_2\\ a_3\\ \vdots\\ a_{10}\end{pmatrix}$$

a_1 through a_{10} = constants of stress polynomials.

The following quadratic stress distribution is assumed for plate bending action:

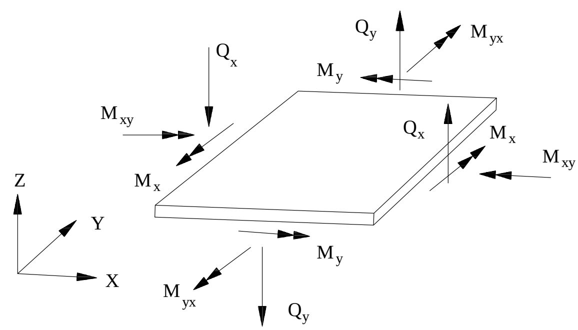


Figure 1.15

Complete quadratic assumed stress distribution:

$$\begin{pmatrix}M_x\\ M_y\\ M_{xy}\\ Q_x\\ Q_y\end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 & x^2 & xy & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & xy & y^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & -xy & 0 & 0 & -xy \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & 0 & -x \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & -y & 0 & x & y \end{bmatrix} \begin{pmatrix} a_1\\ a_2\\ a_3\\ \vdots\\ \vdots\\ a_{13}\end{pmatrix}$$

a_1 through a_{13} = constants of stress polynomials.

The distinguishing features of this finite element are:

1) Displacement compatibility between the plane stress component of one element and the plate bending component of an adjacent element which is at an angle to the first (see Fig. below) is achieved by the elements. This compatibility requirement is usually ignored in most flat shell/plate elements.

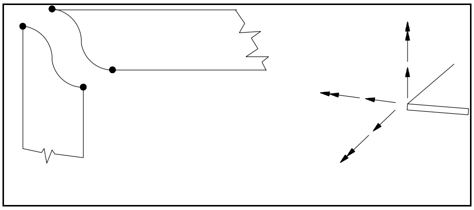


Figure 1.16

2) The out of plane rotational stiffness from the plane stress portion of each element is usefully incorporated and not treated as a dummy as is usually done in most commonly available commercial software.

3) Despite the incorporation of the rotational stiffness mentioned previously, the elements satisfy the patch test absolutely.

4) These elements are available as triangles and quadrilaterals, with corner nodes only, with each node having six degrees of freedom.

5) These elements are the simplest forms of flat shell/plate elements possible with corner nodes only and six degrees of freedom per node. Yet solutions to sample problems converge rapidly to accurate answers even with a large mesh size.

6) These elements may be connected to plane/space frame members with full displacement compatibility. No additional restraints/releases are required.

7) Out of plane shear strain energy is incorporated in the formulation of the plate bending component. As a result, the elements respond to Poisson boundary conditions which are considered to be more accurate than the customary Kirchoff boundary conditions.

8) The plate bending portion can handle thick and thin plates, thus extending the usefulness of the plate elements into a multiplicity of problems. In addition, the thickness of the plate is taken into consideration in calculating the out of plane shear.

9) The plane stress triangle behaves almost on par with the well known linear stress triangle. The triangles of most similar flat shell elements incorporate the constant stress triangle which has slow rates of convergence. Thus the triangular shell element is useful in problems with double curvature where the quadrilateral element may not be suitable.

10) Stress retrieval at nodes and at any point within the element.

Plate Element Local Coordinate System

The orientation of local coordinates is determined as follows:

1) The vector pointing from I to J is defined to be parallel to the local x- axis.

2) The cross-product of vectors IJ and IK defines a vector parallel to the local z-axis, i.e., z = IJ x IK.

3) The cross-product of vectors z and x defines a vector parallel to the local y- axis, i.e., y = z x x.

4) The origin of the axes is at the center (average) of the 4 joint locations (3 joint locations for a triangle).

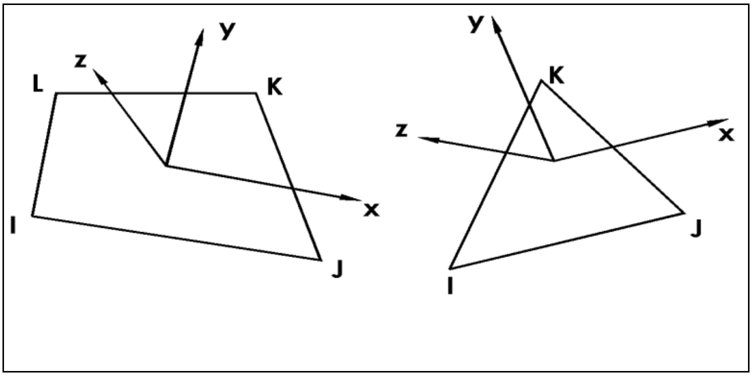


Figure 1.17

Output of Plate Element Stresses and Moments

For the sign convention of output stress and moments, please see Fig. 1.13.

ELEMENT stress and moment output is available at the following locations:

A. Center point of the element.
B. All corner nodes of the element.
C. At any user specified point within the element.

Following are the items included in the ELEMENT STRESS output.

SQX, SQY	Shear stresses (Force/ unit len./ thk.)
SX, SY, SXY	Membrane stresses (Force/unit len./ thk)
MX, MY, MXY	Moments per unit width (Force x Length/length) (For Mx, the unit width is a unit distance parallel to the local Y axis. For My, the unit width is a unit distance parallel to the local X axis. Mx and My cause bending, while Mxy causes the element to twist out-of-plane.)
SMAX, SMIN	Principal stresses in the plane of the element (Force/unit area). The 3rd principal stress is 0.0
TMAX	Maximum 2D shear stress in the plane of the element (Force/unit area)
ANGLE	Orientation of the 2D principal plane (Degrees)
VONT, VONB	3D Von Mises stress at the top and bottom surfaces, where

$$VM = 0.707\sqrt{(SMAX - SMIN)^2 + SMAX^2 + SMIN^2}$$

TRESCAT, TRESCAB Tresca stress, where

TRESCA = MAX[|(Smax-Smin)| , |(Smax)| , |(Smin)|]

Notes:

1. All element stress output is in the local coordinate system. The direction and sense of the element stresses are explained in Fig. 1.13.
2. To obtain element stresses at a specified point within the element, the user must provide the location (local X, local Y) in the coordinate system for the element. The origin of the local coordinate system coincides with the center of the element.
3. The 2 nonzero Principal stresses at the surface (SMAX & SMIN), the maximum 2D shear stress (TMAX), the 2D orientation of the principal plane (ANGLE), the 3D Von Mises stress (VONT & VONB), and the 3D Tresca stress (TRESCAT & TRESCAB) are also printed for the top and bottom surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local z-axis.
4. The third principal stress is assumed to be zero at the surfaces for use in Von Mises and Tresca stress calculations. However, the TMAX and ANGLE are based only on the 2D inplane stresses (SMAX & SMIN) at the surface. The 3D maximum shear stress at the surface is not calculated but would be equal to the 3D Tresca stress divided by 2.0.

Sign Convention of Plate Element Stresses and Moments

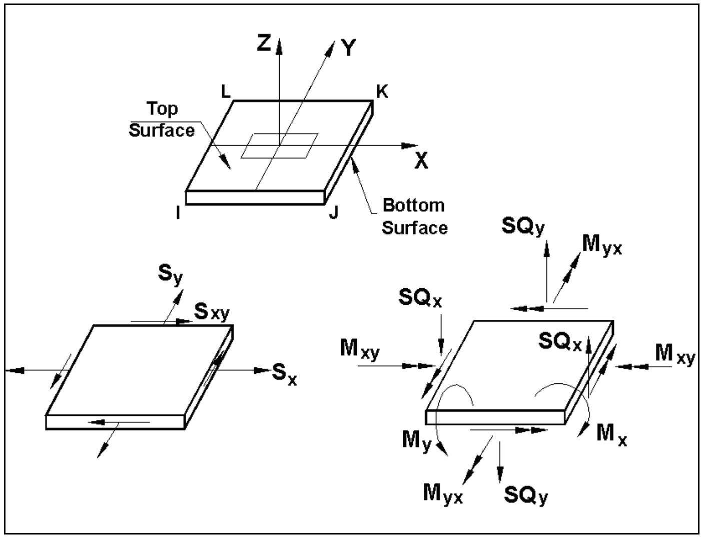


Figure 1.18

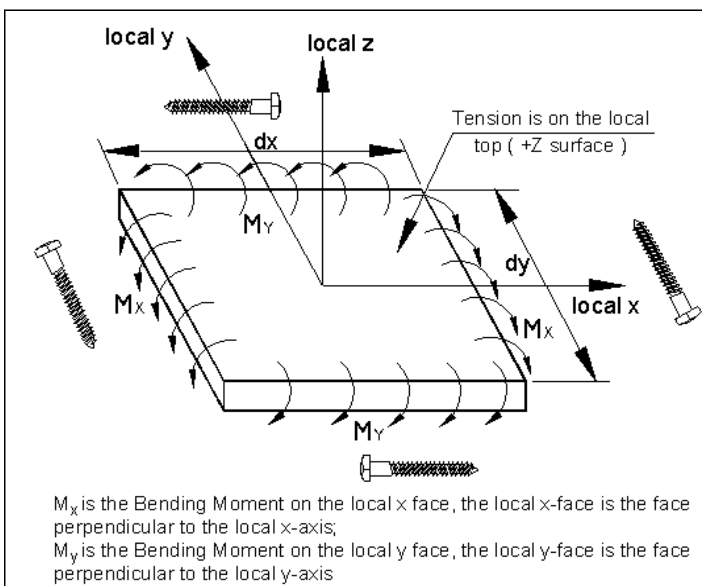


Figure 1.19

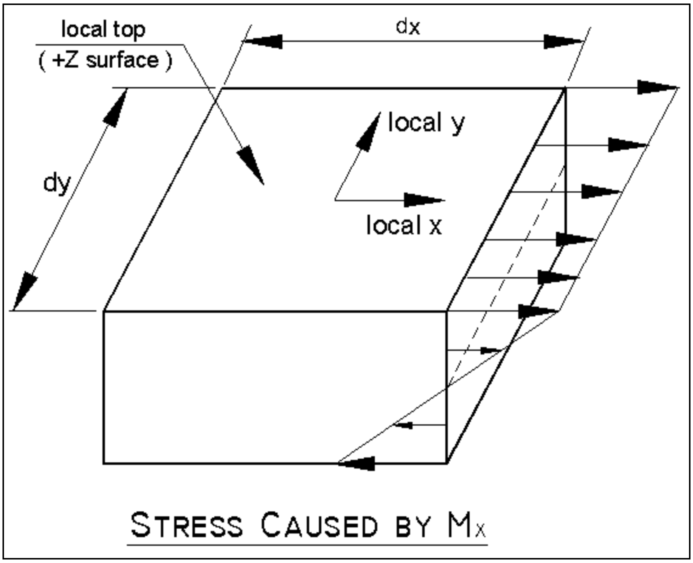


Figure 1.20

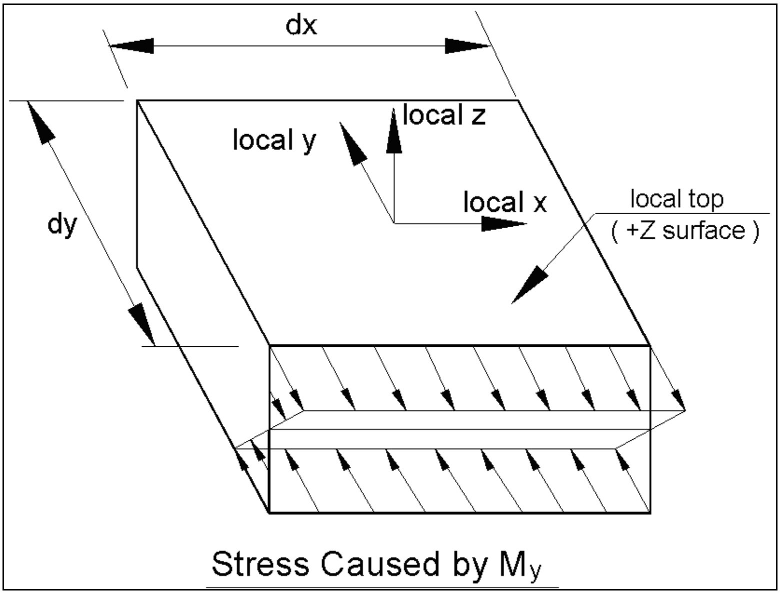


Figure 1.21

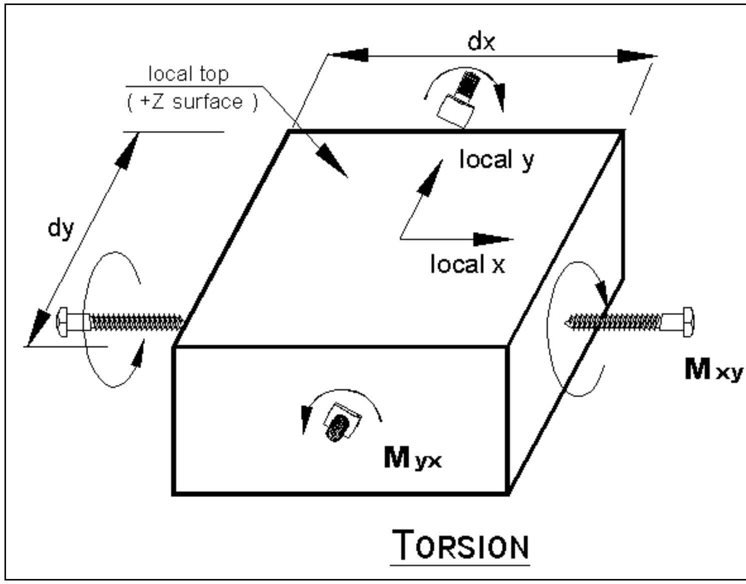


Figure 1.22

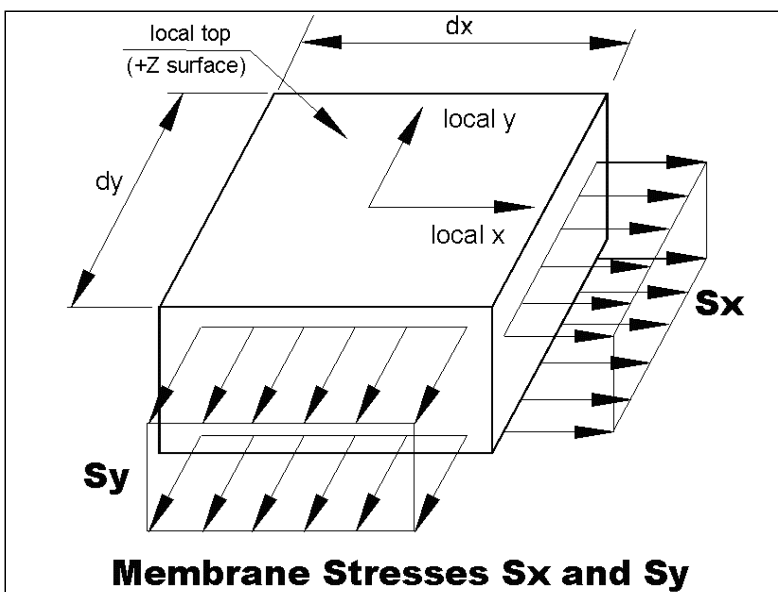


Figure 1.23

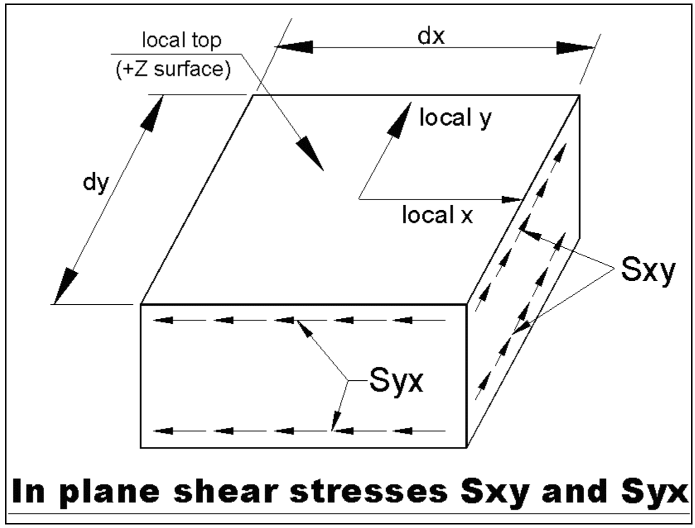


Figure 1.24

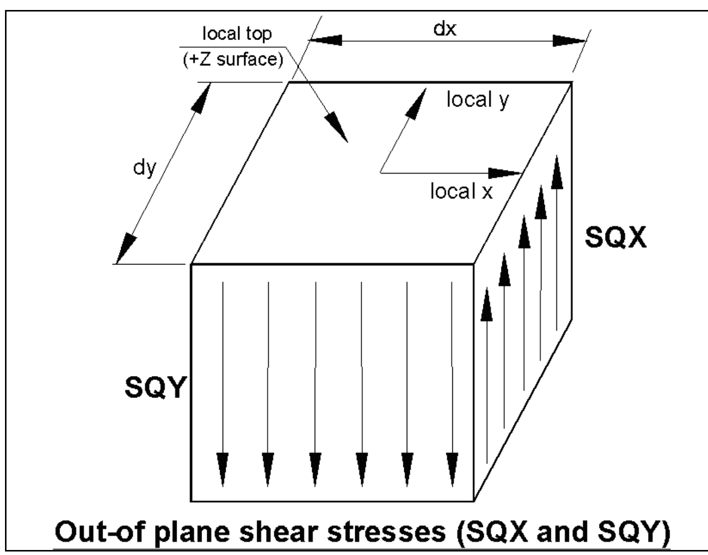


Figure 1.25

Members, plate elements, solid elements and surface elements can all be part of a single STAAD model. The MEMBER INCIDENCES input must precede the INCIDENCE input for plates, solids or surfaces. All INCIDENCES must precede other input such as properties, constants, releases, loads, etc. The selfweight of the finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.

Plate Element Numbering

During the generation of element stiffness matrix, the program verifies whether the element is same as the previous one or not. If it is same, repetitive calculations are not performed. The sequence in which the element stiffness matrix is generated is the same as the sequence in which elements are input in element incidences.

Therefore, to save some computing time, similar elements should be numbered sequentially. Fig. 1.14 shows examples of efficient and non-efficient element numbering.

However the user has to decide between adopting a numbering system which reduces the computation time versus a numbering system which increases the ease of defining the structure geometry.

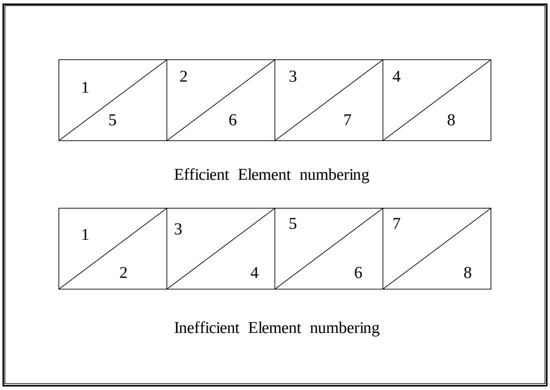


Figure 1.26

1.6.2 Solid Element

Solid elements enable the solution of structural problems involving general three dimensional stresses. There is a class of problems such as stress distribution in concrete dams, soil and rock strata where finite element analysis using solid elements provides a powerful tool.

Theoretical Basis

The solid element used in STAAD is of eight noded isoparametric type. These elements have three translational degrees-of-freedom per node.

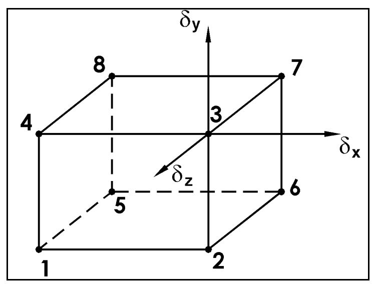


Figure 1.27

By collapsing various nodes together, an eight noded solid element can be degenerated to the following forms with four to seven nodes. Joints 1, 2, and 3 must be retained as a triangle.

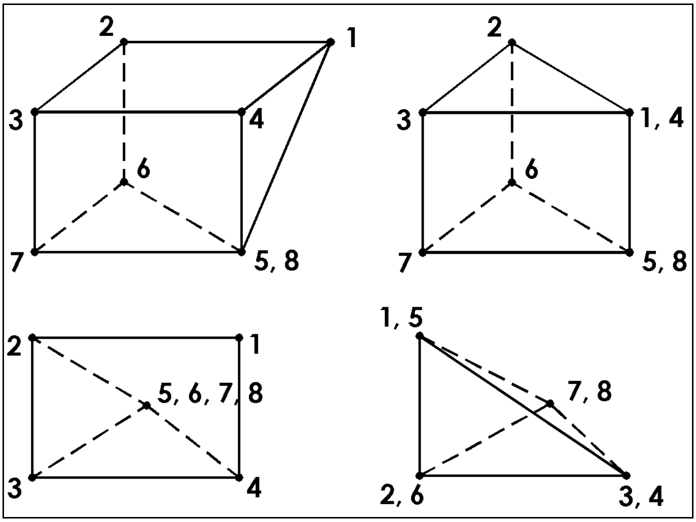


Figure 1.28

The stiffness matrix of the solid element is evaluated by numerical integration with eight Gauss-Legendre points. To facilitate the numerical integration, the geometry of the element is expressed by interpolating functions using natural coordinate system, (r,s,t) of the element with its origin at the center of gravity. The interpolating functions are shown below:

$$x = \sum_{i=1}^{8} h_i x_i , \quad y = \sum_{i=1}^{8} h_i y_i , \quad z = \sum_{i=1}^{8} h_i z_i$$

where x, y and z are the coordinates of any point in the element and x_i, y_i, z_i, i=1,..,8 are the coordinates of nodes defined in the global coordinate system. The interpolation functions, h_i are defined in the natural coordinate system, (r,s,t). Each of r, s and t varies between -1 and +1. The fundamental property of the unknown interpolation functions h_i is that their values in natural coordinate system is unity at node, i, and zero at all other nodes of the element. The element displacements are also interpreted the same way as the geometry. For completeness, the functions are given below:

$$u = \sum_{i=1}^{8} h_i u_i , \quad v = \sum_{i=1}^{8} h_i v_i , \quad w = \sum_{i=1}^{8} h_i w_i$$

where u, v and w are displacements at any point in the element and u_i, v_i, w_i, i=1,8 are corresponding nodal displacements in the coordinate system used to describe the geometry.

Three additional displacement “bubble” functions which have zero displacements at the surfaces are added in each direction for improved shear performance to form a 33x33 matrix. Static condensation is used to reduce this matrix to a 24x24 matrix at the corner joints.

Local Coordinate System

The local coordinate system used in solid elements is the same as the global system as shown below :

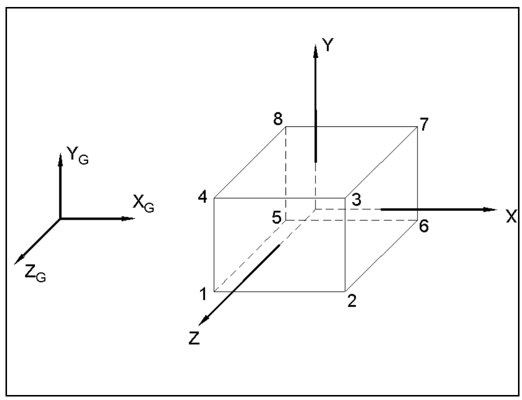


Figure 1.29

Properties and Constants

Unlike members and shell (plate) elements, no properties are required for solid elements. However, the constants such as modulus of elasticity and Poisson’s ratio are to be specified. Also, Density needs to be provided if selfweight is included in any load case.

Output of Solid Element Stresses

Element stresses may be obtained at the center and at the joints of the solid element. The items that are printed are :

Normal Stresses : SXX, SYY and SZZ
Shear Stresses : SXY, SYZ and SZX
Principal stresses : S1, S2 and S3.
Von Mises stresses:

$$\text{SIGE} = .707 \sqrt{(S1-S2)^2 + (S2-S3)^2 + (S3-S1)^2}$$

Direction cosines : 6 direction cosines are printed, following the expression DC, corresponding to the first two principal stress directions.

1.6.3 Surface Element

For any panel type of structural component, modeling requires breaking it down into a series of plate elements for analysis purposes. This is what is known in stress analysis parlance as meshing. When a user chooses to model the panel component using plate elements, he/she is taking on the responsibility of meshing. Thus, what the program sees is a series of elements. It is the user's responsibility to ensure that meshing is done properly. Examples of these are available in example problems 9, 10, 23, 27, etc. (of the Examples manual) where individual plate elements are specified.

With the new Surface type of entity, the burden of meshing is shifted from the user to the program to some degree. The entire wall or slab is hence represented by just a few "Surface" entities, instead of hundreds of elements. When the program goes through the analysis phase, it will subdivide the surface into elements by itself. The user does not have to instruct the program in what manner to carry out the meshing.

The attributes associated with surfaces, and the sections of this manual where the information may be obtained, are listed below:

Attributes	**Related Sections**
Surfaces incidences -	5.13.3
Openings in surfaces -	5.13.3
Local coordinate system for surfaces -	1.6.3
Specifying sections for stress/force output -	5.13.3
Property for surfaces -	5.21.2
Material constants -	5.26.3
Surface loading -	5.32.3.4
Stress/Force output printing -	5.42
Shear Wall Design -	3.8.2, 5.54

Local Coordinate system for surfaces

The origin and orientation of the local coordinate system of a surface element depends on the order in which the boundary nodal points are listed and position of the surface element in relation to the global coordinate system.

Let **X**, **Y**, and **Z** represent the local and **GX**, **GY**, and **GZ** the global axis vectors, respectively. The following principles apply:

a. Origin of **X-Y-Z** is located at the first node specified.

b. Direction of **Z** may be established by the right hand corkscrew rule, where the thumb indicates the positive **Z** direction, and the fingers point along the circumference of the element from the first to the last node listed.

c. **X** is a vector product of **GY** and **Z** (**X** = **GY** x **Z**). If **GY** and **Z** are parallel, **X** is taken as a vector parallel to **GX**.

d. Finally, **Y** is a vector product of **Z** and **X** (**Y** = **Z** x **X**).

The diagram below shows directions and sign convention of local axes and forces.

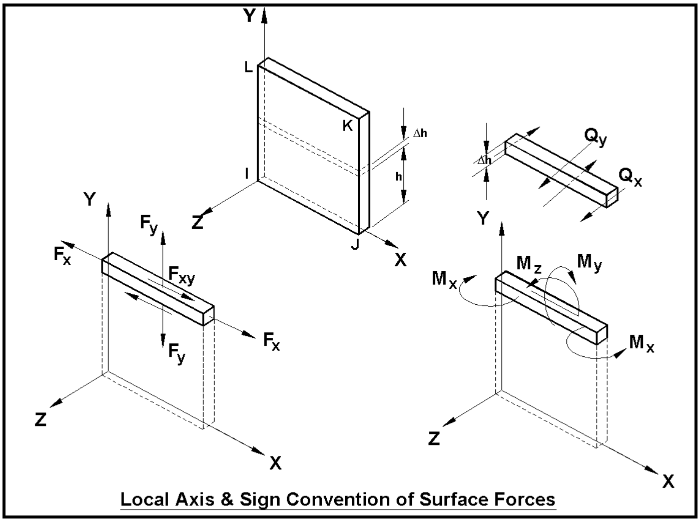


Figure 1.30

1.7 Member Properties

The following types of member property specifications are available in STAAD:

See section 5.20

A) PRISMATIC property specifications
B) Standard Steel shapes from built-in section library
C) User created steel tables
D) TAPERED sections
E) Through ASSIGN command
F) CURVED specification

Shear Area for members refers to the shear stiffness effective area. Shear stiffness effective area is used to calculate shear stiffness for the member stiffness matrix. As an example: for a rectangular cross section, the shear stiffness effective area is usually taken as 0.83 (Roark) to 0.85 (Cowper) times the cross sectional area. A shear area of less than the cross sectional area will reduce the stiffness. A typical shearing stiffness term is

$(12EI/L^3)/(1+\Phi)$ where

$\Phi = (12\ EI) / (GA_s\ L^2)$ and A_s is the shear stiffness effective area.

PHI (Φ)is usually ignored in basic beam theory. STAAD will include the PHI term unless the SET SHEAR command is entered.

Shear stress effective area is a different quantity that is used to calculate shear stress and in code checking. For a rectangular cross section, the shear stress effective area is usually taken as 2/3 rds of the cross sectional area.

Shear stress in STAAD may be from one of 3 methods.

1. (Shear Force)/(Shear stress effective area)
   This is the case where STAAD computes the area based on the cross section parameters.
2. (Shear Force)/(Shear stiffness effective area)
   This is the case where STAAD uses the shear area entered.
3. (V Q)/(I t)
   In some codes and for some cross sections, STAAD uses this method.

The values that STAAD uses for shear area for shear deformation calculation can be obtained by specifying the command

PRINT MEMBER PROPERTIES

The output for this will provide us this information in all circumstances - when AY and AZ are not provided, when AY and AZ are set to zero, when AY and AZ are set to very large numbers, when properties are specified using PRISMATIC, when properties are specified through a user table, when properties are specified through from the built-in-table, etc.

1.7.1 Prismatic Properties

See section 5.20.2

The following prismatic properties are required for analysis:

AX = Cross sectional area
IX = Torsional constant
IY = Moment of inertia about y-axis.
IZ = Moment of inertia about z-axis.

In addition, the user may choose to specify the following properties:

AY = Effective shear stiffness area for shear force parallel to local y-axis.
AZ = Effective shear stiffness area for shear force parallel to local z-axis.
YD = Depth of section parallel to local y-axis.
ZD = Depth of section parallel to local z-axis.

For T-beams, YD, ZD, YB & ZB must be specified. These terms, which are shown in the next figure are :

YD = Total depth of section (top fiber of flange to bottom fiber of web)
ZD = Width of flange
YB = Depth of stem
ZB = Width of stem

For Trapezoidal beams, YD, ZD & ZB must be specified. These terms, which too are shown in the next figure are :

YD = Total depth of section
ZD = Width of section at top fiber
ZB = Width of section at bottom fiber

Top & bottom are defined as positive side of the local Z axis, and negative side of the local Z axis respectively.

STAAD automatically considers the additional deflection of members due to pure shear (in addition to deflection due to ordinary bending theory). To ignore the shear deflection, enter a SET SHEAR command before the joint coordinates. This will bring results close to textbook results.

The depths in the two major directions (YD and ZD) are used in the program to calculate the section moduli. These are needed only to calculate member stresses or to perform concrete design. The user can omit the YD & ZD values if stresses or design of these members are of no interest. The default value is 253.75 mm (9.99 inches) for YD and ZD. All the prismatic properties are input in the local member coordinates.

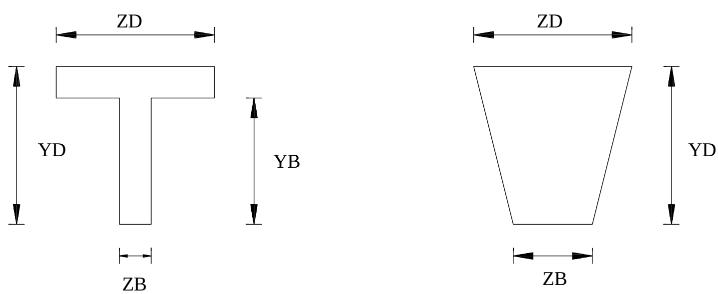


Figure 1.31

To define a concrete member, the user must not provide AX, but instead, provide YD and ZD for a rectangular section and just YD for a circular section. If no moment of inertia or shear stiffness areas are provided, the program will automatically calculate these from YD and ZD.

Table 1.1 is offered to assist the user in specifying the necessary section values. It lists, by structural type, the required section properties for any analysis. For the PLANE or FLOOR type analyses, the choice of the required moment of inertia depends upon the beta angle. If BETA equals zero, the required property is IZ.

Table 1.1 Required properties

Structural Type	Required Properties
TRUSS structure	AX
PLANE structure	AX, IZ or IY
FLOOR structure	IX, IZ or IY
SPACE structure	AX, IX, IY, IZ

1.7.2 Built-In Steel Section Library

See section 2.2.1 and 5.20.1

This feature of the program allows the user to specify section names of standard steel shapes manufactured in different countries. Information pertaining to the American steel shapes is available in section 2.

For information on steel shapes for other countries, please refer to the International Codes manual.

STAAD.Pro comes with the non-composite castellated beam tables supplied by the steel products manufacturer SMI Steel Products. Details of the manufacture and design of these sections may be found at

http://www.smisteelproducts.com/English/About/design.html

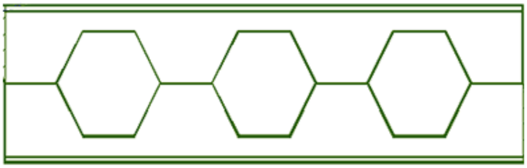


Figure 1.32

Since the shear stiffness areas of the sections are built into the tables, shear deformation is always considered for these sections.

1.7.3 User Provided Steel Table

See sections 5.19, 5.20.4 and Examples Manual problem 17

The user can provide a customized steel table with designated names and proper corresponding properties. The program can then find member properties from those tables. Member selection may also be performed with the program selecting members from the provided tables only.

These tables can be provided as a part of a STAAD input or as separately created files from which the program can read the properties. The user who does not use standard rolled shapes or who uses a limited number of specific shapes may create permanent member property files. Analysis and design can be limited to the sections in these files.

1.7.4 Tapered Sections

See section 5.20.3

Properties of tapered I-sections and several types of tapered tubes may be provided through MEMBER PROPERTY specifications. Given key section dimensions, the program is capable of calculating cross-sectional properties which are subsequently used in analysis. Specification of TAPERED sections is described in Section 5 of this manual.

1.7.5 Assign Command

See section 5.20.5

If one wishes to avoid the trouble of defining a specific section name, but instead wants to leave it to the program to assign a section name by itself, the ASSIGN command is available. The section types that may be ASSIGNed include BEAM, COLUMN, CHANNEL, ANGLE and DOUBLE ANGLE.

When the keyword BEAM is specified, the program will assign an I-shaped beam section (Wide Flange for AISC, UB section for British).

For the keyword COLUMN also, the program will assign an I-shaped beam section (Wide Flange for AISC, UC section for British).

If steel design-member selection is requested, a similar type section will be selected. See section 5.20.5 for the command syntax and description of the ASSIGN Command.

1.7.6 Steel Joist and Joist Girders

STAAD.Pro now comes with the facilities for specifying steel joists and joist girders. The basis for this implementation is the information contained in the 1994 publication of the American Steel Joist Institute called "Fortieth edition standard specifications, load tables and weight tables for steel joist and joist girders". The following are the salient features of the implementation.

Member properties can be assigned by specifying a joist designation contained in tables supplied with the program. The following joists and joist girder types have been implemented:

Open web steel joists – K series and KCS joists
Longspan steel joists – LH series
Deep Longspan steel joists – DLH series
Joist Girders – G series

The pages in the Steel Joist Institute publication where these sections are listed are shown in the following table.

Joist type	**Beginning page number**
K series	24
KCS	30
LH series	54
DLH series	57
Joist girders	74

The designation for the G series Joist Girders is as shown in page 73 of the Steel Joist Institute publication. STAAD.Pro incorporates the span length also in the name, as shown in the next figure.

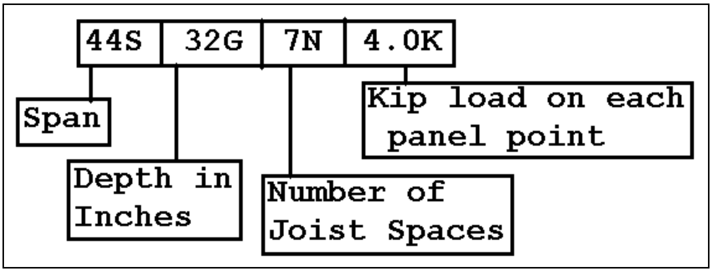


Figure 1.33

Modeling the joist - Theoretical basis

Steel joists are prefabricated, welded steel trusses used at closely spaced intervals to support floor or roof decking. Thus, from an analysis standpoint, a joist is not a single member in the same sense as beams and columns of portal frames that one is familiar with. Instead, it is a truss assembly of members. In general, individual manufacturers of the joists decide on the cross section details of the members used for the top and bottom chords, and webs of the joists. So, joist tables rarely contain any information on the cross-section properties of the individual components of a joist girder. The manufacturer’s responsibility is to guarantee that, no matter what the cross section details of the members are, the joist simply has to ensure that it provides the capacity corresponding to its rating.

The absence of the section details makes it difficult to incorporate the true truss configuration of the joist in the analysis model of the overall structure. In STAAD, selfweight and any other member load applied on the joist is transferred to its end nodes through simply supported action. Also, in STAAD, the joist makes no contribution to the stiffness of the overall structure.

As a result of the above assumption, the following points must be noted with respect to modeling joists:

1) The entire joist is represented in the STAAD input file by a single member. Graphically it will be drawn using a single line.
2) After creating the member, the properties should be assigned from the joist database.
3) The 3D Rendering feature of the program will display those members using a representative Warren type truss.
4) The intermediate span-point displacements of the joist cannot be determined.

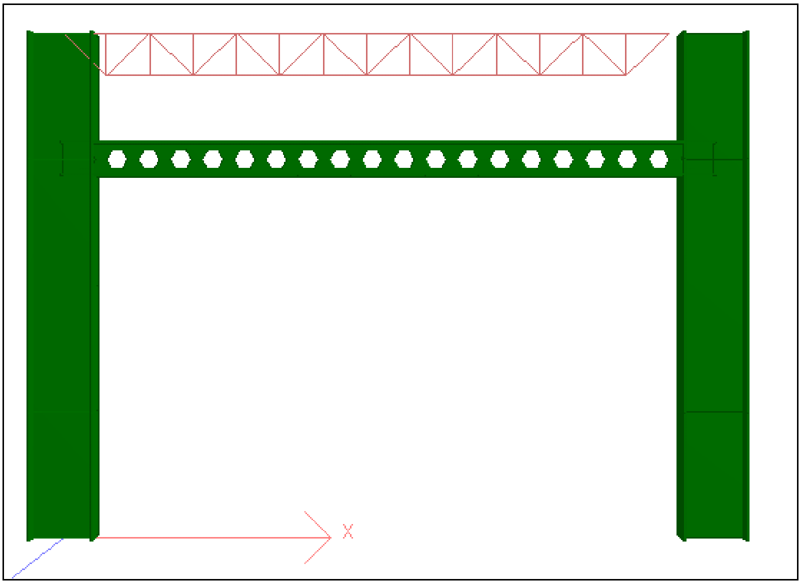


Figure 1.34

Assigning the joists

The procedure for assigning the joists is explained in the Graphical User Interface manual.

The STAAD joists database includes the weight per length of the joists. So, for selfweight computations in the model, the weight of the joist is automatically considered.

An example of a structure with joist (command file input data) is shown below.

```
STAAD SPACE EXAMPLE FOR JOIST GIRDER

UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 0 10 0
3 30 10 0; 4 30 0 0

MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;

MEMBER PROPERTY AMERICAN
1 3 TABLE ST W21x50

MEMBER PROPERTY SJIJOIST
2 TABLE ST 22K6

CONSTANTS
E STEEL ALL
DENSITY STEEL ALL
POISSON STEEL ALL

SUPPORTS
1 4 FIXED

UNIT POUND FEET
LOAD 1
SELFWEIGHT Y -1

LOAD 2
MEMBER LOAD
2 UNI GY -250
```

```
LOAD COMB 3
1 1 2 1

PERF ANALY PRINT STAT CHECK
PRINT SUPP REAC

FINISH
```

1.7.7 Composite Beams and Composite Decks

There are two methods in STAAD for specifying composite beams. Composite beams are members whose property is comprised of an I-shaped steel cross section (like an American W shape) with a concrete slab on top. The steel section and concrete slab act monolithically. The two methods are:

a) The EXPLICIT definition method – In this method, the member geometry is first defined as a line. It is then assigned a property from the steel database, with the help of the 'CM' attribute. This method is described in Section 5.20.1 of this manual. Additional parameters like CT (thickness of the slab), FC (concrete strength), CW (effective width of slab), CD (concrete density), etc., some optional and some mandatory, are also provided.

Hence, the responsibility of determining the attributes of the composite member, like concrete slab width, lies upon the user. If the user wishes to obtain a design, additional terms like rib height, rib width, etc. also have to be separately assigned with the aid of design parameters. Hence, some amount of effort is involved in gathering all the data and assigning them.

b) The composite deck generation method – The laboriousness of the previous procedure can be alleviated to some extent by using the program's composite deck definition facilities. The program then internally converts the deck into individual composite members (calculating attributes like effective width in the process) during the analysis and design phase. The deck is defined best using the graphical tools of the program since a database of deck data from different manufacturers is accessible from easy-to-use dialog boxes. Since all the members which make up the deck are identified as part of a single object, load assignment and alterations to the deck can be done to just the deck object, and not the individual members of the deck.

The graphical procedure for creating the deck can be found in section AD.2004.22.2 of the Software Release Report for STAAD.Pro 2004's second edition. The command input is described in section 5.20.7 of this manual.

1.7.8 Curved Members

See section 5.20.8

Members can be defined as being curved. Tapered sections are not permitted. The cross section should be uniform throughout the length.

1.8 Member/Element Release

STAAD allows releases for members and plate elements.

See section 5.22

One or both ends of a member or element can be released. Members/Elements are assumed to be rigidly framed into joints in accordance with the structural type specified. When this full rigidity is not applicable, individual force components at either end of the member can be set to zero with member release statements. By specifying release components, individual degrees of freedom are removed from the analysis. Release components are given in the local coordinate system for each member. PARTIAL moment release is also allowed.

Only one of the attributes described in sections 1.8 and 1.9 can be assigned to a given member. The last one entered will be used. In other words, a MEMBER RELEASE should not be applied on a member which is declared TRUSS, TENSION ONLY or COMPRESSION ONLY.

1.9 Truss/Tension/Compression - Only Members

See section 5.23

For analyses which involve members that carry axial loads only, i.e. truss members, there are two methods for specifying this condition. When all the members in the structure are truss members, the type of structure is declared as TRUSS whereas, when only some of the members are truss members (e.g. bracings of a building), the MEMBER TRUSS command can be used where those members will be identified separately.

In STAAD, the MEMBER TENSION or MEMBER COMPRESSION command can be used to limit the axial load type the member may carry. Refer to Section 5.23.3 for details on this facility.

1.10 Tension/Compression - Only Springs

See section 5.23

In STAAD, the SPRING TENSION or SPRING COMPRESSION command can be used to limit the load direction the support spring may carry. The analysis will be performed accordingly. Refer to Section 5.23 for details on this facility.

1.11 Cable Members

STAAD supports 2 types of analysis for cable members - linear and non-linear.

1.11.1 Linearized Cable Members

See section 5.23, 5.37 & 1.18.2.5

Cable members may be specified by using the MEMBER CABLE command. While specifying cable members, the initial tension in the cable must be provided. The following paragraph explains how cable stiffness is calculated.

The increase in length of a loaded cable is a combination of two effects. The first component is the elastic stretch, and is governed by the familiar spring relationship:

$$F = Kx \text{ where } K_{elastic} = \frac{EA}{L}$$

The second component of the lengthening is due to a change in geometry (as a cable is pulled taut, sag is reduced). This relationship can be described by

$$F = Kx \text{ but here } K_{sag} = \frac{12T^3}{w^2L^3} \; (\, 1.0 \,/\, \cos^2 \alpha \,)$$

where
w = weight per unit length of cable
T = tension in cable
α = angle that the axis of the cable makes with a horizontal plane (= 0, cable is horizontal; = 90, cable is vertical).

Therefore, the "stiffness" of a cable depends on the initial installed tension (or sag). These two effects may be combined as follows:

$$K_{comb} = \frac{1}{1/K_{sag} + 1/K_{elastic}}$$

$$K_{comb} = (EA/L) / [1+w^2L^2EA(\cos^2\alpha)/12T^3]$$

Note: When T = infinity, K_{comb} = EA/L
When T = 0, K_{comb} = 0

It may be noticed that as the tension increases (sag decreases) the combined stiffness approaches that of the pure elastic situation.

The following points need to be considered when using the linear cable member in STAAD :

1) The linear cable member is only a truss member whose properties accommodate the sag factor and initial tension. The behavior of this cable member is identical to that of the truss member. It can carry axial loads only. As a result, the fundamental rules involved in modeling truss members have to be followed when modeling cable members. For example, when two cable members meet at a common joint, if there isn't a support or a 3rd member connected to that joint, it is a point of potential instability.

2) Due to the reasons specified in 1) above, applying a transverse load on a cable member is not advisable. The load will be converted to two concentrated loads at the 2 ends of the cable and the true deflection pattern of the cable will never be realized.

3) A tension only cable member offers no resistance to a compressive force applied at its ends. When the end joints of the member are subjected to a compressive force, they "give in" thereby causing the cable to sag. Under these circumstances, the cable member has zero stiffness and this situation has to be accounted for in the stiffness matrix and the displacements have to be recalculated. But in STAAD, merely declaring the member to be a cable member does not guarantee that this behavior will be accounted for. It is also important

that the user declare the member to be a tension only member by using the MEMBER TENSION command, after the CABLE command. This will ensure that the program will test the nature of the force in the member after the analysis and if it is compressive, the member is switched off and the stiffness matrix re-calculated.

4) Due to potential instability problems explained in item 1 above, users should also avoid modeling a catenary by breaking it down into a number of straight line segments. The linear cable member in STAAD cannot be used to simulate the behavior of a catenary. By catenary, we are referring to those structural components which have a curved profile and develop axial forces due to their self weight. This behavior is in reality a non-linear behavior where the axial force is caused because of either a change in the profile of the member or induced by large displacements, neither of which are valid assumptions in an elastic analysis. A typical example of a catenary is the main U shaped cable used in suspension bridges.

5) The increase of stiffness of the cable as the tension in it increases under applied loading is updated after each iteration if the cable members are also declared to be MEMBER TENSION. However, iteration stops when all tension members are in tension or slack; not when the cable tension converges.

1.11.2 Non Linear Cable & Truss Members

Cable members for the Non Linear Cable Analysis may be specified by using the MEMBER CABLE command. While specifying cable members, the initial tension in the cable or the unstressed length of the cable may be provided. The user should ensure that all cables will be in sufficient tension for all load cases to converge. Use selfweight in every load case and temperature if appropriate; i.e. don't enter component cases (e.g. wind only).

See section 5.23, 5.37.3 & 1.18.2.7

The nonlinear cable may have large motions and the sag is checked on every load step and every equilibrium iteration.

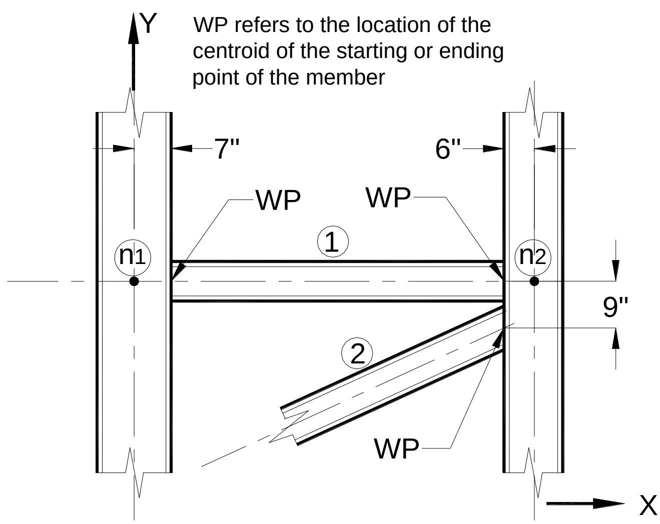
In addition, there is a nonlinear truss which is specified in the Member Truss command. The nonlinear truss is simply any truss with pretension specified. It is essentially the same as a cable without sag but also takes compression. If all cables are taut for all load cases, then the nonlinear truss may be used to simulate cables. The reason for using this substitution is that the truss solution is more reliable.

Points 1, 2, and 4 in the prior section above will not apply to nonlinear cable analysis if sufficient pretension is applied, so joints may be entered along the shape of a cable (in some cases a stabilizing stiffness may be required and entered for the first loadstep). Point 3 above: The Member Tension command is unnecessary and ignored for the nonlinear cable analysis. Point 5 above: The cable tensions are iterated to convergence in the nonlinear cable analysis.

1.12 Member Offsets

See section 5.25

Some members of a structure may not be concurrent with the incident joints thereby creating offsets. This offset distance is specified in terms of global or local coordinate system (i.e. X, Y and Z distances from the incident joint). Secondary forces induced, due to this offset connection, are taken into account in analyzing the structure and also to calculate the individual member forces. The new offset centroid of the member can be at the start or end incidences and the new working point will also be the new start or end of the member. Therefore, any reference from the start or end of that member will always be from the new offset points.



MEMBER OFFSET
1 START 7
1 END -6
2 END -6 -9

Figure 1.35

1.13 Material Constants

The material constants are: modulus of elasticity (E); weight density (DEN); Poisson's ratio (POISS); co-efficient of thermal expansion (ALPHA), Composite Damping Ratio, and beta angle (BETA) or coordinates for any reference (REF) point.

See section 5.26

E value for members must be provided or the analysis will not be performed. Weight density (DEN) is used only when selfweight of the structure is to be taken into account. Poisson's ratio (POISS) is used to calculate the shear modulus (commonly known as G) by the formula,

G = 0.5 x E/(1 + POISS)

If Poisson's ratio is not provided, STAAD will assume a value for this quantity based on the value of E. Coefficient of thermal expansion (ALPHA) is used to calculate the expansion of the members if temperature loads are applied. The temperature unit for temperature load and ALPHA has to be the same.

Composite damping ratio is used to compute the damping ratio for each mode in a dynamic solution. This is only useful if there are several materials with different damping ratios.

BETA angle and REFerence point are discussed in Sec 1.5.3 and are input as part of the member constants.

Note: Poisson’s Ratio must always be defined after the Modulus of Elasticity for a given member/element.

1.14 Supports

See section 5.27

STAAD allows specifications of supports that are parallel as well as inclined to the global axes.

Supports are specified as PINNED, FIXED, or FIXED with different releases (known as FIXED BUT). A pinned support has restraints against all translational movement and none against rotational movement. In other words, a pinned support will have reactions for all forces but will resist no moments. A fixed support has restraints against all directions of movement.

The restraints of a FIXED BUT support can be released in any desired direction as specified in section 5.27.1.

Translational and rotational springs can also be specified. The springs are represented in terms of their spring constants. A translational spring constant is defined as the force to displace a support joint one length unit in the specified global direction. Similarly, a rotational spring constant is defined as the force to rotate the support joint one degree around the specified global direction.

For static analysis, Multi-linear spring supports can be used to model the varying, non-linear resistance of a support (e.g. soil). See section 5.27.3 for descriptions of the elastic footing and elastic foundation mat facilities.

The Support command is also used to specify joints and directions where support displacements will be enforced.

1.15 Master/Slave Joints

See section 5.28

The master/slave option is provided to enable the user to model rigid links in the structural system. This facility can be used to model special structural elements like a rigid floor diaphragm. Several slave joints may be provided which will be assigned same displacements as the master joint. The user is also allowed the flexibility to choose the specific degrees of freedom for which the displacement constraints will be imposed on the slaved joints. If all degrees of freedom (Fx, Fy, Fz, Mx, My and Mz) are provided as constraints, the joints will be assumed to be rigidly connected.

1.16 Loads

Loads in a structure can be specified as joint load, member load, temperature load and fixed-end member load. STAAD can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self-weight can also be applied in any desired direction.

1.16.1 Joint Load

See section 5.32.1

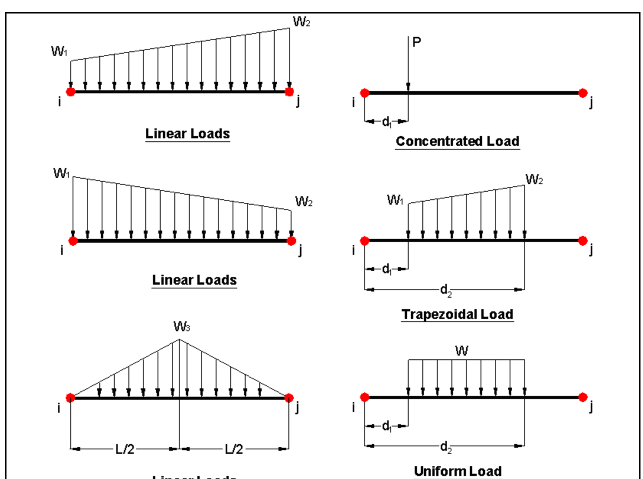
Joint loads, both forces and moments, may be applied to any free joint of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single joint, in which case the loads will be additive on that joint.

1.16.2 Member Load

See section 5.32.2

Three types of member loads may be applied directly to a member of a structure. These loads are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). Uniform loads act on the full or partial length of a member. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a member. Trapezoidal linearly varying loads act over the full or partial length of a member. Trapezoidal loads are converted into a uniform load and several concentrated loads.

Any number of loads may be specified to act upon a member in any independent loading condition. Member loads can be specified in the member coordinate system or the global coordinate system. Uniformly distributed member loads provided in the global coordinate system may be specified to act along the full or projected member length. Refer to Fig. 1.3 to find the relation of the member to the global coordinate systems for specifying member loads. Positive forces act in the positive coordinate directions, local or global, as the case may be.



Member Load Configurations - Figure 1.36

1.16.3 Area Load / Oneway Load / Floor Load

See section 5.32.4

Often a floor is subjected to a uniform pressure. It could require a lot of work to calculate the equivalent member load for individual members in that floor. However, with the AREA, ONEWAY or FLOOR LOAD facilities, the user can specify the pressure (load per unit square area). The program will calculate the tributary area for these members and calculate the appropriate member loads. The Area Load and Oneway load are used for one way distribution and the Floor Load is used for two way distribution.

The following assumptions are made while transferring the area/floor load to member load:

a) The member load is assumed to be a linearly varying load for which the start and the end values may be of different magnitude.

b) Tributary area of a member with an area load is calculated based on half the spacing to the nearest approximately parallel members on both sides. If the spacing is more than or equal to the length of the member, the area load will be ignored.

c) Area/Floor load should not be specified on members declared as MEMBER CABLE, MEMBER TRUSS, MEMBER TENSION, MEMBER COMPRESSION or CURVED.

Figure 1.37 shows a floor structure with area load specification of 0.1.

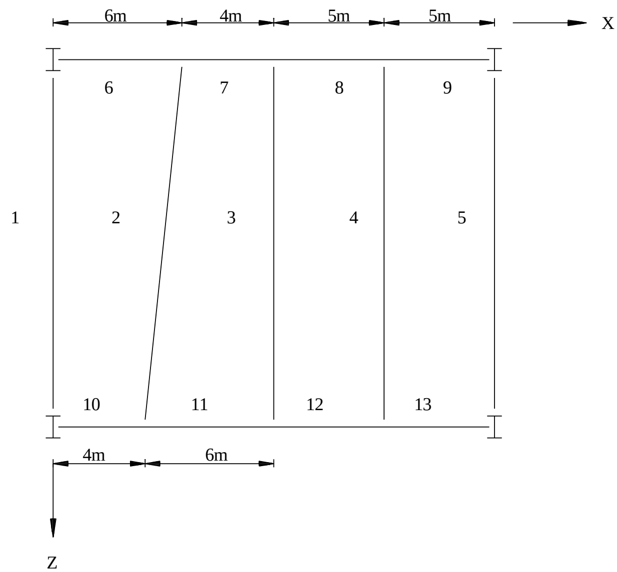


Figure 1.37

Member 1 will have a linear load of 0.3 at one end and 0.2 at the other end. Members 2 and 4 will have a uniform load of 0.5 over the full length. Member 3 will have a linear load of 0.45 and 0.55 at respective ends. Member 5 will have a uniform load of 0.25. The rest of the members, 6 through 13, will have no contributory area load since the nearest parallel members are more than each of the member lengths apart. However, the reactions from the members to the girder will be considered.

Only member loads are generated from the Area, Oneway and Floor load input. Thus, load types specific to plates, solids or surface are not generated. That is because, the basic assumption is that, a floor load or area load is used in situations where the basic entity (plate, solid or surface) which acts as the medium for application of that load, is not part of the structural model.

1.16.4 Fixed End Member Load

See section 5.32.7

Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member. Each end of a member can have six forces: axial; shear y; shear z; torsion; moment y, and moment z.

1.16.5 Prestress and Poststress Member Load

See section 5.32.5

Members in a structure may be subjected to prestress load for which the load distribution in the structure may be investigated. The prestressing load in a member may be applied axially or eccentrically. The eccentricities can be provided at the start joint, at the middle, and at the end joint. These eccentricities are only in the local y-axis. A positive eccentricity will be in the positive local y-direction. Since eccentricities are only provided in the local y-axis, care should be taken when providing prismatic properties or in specifying the correct BETA angle when rotating the member coordinates, if necessary. Two types of prestress load specification are available; PRESTRESS, where the effects of the load are transmitted to the rest of the structure, and POSTSTRESS, where the effects of the load are experienced exclusively by the members on which it is applied.

1) The cable is assumed to have a generalized parabolic profile. The equation of the parabola is assumed to be

$$y = ax^2 + bx + c$$

where $a = \frac{1}{L^2}(2es - 4em + 2ee)$

$$b = \frac{1}{L}(4em - ee - 3es)$$

$$c = es$$

where es = eccentricity of cable at start of member (in local y-axis)

em = eccentricity of cable at middle of member (in local y-axis)
ee = eccentricity of cable at end of member (in local y-axis)
L = Length of member

2) The angle of inclination of the cable with respect to the local x-axis (a straight line joining the start and end joints of the member) at the start and end points is small which gives rise to the assumption that

$$\sin\theta = \theta = dy / dx$$

Hence, if the axial force in the cable is P, the vertical component of the force at the ends is $P(dy/dx)$ and the horizontal component of the cable force is,

$$P\sqrt{1-\left(\frac{dy}{dx}\right)^2}$$

Users are advised to ensure that their cable profile meets this requirement. An angle under 5 degrees is recommended.

3) The member is analyzed for the prestressing/poststressing effects using the equivalent load method. This method is well documented in most reputed books on Analysis and Design of Prestressed concrete. The magnitude of the uniformly distributed load is calculated as

$$udl = \frac{8Pe}{L^2}$$

where P = axial force in the cable

$$e = \frac{(es+ee)}{2} - em$$

L = length of the member

4) The force in the cable is assumed to be same throughout the member length. No reduction is made in the cable forces to account for friction or other losses.

5) The term MEMBER PRESTRESS as used in STAAD signifies the following condition. The structure is constructed first. Then, the prestressing force is applied on the relevant members. As a result, the members deform and depending on their end conditions, forces are transmitted to other members in the structure. In other words, "PRE" refers to the time of placement of the member in the structure relative to the time of stressing.

6) The term MEMBER POSTSTRESS as used in STAAD signifies the following condition. The members on which such load is applied are first cast in the factory. Following this, the prestressing force is applied on them. Meanwhile, the rest of the structure is constructed at the construction site. Then, the prestressed members are brought and placed in position on the partially built structure. Due to this sequence, the effects of prestressing are "experienced" by only the prestressed members and not transmitted to the rest of the structure. In other words, "POST" refers to the time of placement of the member in the structure relative to the time of stressing.

7) As may be evident from Item (6) above, it is not possible to compute the displacements of the ends of the POSTSTRESSED members for the effects of POSTSTRESSing, and hence are assumed to be zero. As a result, displacements of intermediate sections (See SECTION DISPLACEMENT command) are measured relative to the straight line joining the start and end joints of the members as defined by their initial JOINT COORDINATES.

1.16.6 Temperature/Strain Load

See section 5.32.6

Uniform temperature difference throughout members and elements may be specified. Temperature differences across both faces of members and through the thickness of plates may also be specified (uniform temperature only for solids).. The program calculates the axial strain (elongation and shrinkage) due to the temperature difference for members. From this it calculates the induced forces in the member and the analysis is done accordingly. The strain intervals of elongation and shrinkage can be input directly.

1.16.7 Support Displacement Load

See section 5.32.8

Static Loads can be applied to the structure in terms of the displacement of the supports. Displacement can be translational or rotational. Translational displacements are provided in the specified length while the rotational displacements are always in degrees. Displacements can be specified only in directions in which the support has an “enforced” specification in the Support command.

1.16.8 Loading on Elements

On **Plate/Shell** elements, the types of loading that are permissible are:

1) Pressure loading which consists of loads which act perpendicular to the surface of the element. The pressure loads can be of uniform intensity or trapezoidally varying intensity over a small portion or over the entire surface of the element.

2) Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.

3) Temperature loads which may be constant throughout the plate element (causing only elongation / shortening) or may vary across the depth of a plate element causing bending on the plate element. The coefficient of thermal expansion for the material

of the element must be provided in order to facilitate computation of these effects.

4) The self-weight of the elements can be applied using the SELFWEIGHT loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.

On **Solid elements**, the loading types available are

1. The self-weight of the solid elements can be applied using the SELFWEIGHT loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.

2. Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.

3. Temperature loads which may be constant throughout the solid elements (causing only elongation / shortening). The coefficient of thermal expansion for the material of the element must be provided in order to facilitate computation of these effects.

4. Pressure on the faces of solids.

Only translational stiffness is supported in solid elements. Thus, at joints where there are only solid elements, moments may not be applied. For efficiency, rotational supports should be used at these joints.

1.17 Load Generator

Load generation is the process of taking a load causing unit such as wind pressure, ground movement or a truck on a bridge, and converting it to a form such as member load or a joint load which can be then be used in the analysis.

For seismic loads, a static analysis method or a dynamic analysis method can be adopted. The static analysis method, which is the one referred to here, is based on codes such as UBC, IBC, AIJ, IS1893 etc. For dynamic analysis, see the sections in this chapter on response spectrum and time history analysis.

Input for the load generation facility consists of two parts:

1) Definition of the load system(s).
2) Generation of primary load cases using previously defined load system(s).

The following sections describe the salient features of the moving load generator, the seismic load generator and the wind load generator available.

1.17.1 Moving Load Generator

See sections 5.31.1 and 5.32.12

This feature enables the user to generate static loads on members due to vehicles moving on a structure. Moving load system(s) consisting of concentrated loads at fixed specified distances in both directions on a plane can be defined by the user. A user specified number of primary load cases will be subsequently generated by the program and taken into consideration in analysis. American Association of State Highway and Transportation Officials (AASHTO) vehicles are available within the program and can be specified using standard AASHTO designations.

1.17.2 Seismic Load Generator based on UBC, IBC and other codes

See sections 5.31.2 and 5.32.12

The STAAD seismic load generator follows the procedure of equivalent lateral load analysis explained in UBC, IBC and several other codes. It is assumed that the lateral loads will be exerted in horizontal X and Z(or Y if Z up) directions and the global vertical direction, Y or Z. will be the direction of the gravity loads. Thus, for a building model, the vertical axis will be perpendicular to the floors and point upward (all vertical joint coordinates positive). The user is required to set up his model accordingly. Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate equation from the code. IBC 2003, IBC 2000, UBC 1997, 1994, or 1985, IS:1893, Japanese, Colombian, and many more specifications may be used.

For load generation per the codes, the user is required to provide seismic zone coefficients, importance factors, soil characteristic parameters, etc. See section 5.31.2 for the detailed input required for each code.

Instead of using the approximate code based formulas to estimate the building period in a certain direction, the program calculates the period using Raleigh quotient technique. This period is then utilized to calculate seismic coefficient C.

After the base shear is calculated from the appropriate equation, it is distributed among the various levels and roof per the specifications. The distributed base shears are subsequently applied as lateral loads on the structure. These loads may then be utilized as normal load cases for analysis and design.

1.17.3 Wind Load Generator

See sections 5.31.3 and 5.32.12

The Wind Load Generator is a utility which takes as input wind pressure and height ranges over which these pressures act and generates nodal point and member loads.

This facility is available for two types of structures.

a) Panel type or Closed structures
b) Open structures

Closed structures are ones like office buildings where non-structural entities like a glass façade, aluminum sheets, timber panels or non-load bearing walls act as an obstruction to the wind. If these entities are not included in the structural model, the load generated as a result of wind blowing against them needs to be computed. So, the steps involved in load generation for such structures are i) identify the panels – regions circumscribed by members so that a polygonal closed area is formed. The area may also be formed between the ground level along one edge and members along the other. ii) Calculate the panel area and multiply it by the wind pressure. iii) Convert the resulting force into nodal point loads.

Plates and solids are not considered in the calculation of the panel area. Openings within the panels may be modelled with the help of exposure factors. An exposure factor is associated with each joint of the panel and is a fractional number by which the area affecting a joint of the panel can be reduced or increased.

Open structures are those like transmission towers, in which the region between members is “open” allowing the wind to blow through. The procedure for load generation for open structures is i) Calculate the exposed area of the individual members of the model. ii) Multiply that exposed area by the wind pressure to arrive at the force and apply the force on individual members as a uniformly distributed load. It is assumed that all members of the structure within the specified ranges are subjected to the pressure and hence, they will all receive the load. The concept of members on the windward side shielding the members in the inside regions of the structure does not exist for open structures.

As a large structure may consist of hundreds of panels and members, a considerable amount of work in calculating the loads can be avoided by the user with the help of this facility.

1.17.4 Snow Load

STAAD.Pro is now capable of generating snow loading on a structure in accordance with the provisions of the ASCE-7-02 code. The feature is currently implemented for structures with flat or sloping roofs. Snow load generation for members of open lattice structures like electrical transmission towers is currently not part of this facility. Hence, the feature is based on panel areas, not the exposed width of individual members.

See Sections 5.31.5 and 5.32.13 of this manual for details of the implementation.

1.18 Analysis Facilities

The following PERFORM ANALYSIS facilities are available in STAAD.

1. Stiffness Analysis / Linear Static Analysis
2. Second Order Static Analysis
    - P-Delta Analysis
    - P-Delta KG Analysis
    - Direct Analysis for AISC 360-05 (Available effective 2007 Build 03)
    - Buckling Analysis (Available effective 2007 Build 01)
    - Imperfection Analysis
    - Multi Linear Spring Support

Member/Spring Tension/Compression only
Nonlinear Cable/Truss Analysis

3) Dynamic Analysis
Time History
Response Spectrum
Steady State / Harmonic

4) Pushover Analysis

Salient features of each type of analysis are discussed in the following sections. Detailed theoretical treatments of these features are available in standard structural engineering textbooks.

1.18.1 Stiffness Analysis

See section 5.37

The stiffness analysis implemented in STAAD is based on the matrix displacement method. In the matrix analysis of structures by the displacement method, the structure is first idealized into an assembly of discrete structural components (frame members or finite elements). Each component has an assumed form of displacement in a manner which satisfies the force equilibrium and displacement compatibility at the joints.

Structural systems such as slabs, plates, spread footings, etc., which transmit loads in 2 directions have to be discretized into a number of 3 or 4 noded finite elements connected to each other at their nodes. Loads may be applied in the form of distributed loads on the element surfaces or as concentrated loads at the joints. The plane stress effects as well as the plate bending effects are taken into consideration in the analysis.

Assumptions of the Analysis

For a complete analysis of the structure, the necessary matrices are generated on the basis of the following assumptions:

1) The structure is idealized into an assembly of beam, plate and solid type elements joined together at their vertices (nodes). The assemblage is loaded and reacted by concentrated loads acting at the nodes. These loads may be both forces and moments which may act in any specified direction.

2) A beam member is a longitudinal structural member having a constant, doubly symmetric or near-doubly symmetric cross section along its length. Beam members always carry axial forces. They may also be subjected to shear and bending in two arbitrary perpendicular planes, and they may also be subjected to torsion. From this point these beam members are referred to as "members" in the manual.

3) A plate element is a three or four noded planar element having variable thickness. A solid element is a 4-8 noded three dimensional element. These plate and solid elements are referred to as "elements" in the manual.

4) Internal and external loads acting on each node are in equilibrium. If torsional or bending properties are defined for any member, six degrees of freedom are considered at each node (i.e. three translational and three rotational) in the generation of relevant matrices. If the member is defined as truss member (i.e. carrying only axial forces) then only the three degrees (translational) of freedom are considered at each node.

5) Two types of coordinate systems are used in the generation of the required matrices and are referred to as local and global systems.

Local coordinate axes are assigned to each individual element and are oriented such that computing effort for element stiffness matrices are generalized and minimized. Global coordinate axes are a common datum established for all idealized elements so that element forces and displacements may be related to a common frame of reference.

Basic Equation

The complete stiffness matrix of the structure is obtained by systematically summing the contributions of the various member and element stiffness. The external loads on the structure are represented as discrete concentrated loads acting only at the nodal points of the structure.

The stiffness matrix relates these loads to the displacements of the nodes by the equation:

$A_j = a_j + S_j \times D_j$

------- or -----------

$[K]\{\delta\} = \{F\}$

This formulation includes all the joints of the structure, whether they are free to displace or are restrained by supports. Those components of joint displacements that are free to move are called degrees of freedom. The total number of degrees of freedom represent the number of unknowns in the analysis.

Method to Solve for Displacements

There are many methods to solve the unknowns from a series of simultaneous equations.

BASIC SOLVER: An approach which is particularly suited for structural analysis is called the method of decomposition. This method has been selected for use in STAAD. Since the stiffness matrices of all linearly elastic structures are always symmetric, an especially efficient form of the decomposition called Modified Cholesky's method may be applied to these problems. This method is reasonably accurate and well suited for the Gaussian elimination process in solving the simultaneous equations.

ADVANCED SOLVER (Available effective 2007 Build 01): An approach is used that is mathematically equivalent to the modified Choleski. However the order of operations, memory use, and file use is highly optimized. Run times are often 10 to 100 (even 1000) times faster.

Consideration of Bandwidth

For the BASIC SOLVER ONLY. The method of decomposition is particularly efficient when applied to a symmetrically banded matrix. For this type of matrix fewer calculations are required due to the fact that elements outside the band are all equal to zero.

STAAD takes full advantage of this bandwidth during solution, as it is important to have the least bandwidth to obtain the most efficient solution. For this purpose, STAAD offers features by which the program can internally rearrange the joint numbers to provide a better bandwidth.

For the ADVANCED SOLVER only. Internal storage order is automatically calculated to minimize time and memory.

Multiple Structures & Structural Integrity

The integrity of the structure is an important requirement that must be satisfied by all models. Users must make sure that the model developed represents one or more properly connected structures.

An "integral" structure may be defined as a system in which proper "stiffness connections" exist between the members/elements. The entire model functions as one or more integrated load resisting systems. STAAD checks structural integrity using a sophisticated algorithm and reports detection of multiple structures within the model. If you did not intend for there to be multiple structures, then you can fix it before any analysis. There are several additional model checking options within the TOOLS and GEOMETRY menus.

Modeling and Numerical Instability Problems

Instability problems can occur due to two primary reasons.

1) Modeling problem

There are a variety of modeling problems which can give rise to instability conditions. They can be classified into two groups.

a) Local instability - A local instability is a condition where the fixity conditions at the end(s) of a member are such as to cause an instability in the member about one or more degrees of freedom. Examples of local instability are:

   (i) Member Release: Members released at both ends for any of the following degrees of freedom (FX, FY, FZ and MX) will be subjected to this problem.
   (ii) A framed structure with columns and beams where the columns are defined as "TRUSS" members. Such a column has no capacity to transfer shears or moments from the superstructure to the supports.

b) Global Instability - These are caused when the supports of the structure are such that they cannot offer any resistance to sliding or overturning of the structure in one or more directions. For example, a 2D structure (frame in the XY plane) which is defined as a SPACE FRAME with pinned supports and subjected to a force in the Z direction will topple over about the X-axis. Another example is that of a space frame with all the supports released for FX, FY or FZ.

2) Math precision

A math precision error is caused when numerical instabilities occur in the matrix decomposition (inversion) process. One of the terms of the equilibrium equation takes the form 1/(1-A), where A=k1/(k1+k2); k1 and k2 being the stiffness coefficients of two adjacent members. When a very "stiff" member is adjacent to a very "flexible" member, viz., when k1>>k2, or k1+k2 $\cong$ k1, A=1 and hence, 1/(1-A) =1/0. Thus, huge variations in stiffnesses of adjacent members are not permitted. Artificially high E or I values should be reduced when this occurs.

Math precision errors are also caused when the units of length and force are not defined correctly for member lengths, member properties, constants etc.

Users also have to ensure that the model defined represents one or more structures that are each properly connected and supported. For example, in an effort to model an expansion joint, the user may end up defining separate structures within the same input file. Undesired multiple structures defined in one input file can lead to grossly erroneous results.

1.18.2 Second Order Analysis

STAAD offers the capability to perform second order stability analyses.

See section 5.37

1.18.2.1 P-Delta Analysis - Overview

Structures subjected to lateral loads often experience secondary forces due to the movement of the point of application of vertical loads. This secondary effect, commonly known as the P-Delta effect, plays an important role in the analysis of the structure.

In textbooks this secondary effect is referred to as stress stiffening for members in tension (or softening for compression). The stiffness changes due to P-Delta are known as geometric stiffness, [Kg]. There are two types of P-Delta effects for members. P-Δ which is due to the displacement of one end of a member relative to the other end (e.g. story drift of column members). A second effect is P-δ which is due to the bending of the member.

STAAD does not include the effects of geometric stiffness for solids. If the part of the structure that deforms involves non-trivial motions of solids, then the results will be erroneous for P-Delta analysis (as well as for buckling analysis).

1.18.2.1.1 P-Delta Analysis – Large Delta and Small Delta (Available effective 2007 Build 01)

See section 5.37

In STAAD, a procedure has been adopted to incorporate the P-Delta effect into the analysis without re-forming and factorizing the global stiffness matrix on each iteration. Actually only the global stiffness matrix is formed and factorized; which must be done for any analysis. Only the relatively fast forward and backward substitution step for typically 5 to 25 iterations must be performed. This step is done simultaneously for however many cases are being solved. See section 1.18.2.1.2 for an alternate formulation of P-Delta that may be used in dynamics.

If a structure is heavily loaded it may become unstable for some load cases. It may take 10 to 30 iterations for this instability to become obvious by the maximum displacements or bending moment envelope values becoming very large or infinite or reported as NaN.

The procedure consists of the following steps:

1) First, the primary deflections are calculated based on the provided external loading.

2) Primary deflections are used to calculate member axial forces and plate center membrane stresses. By default the large delta effects are calculated. To include the small delta effects as well, enter the “SMALLDELTA” option on the PDELTA command. These forces and stresses are used to calculate geometric stiffness terms. These terms times the displacement results from the prior iteration create the P-Delta secondary loading. This secondary loading is then combined with the originally applied loading to create the effective load vector for the next iteration.

The lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility (see Section 5.32.11) has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

3) The revised load vector is used with the static triangular factorized matrix to generate new deflections.

4) Element/Member forces and support reactions are calculated based on the new deflections.

Repeat steps 2 to 4 for several iterations. Ten to 30 iterations are recommended. This procedure yields reasonably accurate results with small displacement problems. The user is allowed to specify the number of iterations. If the Converged option is used, then set the displacement convergence tolerance by entering a SET DISP f command before the Joint Coordinates. If the change in displacement norm from one iteration to the next is less than f then it is converged.

The P-Delta analysis is recommended by several design codes such as ACI 318, LRFD, IS456-1978, etc. in lieu of the moment magnification method for the calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements. P-Delta has the most effect in structures where there are vertical and horizontal loads in the same load case.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits large buckling displacements if the loads make the structure unstable. You may need to repeat the analysis with only 1 iteration in order to get a pre-collapse solution in order to view the large displacement areas.

1.18.2.1.2 P-Delta Kg Analysis (Available effective 2007 Build 01)

See section 5.37

In STAAD, an alternate procedure has been adopted to incorporate the P-Delta effect into the analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg].

1) First, the primary deflections are calculated by linear static analysis based on the provided external loading.

2) Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

   The lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility (see Section 5.32.11) has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

This procedure yields reasonably accurate results with small displacement problems. STAAD allows the user to specify multiple iterations of this P-Delta-KG procedure; however 1 iteration is almost always sufficient.

The P-Delta analysis is recommended by several design codes such as ACI 318, LRFD, IS456-1978, etc. in lieu of the moment magnification method for the calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits buckling. You may need to repeat the analysis with only 1 iteration or as a static case in order to get a pre-collapse solution in order to view the large displacement areas.

Buckling may also cause the analysis to fail with a negative definite matrix failure. In this case, a message is printed and the results of the case are set to zero.

1.18.2.1.3 P-Delta K+Kg Dynamic Analysis (Available effective 2007 Build 01)

See section 5.37

In STAAD, an alternate procedure has been adopted to incorporate the P-Delta effect into dynamic analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg].

This method uses the resulting [K+Kg] matrices from the last static case before the PDELTA KG command in the dynamic cases that precede the PDELTA KG command.

```
LOAD n
Static case input
LOAD n+1
Dynamic case input
PDELTA KG ANALYSIS
```

1) First, the primary deflections are calculated by linear static analysis based on the provided external loading for case n.

2) Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

The final triangular factorization for case n is then used in the dynamic case n+1 along with the masses specified in case n+1 to solve the dynamic analysis.

Lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility (see Section 5.32.11) has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

P-Delta effects are calculated for frame members and plate elements only. They are not calculated for solid elements. P-Delta is restricted to structures where members and plate elements carry the vertical load from one structure level to the next.

The maximum displacement should be reviewed for P-Delta analyses because this analysis type permits buckling. You may need to repeat the analysis with only 1 iteration or as a static case in order to get a pre-collapse solution in order to view the large displacement areas.

Buckling may also cause the analysis to fail with a negative definite matrix failure. In this case a message is printed and the results of the case are set to zero. The dynamic results should be ignored if this type of failure should occur.

1.18.2.1.4 AISC 360-05 DIRECT Analysis (Available effective 2007 Build 03)

See section 5.37

From STAAD.Pro 2007 Build 03 onwards, the ANSI/AISC 360-05 Direct Analysis procedure has been adopted to incorporate the P-Delta effect into a static analysis by combining the global stiffness matrix and the global geometric stiffness matrix [K+Kg]; plus flexural stiffness reduction; plus axial stiffness reduction; plus an additional flexure reduction if member axial compression forces are above 50% of yield; plus the addition of notional loads.

1) First, the primary deflections are calculated by linear static analysis based on the provided external loading for case n. The stiffness reductions and notional loads are included here.

1) Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are added to the global stiffness matrix K.

2) The final triangular factorization for case n is then used to calculate displacements and member forces.

   Lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility (see Section 5.32.11) has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

3) The axial force is compared to yield force to calculate τ_b (see Appendix 7 of AISC 360-05). Flexure stiffness of selected members is set to (0.8 * τ_b * EI)

4) Steps 2 to 4 are repeated until convergence or the iteration limit is reached.

1.18.2.2 Buckling Analysis (Available effective 2007 Build 01)

See section 5.37.4

In STAAD, two procedures have been adopted to incorporate the calculation of the Buckling Factor for a load case. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure.

STAAD does not include the effects of geometric stiffness for solids. If the part of the structure that deforms during buckling involves non-trivial motions of solids, then the results will be erroneous for buckling (as well as for P-Delta analysis).

1.18.2.2.1 Buckling Analysis – Basic Solver

In STAAD, a simple procedure has been adopted to incorporate the calculation of the Buckling Factor for any number of primary load cases. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure.

1) First, the primary deflections are calculated by linear static analysis based on the provided external loading.

2) Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects are calculated. These terms are the terms of the Kg matrix which are multiplied by the estimated BF (buckling factor) and then added to the global stiffness matrix K.

   Buckling Kg matrix effects are calculated for frame members and plate elements only. They are not calculated for solid elements. So buckling analysis is restricted to structures where members and plate elements carry the vertical load from one structure level to the next.

3) For compressive cases, the Kg matrix is negative definite. If the buckling factor is large enough, then [[K]+BF*[Kg]] will also be negative definite which indicates that BF times the applied loads is greater than the loading necessary to cause buckling.

4) STAAD starts an iterative procedure with a BF estimate of 1.0. If that BF causes buckling, then a new, lower BF estimate is used in the next trial. If the BF does not cause buckling, then a higher BF estimate is used. On the first iteration, if the

determinant of the K matrix is positive and lower than the determinant of the K+Kg matrix, then the loads are in the wrong direction to cause buckling; and STAAD will stop the buckling calculation for that case.

5) After a few iterations, STAAD will have the largest BF that did not cause buckling (lower bound) and the lowest BF that did cause buckling (upper bound). Then each trial will use a BF estimate that is halfway between the current upper and lower bounds for BF.

6) After the default iteration limit is reached or the user specified iteration limit, MAXSTEPS, is reached or when two consecutive BF estimates are within 0.1% of each other; then the iteration is terminated.

7) Results for this load case are based on the last lower bound BF calculated.

- Only primary load cases may be solved
- Any number of buckling cases may be solved.
- Only the first buckling mode (lowest BF) is calculated.
- The buckling shape may not be as expected even though the buckling factor is OK. To enhance the mode shape result, apply small loads in the locations and directions where you expect the large displacements.

1.18.2.2.2 Buckling Analysis – Advanced Solver

In STAAD, a second procedure has been adopted to incorporate the calculation of the Buckling Factor for one primary load case. The buckling factor is the amount by which all of the loadings in a load case must be factored to cause global buckling of the structure. This procedure is an eigenvalue calculation to get buckling factors and buckling shapes.

1) First, the primary deflections are calculated by linear static analysis based on the provided external loading.

2) Primary deflections are used to calculate member axial forces and plate center membrane stresses. These forces and stresses are used to calculate geometric stiffness terms. Both the large delta effects and the small delta effects for members are calculated. These terms are the terms of the Kg matrix.

3) An eigenvalue problem is formed. $| [K] - BF_i*[Kg] | = 0$

There will be up to 4 buckling factors (BF) and associated buckling mode shapes calculated. The buckling factor is the amount by which the static load case needs to be multiplied by to just cause buckling (Euler buckling). BF less than 1.0 means that the load causes buckling; greater than 1.0 means buckling has not occurred. If BF is negative, then the static loads are in the opposite direction of the buckling load.

Notes:

- Solid elements do not contribute to Kg in STAAD.
- Buckling shapes may be displayed in the postprocessor.
- The displacement and member/element results are not calculated for the load case times the buckling factor.
- Only one primary case can be solved for this type of buckling analysis per run.

1.18.2.3 Non Linear Analysis (available in limited form)

See section 5.37

REMOVED. Contact Technical Support for further information.

1.18.2.4 Imperfection Analysis

See sections 5.37&5.26.6

Structures subjected to vertical and lateral loads often experience secondary forces due to curvature imperfections in the columns and beams. This secondary effect is similar to the P-Delta effect. In STAAD the procedure consists of the following steps:

1. First, the deflections and the axial forces in the selected imperfect members are calculated based on the provided external loading.

2. The axial forces and the input imperfections are then used to compute an additional loading on the selected imperfect members that are in compression. These additional loads are combined with the originally applied loading.

3. The static analysis is now performed with the combined loading to obtain the final result.

1.18.2.5 Multi-Linear Analysis

When soil is to be modeled as spring supports, the varying resistance it offers to external loads can be modeled using this facility, such as when its behavior in tension differs from its behavior in compression. Stiffness-Displacement characteristics of soil can be represented by a multi-linear curve. Amplitude of this curve will represent the spring characteristic of the soil at different displacement values. The load cases in a multi-linear spring analysis must be separated by the CHANGE command and PERFORM ANALYSIS command. The SET NL command must be provided to specify the total number of primary load cases. There may not be any PDELTA, dynamic, or TENSION/ COMPRESSION member cases. The multi-linear spring command will initiate an iterative analysis which continues to convergence.

1.18.2.6 Tension / Compression Only Analysis

When some members or support springs are linear but carry only tension (or only compression), then this analysis may be used. This analysis is automatically selected if any member or spring has been given the tension or compression only characteristic. This analysis is an iterative analysis which continues to convergence. Any member/ spring that fails its criteria will be inactive (omitted) on the next iteration. Iteration continues until all such members have the proper load direction or are inactive (default iteration limit is 10).

This is a simple method that may not work in some cases because members are removed on interim iterations that are needed for stability. If instability messages appear on the 2nd and subsequent iterations that did not appear on the first cycle, then do not use the solution. If this occurs on cases where only springs are the tension/compression entities, then use multi-linear spring analysis.

There may not be any Multi-linear springs, or dynamic cases.

1.18.2.7 Non Linear Cable/Truss Analysis (available in limited form)

See sections 5.23, 5.37.3, 1.11.2

When all of the members, elements and support springs are linear except for cable and/or preloaded truss members, then this analysis type may be used. This analysis is based on applying the load in steps with equilibrium iterations to convergence at each step. The step sizes start small and gradually increase (145 steps is the default). Iteration continues at each step until the change in deformations is small before proceeding to the next step. If not converged, then the solution is stopped. The user can then change analysis parameters or modify the structure and rerun.
The user has control of the number of steps, the maximum number of iterations per step, the convergence tolerance, the artificial stabilizing stiffness, and the minimum amount of stiffness remaining after a cable sags.

This method assumes small displacement theory for all members/trusses/elements other than cables & preloaded trusses. The cables and preloaded trusses can have large displacement and moderate/large strain. Preloaded trusses may carry tension and compression while cables have a reduced E modulus if not fully taut. Pretension is the force necessary to stretch the cable/truss from its unstressed length to enable it to fit between the two end joints. Alternatively, you may enter the unstressed length for cables.

The procedure was developed for structures, loadings, and pretensioning loads that will result in sufficient tension in every cable for all loading conditions. Most design codes strongly recommend cables to be in tension to avoid the undesirable dynamic effects of a slack cable such as galloping, singing, or pounding. The engineer should specify realistic initial preloading tensions which will ensure that all cable results are in tension. To minimize the compression the SAGMIN input variable can be set to a small value such as 0.01, however that can lead to a failure to converge unless many more steps are specified and a higher equilibrium iteration limit is specified. SAGMIN values below 0.70 generally requires some adjustments of the other input parameters to get convergence.

Currently the cable is not automatically loaded by selfweight, but the user should ensure that selfweight is applied in every load case. Do not enter component load cases such as wind only; every case must be realistic. Member loads will be lumped at the ends for cables and trusses. Temperature load may also be applied to the cables and trusses. It is OK to break up the cable/truss into several members and apply forces to the intermediate joints. Y-up is assumed and required.

The member force printed for the cable is Fx and is along the chord line between the displaced positions of the end joints.

The analysis sequence is as follows:

1. Compute the unstressed length of the nonlinear members based on joint coordinates, pretension, and temperature.

2. Member/Element/Cable stiffness is formed. Cable stiffness is from EA/L and the sag formula plus a geometric stiffness based on current tension.
3. Assemble and solve the global matrix with the percentage of the total applied load used for this load step.
4. Perform equilibrium iterations to adjust the change in directions of the forces in the nonlinear cables, so that the structure is in static equilibrium in the deformed position. If force changes are too large or convergence criteria not met within default number of iterations then stop the analysis.
5. Go to step 2 and repeat with a greater percentage of the applied load. The nonlinear members will have an updated orientation with new tension and sag effects.
6. After 100% of the applied load has converged, then proceed to compute member forces, reactions, and static check. The static check is not exactly in balance due to the displacements of the applied static equivalent joint loads.

The load cases in a non linear cable analysis must be separated by the CHANGE command and PERFORM CABLE ANALYSIS command. The SET NL command must be provided to specify the total number of primary load cases. There may not be any Multi-linear springs, compression only, PDelta, NONLINEAR, or dynamic cases.

Also for cables and preloaded trusses:

1. Do not use Member Offsets.
2. Do not include the end joints in Master/Slave command.
3. Do not connect to inclined support joints.
4. Y direction must be up.
5. Do not impose displacements.
6. Do not use Support springs in the model.
7. Applied loads do not change global directions due to displacements.
8. Do not apply Prestress load, Fixed end load.
9. Do not use Load Combination command to combine cable analysis results. Use a primary case with Repeat Load instead.

1.18.3 Dynamic Analysis

Currently available dynamic analysis facilities include solution of the free vibration problem (eigenproblem), response spectrum analysis and forced vibration analysis.

1.18.3.1 Solution of the Eigenproblem

See sections 5.30, 5.32.10, 5.34

The eigenproblem is solved for structure frequencies and mode shapes considering a diagonal, lumped mass matrix, with masses possible at all active d.o.f. included. Two solution methods may be used: the subspace iteration method for all problem sizes (default for all problem sizes), and the optional determinant search method for small problems.

1.18.3.2 Mass Modeling

The natural frequencies and mode shapes of a structure are the primary parameters that affect the response of a structure under dynamic loading. The free vibration problem is solved to extract these values. Since no external forcing function is involved, the natural frequencies and mode shapes are direct functions of the stiffness and mass distribution in the structure. Results of the frequency and mode shape calculations may vary significantly depending upon the mass modeling. This variation, in turn, affects the response spectrum and forced vibration analysis results. Thus, extreme caution should be exercised in mass modeling in a dynamic analysis problem.

In STAAD, all masses that are capable of moving should be modeled as loads applied in all possible directions of movement. Even if the loading is known to be only in one direction there is usually mass motion in other directions at some or all joints and these mass directions (“loads” in weight units) must be entered to be correct. Joint moments that are entered will be considered to be weight moment of inertias (force-length^2 units).

Please enter selfweight, joint and element loadings in global directions with the same sign as much as possible so that the “masses” do not cancel each other.

Member/Element loadings may also be used to generate joint translational masses. Member end joint moments that are generated by the member loading (including concentrated moments) are discarded as irrelevant to dynamics. Enter mass moments of inertia, if needed, at the joints as joint moments. STAAD uses a diagonal mass matrix of 6 lumped mass equations per joint. The selfweight or uniformly loaded member is lumped 50% to each end joint without rotational mass moments of inertia. The other element types are integrated but roughly speaking the weight is distributed equally amongst the joints of the element.

The members/elements of finite element theory are simple mathematical representations of deformation meant to apply over a small region. The FEA procedures will converge if you subdivide the elements and rerun; then subdivide the elements that have significantly changed results and rerun; etc. until the key results are converged to the accuracy needed.

An example of a simple beam problem that needs to subdivide real members to better represent the mass distribution (and the dynamic response and the force distribution response along members) is a simple floor beam between 2 columns will put all of the mass on the column joints. In this example, a vertical ground motion will not bend the beam even if there is a concentrated force (mass) at mid span.

In addition, the dynamic results will not reflect the location of a mass within a member (i.e. the masses are lumped at the joints). This means that the motion, of a large mass in the middle of a member relative to the ends of the member, is not considered. This may affect the frequencies and mode shapes. If this is important to the solution, split the member into two. Another effect of moving the masses to the joints is that the resulting shear/moment distribution is based as if the masses were not within the member. Note also that if one end of a member is a support, then half of the that member mass is lumped at the support and will not move during the dynamic response.

1.18.3.3 Damping Modeling

Damping may be specified by entering values for each mode, or using a formula based on the first two frequencies, or by using composite modal damping. Composite modal damping permits computing the damping of a mode from the different damping ratios for different materials (steel, concrete, soil). Modes that deform mostly the steel would have steel damping ratio, whereas modes that mostly deform the soil, would have the soil damping ratio.

1.18.3.4 Response Spectrum

See section 5.32.10

This capability allows the user to analyze the structure for seismic loading. For any supplied response spectrum (either acceleration vs. period or displacement vs. period), joint displacements, member forces, and support reactions may be calculated. Modal responses may be combined using one of the square root of the sum of squares (SRSS), the complete quadratic combination (CQC), the ASCE4-98 (ASCE), the Ten Percent (TEN) or the absolute (ABS) methods to obtain the resultant responses. Results of the response spectrum analysis may be combined with the results of the static analysis to perform subsequent design. To account for reversibility of seismic activity, load combinations can be created to include either the positive or negative contribution of seismic results.

1.18.3.5 Response Time History

See Sections 5.31.4 and 5.32.10.2

STAAD is equipped with a facility to perform a response history analysis on a structure subjected to time varying forcing function loads at the joints and/or a ground motion at its base. This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. Please refer to the section above on **"mass modeling"** for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the "p" lowest modes. The equilibrium equations are written as

$$[m]\{\ddot{x}\} + [c]\{\dot{x}\} + [k]\{x\} = \{P(t)\} \quad \ldots \quad \ldots \quad (1)$$

Using the transformation

$$\{x\} = \sum_{i=1}^{p} \{\phi\}_i \, q_i \quad \ldots \quad \ldots \quad \ldots \quad \ldots \quad (2)$$

Equation 1 reduces to "p" separate uncoupled equations of the form

$$\ddot{q}_i + 2\xi_i \omega_i \dot{q}_i + \omega_i^2 q_i = R_i(t) \quad \ldots \quad \ldots \quad (3)$$

where ξ is the modal damping ratio and ω the natural frequency for the i^{th} mode.

These are solved by the Wilson- θ method which is an unconditionally stable step by step scheme. The time step for the response is entered by the user or set to a default value, if not entered. The q_is are substituted in equation 2 to obtain the displacements {x} at each time step.

Time History Analysis for a Structure Subjected to a Harmonic Loading

A Harmonic loading is one which can be described using the following equation

$$F(t) = F_0 \sin\ (\omega t + \phi)$$

In the above equation,

$F(t)$ = Value of the forcing function at any instant of time "t"
F_0 = Peak value of the forcing function
ω = Frequency of the forcing function
ϕ = Phase Angle

A plot of the above equation is shown in the figure below.

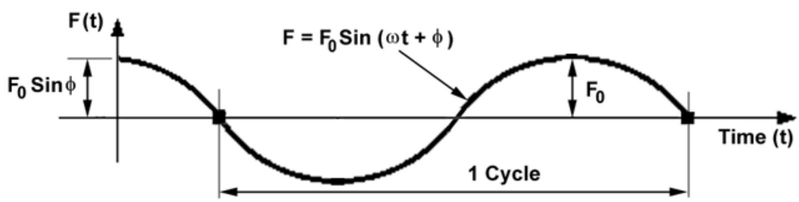


Figure 1.38

The results are the maximums over the entire time period, including start-up transients. So, they do not match steady-state response.

Definition of Input in STAAD for the above Forcing Function

As can be seen from its definition, a forcing function is a continuous function. However, in STAAD, a set of discrete time-force pairs is generated from the forcing function and an analysis is performed using these discrete time-forcing pairs. What that means is that based on the number of cycles that the user specifies for the loading, STAAD will generate a table consisting of the magnitude of the force at various points of time. The time values are chosen from this time ′0′ to n*tc in steps of "STEP" where n is the number of cycles and tc is the duration of one cycle. STEP is a value that the user may provide or may choose the default value that is built into the program. STAAD will adjust STEP so that a ¼ cycle will be evenly divided into one or more steps. Users may refer to section 5.31.4 of this manual for a list of input parameters that need to be specified for a Time History Analysis on a structure subjected to a Harmonic loading.

The relationship between variables that appear in the STAAD input and the corresponding terms in the equation shown above is explained below.

F_0 = AMPLITUDE
ω = FREQUENCY
ϕ = PHASE

1.18.3.6 Steady State and Harmonic Response

A structure [subjected only to harmonic loading, all at a given forcing frequency and with non-zero damping] will reach a steady state of vibration that will repeat every forcing cycle. This steady state response can be computed without calculating the transient time history response prior to the steady state condition.

$$R(t) = R_0 \sin\ (\omega t + \phi)$$

The result, R, has a maximum value of R_0 and a phase angle ϕ. These two values for displacement, velocity, and acceleration at each joint may be printed or displayed.

This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. Please refer to the section above on **"mass modeling"** for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the "p" lowest modes.

A Harmonic loading is one which can be described using the following equation

$$F(t) = F_0 \sin\ (\omega t + \phi)$$

In the above equation,

$F(t)$ = Value of the forcing function at any instant of time "t"
F_0 = Peak value of the forcing function
ω = Frequency of the forcing function
ϕ = Phase Angle

A plot of the above equation is shown in the figure below.

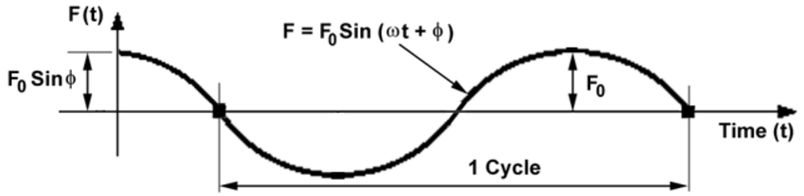


Figure 1.38

The results are the steady-state response which is the absolute maximum of displacement (and other output quantities) and the corresponding phase angle after the steady state condition has been reached.

In addition, a Harmonic response can be calculated. This response consists of a series of Steady State responses for a list of frequencies. The joint displacement, velocity, or acceleration can be displayed as the response value versus frequency. Load case results are the maximums over all of the frequencies.

All results are positive as in the Response Spectrum and Time history analyses. This means section results should be ignored (BEAM 0.0 in Parameters for code checking). Because of this, you may want to add the steady state response to Dead & Live loads for one combination case and subtract the steady state response from those loads for another combination case.

Ground motion or a joint force distribution may be specified. Each global direction may be at a different phase angle.

Output frequency points are selected automatically for modal frequencies and for a set number of frequencies between modal frequencies. There is an option to change the number of points between frequencies and an option to add frequencies to the list of output frequencies.

The load case that defines the mass distribution must be the case just before the PERFORM STEADY STATE ANALYSIS command. Immediately after that command is a set of data starting with BEGIN STEADY and ending with END STEADY. The list of additional frequencies and the steady state load cases with joint loads or ground accelerations and phasing data are entered here. The optional print command for the maximum displacement and associated phase angle for selected joints must be at the end of this block of input.

[Stardyne-Dynre2 data beginning with START2 and ending with ALL DONE may substitute for the BEGIN to END STEADY data if the STRESS data is omitted.]

Users will require a license for the advanced analysis module to access this feature.

1.18.3.7 Pushover Analysis

Pushover analysis is a static, nonlinear procedure using simplified nonlinear technique to estimate seismic structural deformations. It is an incremental static analysis used to determine the force-displacement relationship, or the capacity curve, for a structure or structural element.

In STAAD, the basis for this analysis is the information published in the documents FEMA 356 : 2000 and ATC 40.

Please contact the technical support group for a separate document containing the details of the implementation.

Users will require a license for the advanced analysis module to access this feature.

1.19 Member End Forces

See section 5.41

Member end forces and moments in the member result from loads applied to the structure. These forces are in the local member coordinate system. Figures 1.39a through 1.39d shows the member end actions with their directions.

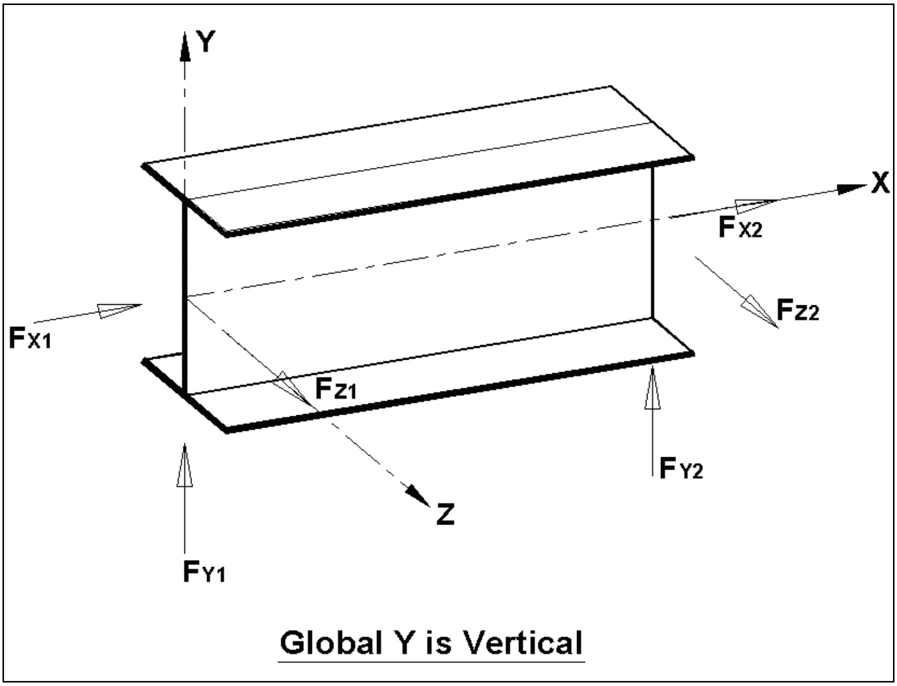


Figure 1.39a

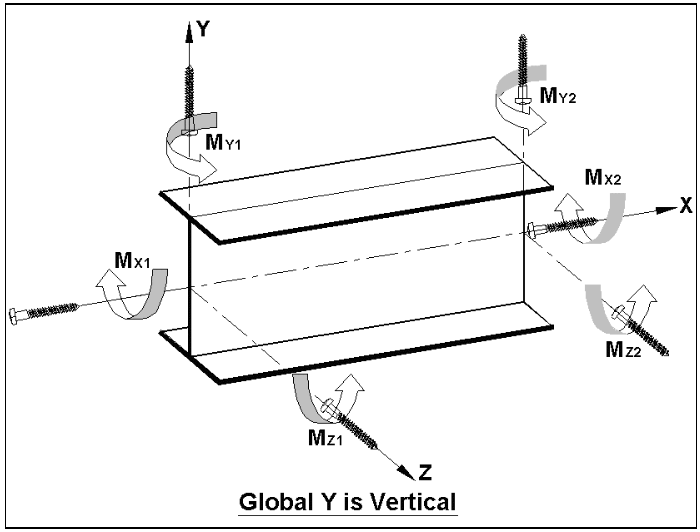


Figure 1.39b

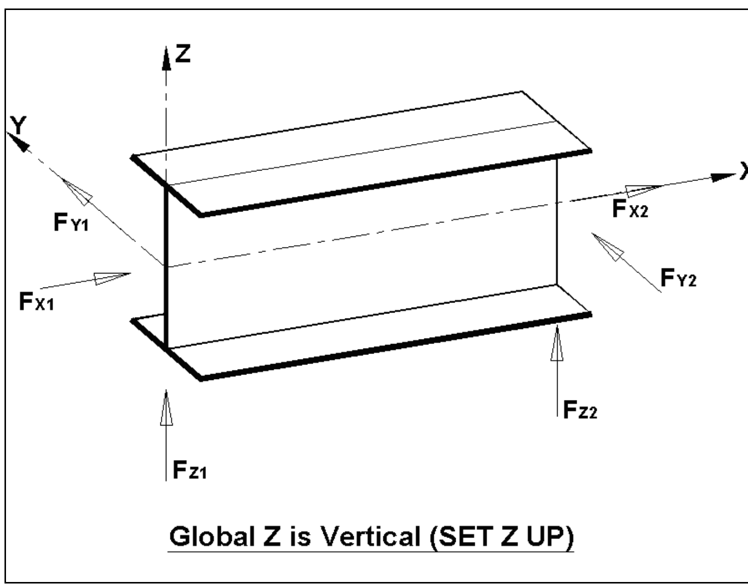


Figure 1.39c

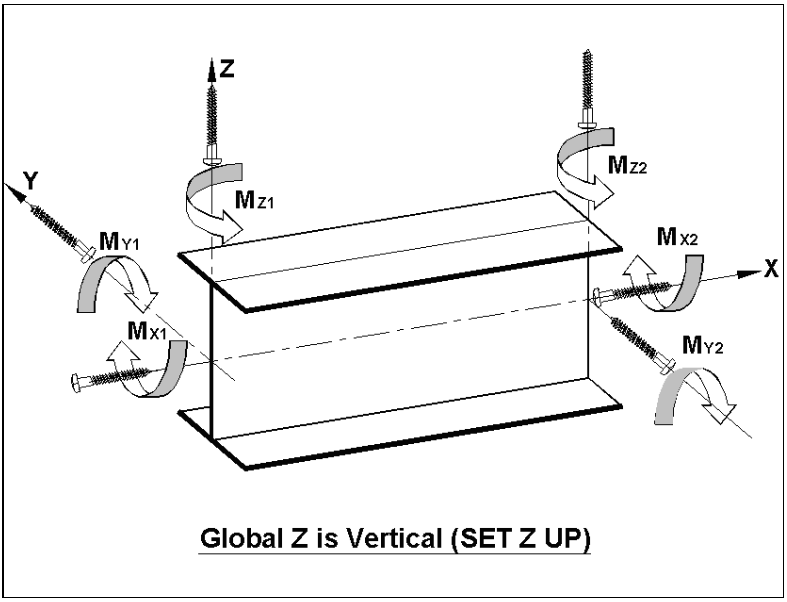


Figure 1.39d

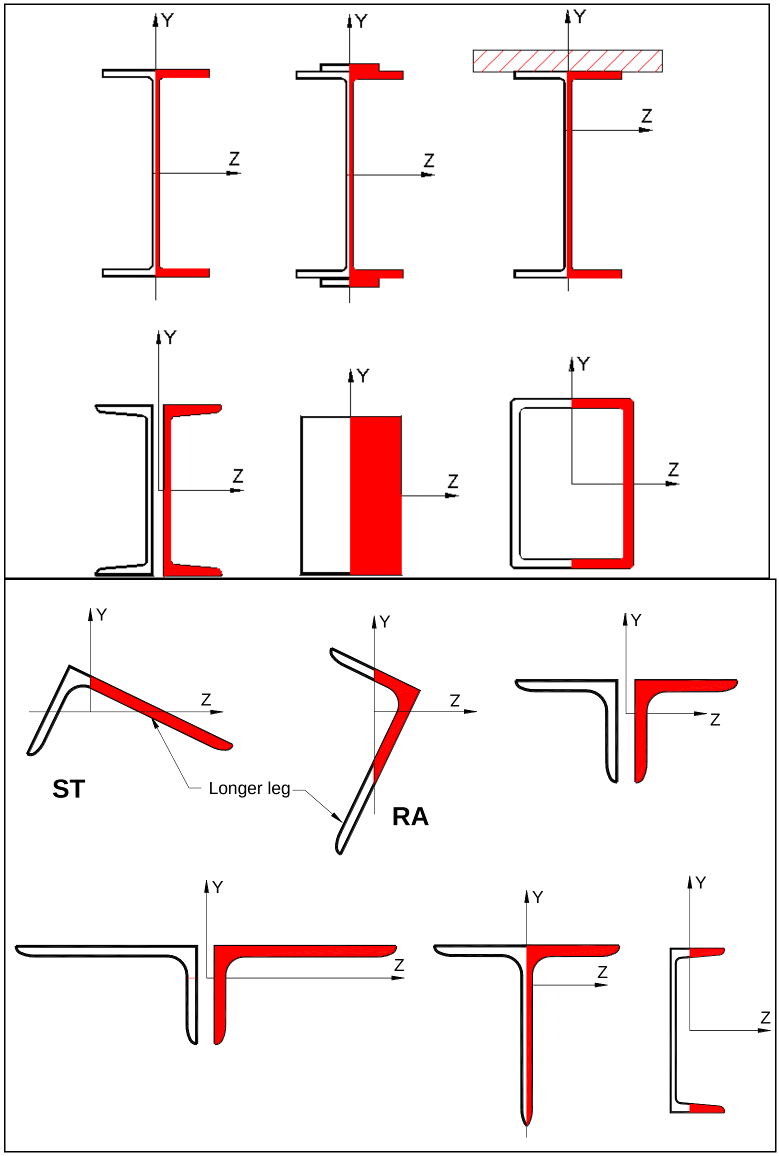


Figure 1.40 - Stress Zones due to bending about Y axis (MY)

Notes: Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension

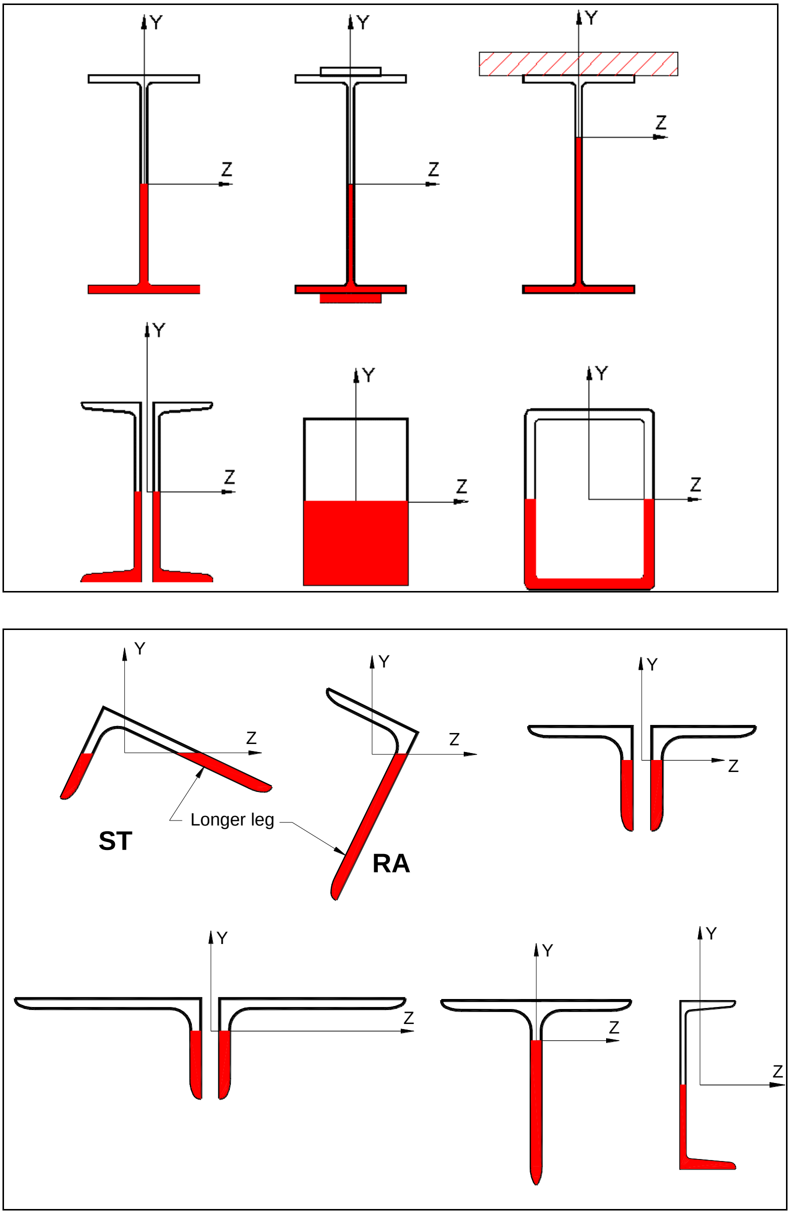


Figure 1.41 - Stress Zones due to bending about Z axis (MZ)

Notes: Local X axis goes into the page; Global Y is vertically upwards; Shaded area indicates zone under compression; Non-shaded area indicates zone under tension

1.19.1 Secondary Analysis

See sections 5.40, 5.41, 5.42 and 5.43

Solution of the stiffness equations yield displacements and forces at the joints or end points of the member. STAAD is equipped with the following secondary analysis capabilities to obtain results at intermediate points within a member.

1) Member forces at intermediate sections.
2) Member displacements at intermediate sections.
3) Member stresses at specified sections.
4) Force envelopes.

The following sections describe the secondary analysis capabilities in detail.

1.19.2 Member Forces at Intermediate Sections

See sections 5.40 and 5.41

With the SECTION command, the user may choose any intermediate section of a member where forces and moments need to be calculated. These forces and moments may also be used in design of the members. The maximum number of sections specified may not exceed five, including one at the start and one at the end of a member. If no intermediate sections are requested, the program will consider the start and end member forces for design. However, of the sections provided, they are the only ones to be considered for design.

1.19.3 Member Displacements at Intermediate Sections

See sections 5.42 and 5.45

Like forces, displacements of intermediate sections of members can be printed or plotted. This command may not be used for truss or cable members.

1.19.4 Member Stresses at Specified Sections

See sections 5.40 and 5.41

Member stresses can be printed at specified intermediate sections as well as at the start and end joints. These stresses include:

a) Axial stress, which is calculated by dividing the axial force by the cross sectional area,
b) Bending-y stress, which is calculated by dividing the moment in local-y direction by the section modulus in the same direction,
c) Bending-z stress, which is the same as above except in local-z direction,
d) Shear stresses (in y and z directions), and
e) Combined stress, which is the sum of axial, bending-y and bending-z stresses.

All the stresses are calculated as the absolute value.

1.19.5 Force Envelopes

See section 5.43

Force envelopes of the member forces FX (axial force), FY (Shear-y), and MZ (moment around local z-axis, i.e. strong axis) can be printed for any number of intermediate sections. The force values include maximum and minimum numbers representing maximum positive and maximum negative values. The following is the sign convention for the maximum and minimum values:

FX A positive value is compression, and negative tension.

FY A positive value is shear in the positive y-direction, and negative in the negative y-direction.

FZ Same as above, except in local z-direction.

MZ A positive moment will mean a moment causing tension at the top of the member. Conversely, a negative moment will cause tension at the bottom of the member. The top of a member is defined as the side towards positive local y-axis.

MY Same as above, except about local z axis.

1.20 Multiple Analyses

Structural analysis/design may require multiple analyses in the same run. STAAD allows the user to change input such as member properties, support conditions etc. in an input file to facilitate multiple analyses in the same run. Results from different analyses may be combined for design purposes.

For structures with bracing, it may be necessary to make certain members inactive for a particular load case and subsequently activate them for another. STAAD provides an INACTIVE facility for this type of analysis. The INACTIVE option is discussed in the following paragraph.

Inactive Members

See section 5.18

With the INACTIVE command, members can be made inactive. These inactive members will not be considered in the stiffness analysis or in any printout. The members made inactive by the INACTIVE command are made active again with the CHANGE command. This can be useful in an analysis where stage construction is modeled due to which, a set of members should be inactive for certain load cases. This can be accomplished by:

a) making the desired members inactive
b) providing the relevant load cases for which the members are inactive
c) performing the analysis
d) using the CHANGE command to make all the inactive members active
e) making another set of members inactive and providing the proper load cases for which the members are meant to be inactive, performing the analysis and repeating the procedure as necessary

1.21 Steel/Concrete/Timber Design

See sections 2, 3 and 4

Extensive design capabilities are available in STAAD for steel, concrete and timber sections. Detailed information on steel, concrete and timber design is presented in Sections 2, 3 and 4 respectively.

1.22 Footing Design

Removed. Contact Technical Support for further information.

1.23 Printing Facilities

All input data and output may be printed using PRINT commands available in STAAD. The input is normally echoed back in the output. However, if required, the echo can be switched off.

Extensive listing facilities are provided in almost all PRINT commands to allow the user to specify joints, members and elements for which values are required.

1.24 Plotting Facilities

Please refer to the STAAD.Pro Graphical Environment Manual for a complete description of the extensive screen and hardcopy graphical facilities available and information on using them.

1.25 Miscellaneous Facilities

STAAD offers the following miscellaneous facilities for problem solution.

Perform Rotation

See section 5.17

This command can be used to rotate the structure shape through any desired angle about any global axis. The rotated configuration can be used for further analysis and design. This command may be entered after the Joint Coordinates or between two Joint Coordinate commands or after all Member/Element Incidences are specified.

Substitute

See section 5.15

Joint and member numbers may be redefined in STAAD through the use of the SUBSTITUTE command. After a new set of numbers are assigned, input and output values will be in accordance with the new numbering scheme. This facility allows the user to specify numbering schemes that will result in simple input specification as well as easy interpretation of data.

Calculation of Center of Gravity

See section 5.41

STAAD is capable of calculating the center of gravity of the structure. The PRINT CG command may be utilized for this purpose.

1.26 Post Processing Facilities

All output from the STAAD run may be utilized for further processing by the STAAD.Pro GUI. Please refer to the STAAD.Pro Graphical Environment Manual for a complete description of the extensive screen and hardcopy graphical facilities available and for information on how to use them.

American Steel Design

Section 2

2.1 Design Operations

STAAD contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

Specify the members and the load cases to be considered in the design.

Specify whether to perform code checking or member selection.

Specify design parameter values, if different from the default values.

These operations may be repeated by the user any number of times depending upon the design requirements.

Steel Design may be performed according to several codes such as AISC-ASD (9th edition), AISC-LRFD, AISC 13th edition, AISI, AASHTO, etc. A brief description of each is presented in the following pages.

Currently, STAAD supports steel design of wide flange, S, M, HP shapes, tees, angle, double angle, channel, double channel, pipes, tubes, beams with cover plate and composite beams (I shapes with concrete slab on top.

2.2 Member Properties

For specification of member properties of standard American steel sections, the steel section library available in STAAD may be used. The syntax for specifying the names of built-in steel shapes is described in the next section.

2.2.1 Built - in Steel Section Library

The following sections describe specification of steel sections from the AISC Steel Tables.

AISC Steel Table

Almost all AISC steel shapes are available for input. Following are the descriptions of all the types of sections available:

Wide Flanges (W shapes)

All wide flange sections as listed in AISC are available the way they are written, e.g. W10X49, W21X50, etc.

```
20 TO 30 TA ST W10X49
33 36 TA ST W18X86
```

C, MC, S, M, HP Shapes

The above shapes are available as listed in AISC (9th Edition) without decimal points. For example, C8X11.5 will be input as C8X11 and S15X42.9 will be input as S15X42, omitting the decimal weights. (Exception: MC6X151 for MC6X15.1 and MC6X153 for MC6X15.3.)

```
10 TO 20 BY 2 TA ST C15X40
1 2 TA ST MC8X20
```

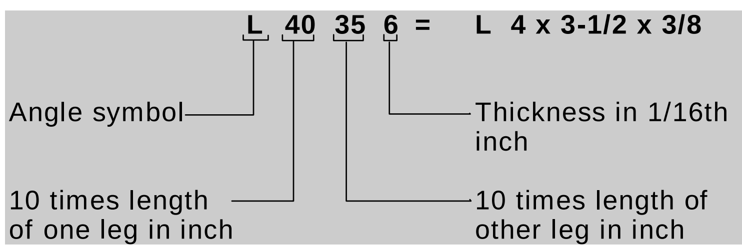
Double Channels

Back to back double channels, with or without spacing between them, are available. The letter D in front of the section name will specify a double channel.

```
21 22 24 TA D MC9X25
55 TO 60 TA D C8X18
```

Angles

Angle specifications in STAAD are different from those in the AISC manual. The following example illustrates angle specifications.



Similarly, L505010 = L 5 x 5 x 5/8 and L904016 = L 9 x 4 x 1

At present, there are two ways to define the local y and z-axes for an angle section. To make the transition from the AISC Manual to the program data easy, the standard section for an angle is specified:

```
51 52 53 TA ST L40356
```

This specification has the local z-axis (i.e., the minor axis) corresponding to the Z-Z axis specified in the steel tables. Many engineers are familiar with a convention used by some other programs in which the local y-axis is the minor axis. STAAD provides for this convention by accepting the command:

54 55 56 TA RA L40356	(RA denotes reverse angle)

Double Angles

Short leg back to back or long leg back to back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle either LD or SD will serve the purpose.

14 TO 20 TA LD L35304 SP 0.5	Long leg back to back L3-1/2x3x1/4 with 0.5 space
23 27 TA SD L904012	Short leg back to back L9x4x3/4

Tees

Tees are not input by their actual names, as they are listed in the AISC manual, but instead by designating the beam shapes (W and S) from which they are cut. For example,

1 2 5 8 TA T W8X24 tee cut from **W8X24** which is **WT4X12**

Pipes

Two types of specifications can be used for pipe sections. In general pipes may be input by their outer and inner diameters. For example,

1 TO 9 TA ST PIPE OD 2.0 ID 1.875	will mean a pipe with O.D. of 2.0 and I.D. of 1.875 in current input units.

Pipe sections listed in the AISC manual can be specified as follows.

5 TO 10 TA ST PIPX20

PIP X 20 denotes extra strong pipe of 2 in.dia

Pipe symbol

10 X Dia. in inches specify only portion before decimal point

Strength spec
(S = Standard, X = Extra-strong, D = Double extra-strong)

Tubes

Tubes from the AISC tables can be specified as follows.

5 TO 10 TA ST TUB120808

TUB 120 80 8

Tube Symbol

Thickness in 1/16th in.

Height x 10 (in.)

Width x 10 (in.)

Tubes, like pipes, can be input by their dimensions (Height, Width and Thickness) as follows.

> **6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5**
> is a tube that has a height of 8, a width of 6, and a wall thickness of 0.5.

Member Selection cannot be performed on tubes specified in the latter way. Only code checking can be performed on these sections.

Welded Plate Girders

The AISC welded plate girder shapes (pages 2-230 and 2-231 – AISC 9th edition) are available in the Steel Section library of the program.

B 61 20 10

- B – Built-up section symbol
- 61 – Nominal Depth in inches
- 20 – Nominal flange Width (inches)
- 10 – Thickness of flange in inches X 10 (Only use portion before decimal point)

Example :

```
1 TO 10 TA ST B612017
15 16 TA ST B682210
```

Castellated Beams

STAAD.Pro incorporates the non-composite castellated beam tables supplied by the steel products manufacturer SMI Steel Products. See section 2.16 for details.

2.3 Allowables per AISC Code

For steel design, STAAD compares the actual stresses with the allowable stresses as defined by the American Institute of Steel Construction (AISC) Code. The ninth edition of the AISC Code, as published in 1989, is used as the basis of this design (except for tension stress). Because of the size and complexity of the AISC codes, it would not be practical to describe every aspect of the steel design in this manual. Instead, a brief description of some of the major allowable stresses are described herein.

2.3.1 Tension Stress

Allowable tensile stress on the net section is calculated as 0.60 Fy.

2.3.2 Shear Stress

Allowable shear stress on the gross section,

$$F_v = 0.4F_y$$

2.3.3 Stress Due To Compression

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the formula E-1 in the AISC Code, when the largest effective slenderness ratio (Kl/r) is less than C_c. If Kl/r exceeds C_c, allowable compressive stress is decreased as per formula 1E2-2 of the Code.

$$C_c = \sqrt{2\pi^2 E / F_y}$$

2.3.4 Bending Stress

Allowable bending stress for tension and compression for a symmetrical member loaded in the plane of its minor axis, as given in Section 1.5.1.4 is:

$$F_b = 0.66F_y$$

If meeting the requirements of this section of:

a) $b_f/2t_f$ is less than or equal to $65/\sqrt{F_y}$
b) b_f/t_f is less than or equal to $190/\sqrt{F_y}$
c) d/t is less than or equal to $640(1-3.74(f_a/F_y))/\sqrt{F_y}$ when $(f_a/F_y) < 0.16$, or than $257/\sqrt{F_y}$ if $(f_a/F_y) > 0.16$
d) The laterally unsupported length shall not exceed $76.0b_f/F_y$ (except for pipes or tubes), nor $20{,}000/(dF_y/A_f)$
e) The diameter-thickness ratio of pipes shall not exceed $3300/F_y$

If for these symmetrical members, $b_f/2t_f$ exceeds $65/\sqrt{F_y}$, but is less than $95/\sqrt{F_y}$, $F_b=F_y(0.79-0.002(b_f/2t_f)\sqrt{F_y})$

For other symmetrical members which do not meet the above, F_b is calculated as the larger value computed as per AISC formulas F1-6 or F1-7 and F1-8 as applicable, but not more than $0.60F_y$. An unstiffened member subject to axial compression or compression due to bending is considered fully effective when the width-thickness ratio is not greater than the following:

$76.0/\sqrt{F_y}$, for single angles or double angles with separators.
$95.0/\sqrt{F_y}$, for double angles in contact.
$127./\sqrt{F_y}$, for stems of tees.

When the actual width-thickness ratio exceeds these values, the allowable stress is governed by B5 of the AISC code.

Tension and compression for the doubly symmetric (I & H) sections with $b_f/2t_f$ less than $65/\sqrt{F_y}$ and bent about their minor axis, $F_b = 0.75F_y$ If $b_f/2t_f$ exceeds $65/\sqrt{F_y}$, but is less than $95/\sqrt{F_y}$, $F_b = F_y(1.075-0.005(b_f/2t_f)\sqrt{F_y})$

For tubes, meeting the subparagraphs b and c of this Section, bent about the minor axis, $F_b= 0.66F_y$; failing the subparagraphs b and c but a width-thickness ratio less than $238/\sqrt{F_y}$, $F_b=0.6F_y$.

2.3.5 Combined Compression and Bending

Members subjected to both axial compression and bending stresses are proportioned to satisfy AISC formula H1-1 and H1-2 when f_a/F_a is greater than 0.15, otherwise formula H1-3 is used. It should be noted that during code checking or member selection, if f_a/F_a exceeds unity, the program does not compute the second and third part of the formula H1-1, because this would result in a misleadingly liberal ratio. The value of the coefficient C_m is taken as 0.85 for sidesway and 0.6 - 0.4 (M1/M2), but not less than 0.4 for no sidesway.

2.3.6 Singly Symmetric Sections

For double angles and Tees which have only one axis of symmetry, the KL/r ratio about the local Y-Y axis is determined using the clauses specified on page 3-53 of the AISC ASD 9th ed. Manual.

2.3.7 Torsion per Publication T114

The AISC 89 code of specifications for steel design currently does not have any provisions specifically meant for design of sections for Torsion. However, AISC has published a separate document called “Torsional Analysis of Steel Members” which provides guidelines on transforming torsional moments into normal stresses and shear stresses which can then be incorporated into the interaction equations explained in Chapter H of the AISC 89 code. The guidelines of the publication have been incorporated into the AISC-89 steel design modules of STAAD.

To consider stresses due to torsion in the code checking or member selection procedure, specify the parameter TORSION with a value of 1.0. See Table 2.1 for more details.

Methodology

If the user were to request design for torsion, the torsional properties required for calculating the warping normal stresses, warping shear stresses and pure shear stresses are first determined. These depend of the ”boundary” conditions that prevail at the ends of the member. These boundary conditions are defined as “Free”, “Pinned” or “Fixed”. They are explained below:

Free : “Free” represents the boundary condition such as that which exists at the free end of a cantilever beam. It means that there is no other member connected to the beam at that point.

Pinned : “Pinned” represents the condition that corresponds to either a pinned support defined at the joint through the Support command or a release of any of the moments at the joint through a Member Release specification.

Fixed : “Fixed” represents the condition where a fixed support exists at the joint. In the absence of a support at that joint, it represents a condition where a rigid frame connection exists between the given member and at least one other member connected to that joint. Also, no member releases should be present at that joint on the given member.

After the boundary conditions are determined, the normal and shear stresses are determined. The guidelines specified in the publication T114 for concentrated torsional moments acting at the ends of the member are used to determine these stresses.

The warping normal stresses are added to the axial stresses caused by axial load. These are then substituted into the interaction equations in Chapter H of the AISC 89 code for determining the ratio. The plane shear and warping shear stresses are added to the shear stresses causes by actual shear forces and compared against the allowable shear stresses on the cross section.

Restrictions

This facility is currently available for Wide Flange shapes (W, M & S), Channels, Tee shapes, Pipes and Tubes. It is not available for Single Angles, Double Angles, members with the PRISMATIC property specification, Composite sections (Wide Flanges with concrete slabs or plates on top), or Double Channels. Also, the stresses are calculated based on the rules for concentrated torsional moments acting at the ends of the member.

2.3.8 Design of Web Tapered Sections

Appendix F of AISC-89 provides specifications for design of Web-Tapered members. These specifications have been incorporated into STAAD to perform code checking on web tapered wide flange shapes. Member selection cannot be performed on web-tapered members.

2.3.9 Slender compression elements

For cross sections with elements which fall in the category of slender as per Table B5.1 of the AISC ASD code (the others being compact and non-compact), the rules of Appendix B of the code have been implemented. For stiffened compression elements, the effective cross section properties are calculated and used. For unstiffened compression elements, the allowable stresses are reduced per the Appendix.

2.4 Design Parameters

See Table 2.1 and Section 5.48.1

The program contains a large number of parameter names which are needed to perform designing and code checking. These parameter names, with their default values, are listed in Table 2.1. These parameters communicate design decisions from the engineer to the program.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ (k value in local z-axis) value of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instructions (Section 6). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

The parameters PROFILE, DMAX and DMIN are only used for member selection.

Table 2.1 - AISC (Ninth edition) Design Parameters

Parameter Name	Default Value	Description
KX	1.0	K value used in computing KL/r for flexural torsional buckling for tees and double angles
LX	Member Length	L value used in computing KL/r for flexural torsional buckling for tees and double angles
KY	1.0	K value in local y-axis. Usually, this is minor axis.
KZ	1.0	K value in local z-axis. Usually, this is major axis.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in local z-axis.
FYLD	36 KSI	Yield strength of steel in current units.
FU	Depends on FYLD	Ultimate tensile strength of steel in current units. If FYLD < 40 KSI, FU = 58 KSI If 40 KSI ≤ FYLD ≤ 50 KSI, FU = 60 KSI If FYLD > 50 KSI, FU = 65 KSI
NSF	1.0	Net section factor for tension members.
UNT	Member Length	Unsupported length of the top* flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the top flange.

Table 2.1 - AISC (Ninth edition) Design Parameters

Parameter Name	Default Value	Description
UNB	Member Length	Unsupported length of the bottom* flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the bottom flange.
CB	1.0	Cb value as used in section 1.5 of AISC. 0.0 = Cb value to be calculated. Any other value will mean the value to be used in design.
SSY	0.0	0.0 = Sidesway in local y-axis. 1.0 = No sidesway
SSZ	0.0	Same as above except in local z-axis.
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes
MAIN	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check Any value greater than 1 = Allowable KL/r in compression
TMAIN	300	Any value greater than 1 = Allowable KL/r in tension
STIFF	Member length or depth of beam whichever is greater	Spacing of stiffeners for plate girder design
TRACK	0.0	Controls the level of detail to which results are reported. 0 = Minimum detail 1 = Intermediate detail level 2 = Maximum detail (see Figure2.1)
DMAX	1000 in.	Maximum allowable depth.
DMIN	0.0 in.	Minimum allowable depth.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
WELD	1 for closed sections 2 for open sections	Weld type, as explained in section 2.12. A value of 1 will mean welding is on one side only except for wide-flange or tee sections, where the web is always assumed to be welded on both sides. A value of 2 will mean welding on both sides. For closed sections like pipe or tube, the welding will be on one side only.

Table 2.1 - AISC (Ninth edition) Design Parameters

Parameter Name	Default Value	Description
BEAM	1.0	0.0 = design at ends and those locations specified by the SECTION command. 1.0 = design at ends and at every 1/12th point along member length. (Default)
PROFILE	None	Used in member selection. See section 5.47.1 for details.
WMIN	See Sect. 2.12	Minimum welding thickness.
WMAX	See Sect. 2.12	Maximum welding thickness.
WSTR	0.4 x FYLD	Allowable welding stress.
DFF	None (Mandatory for deflection check)	"Deflection Length" / Maxm. allowable local deflection
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (See Note 1)
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (See Note 1)
CAN	0	0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = deflection check based on the principle that maximum deflection is of the cantilever type (see note below)
TORSION	0.0	0.0 = No torsion check performed. 1.0 = Perform torsion check based on rules of AISC T114.
TAPER	1.0	0.0 = Design tapered I-section based on rules of Chapter F and Appendix B only. Do not use the rules of Appendix F. 1.0 = Design tapered I-sections based on rules of Appendix F of AISC-89.
OVR	1.0	Overstress Factor. All the allowable stresses are multiplied by this number. It may be assigned any value greater than 0.0. Is used to communicate increases in allowable stress for loads like wind and earthquake.

Table 2.1 - AISC (Ninth edition) Design Parameters

Parameter Name	Default Value	Description
AXIS	1	1 - Design single angles for bending based on principal axis. 2 - Design single angles for bending based on geometric axis.
FLX	1	1 – Single Angle Member is not fully braced against lateral torsional buckling. 2 - Single Angle Member is fully braced against lateral torsional buckling. 3 - Single Angle Member is braced against lateral torsional buckling at the point of maximum moment.
STP	1	Section Type as defined in Table b%.1, page 5-36, ASD Manual. (1=Rolled, 2=Welded)

*Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if SET Z UP is used.

NOTES:

1) When performing the deflection check, the user can choose between two methods. The first method, defined by a value 0 for the CAN parameter, is based on the local displacement. Local displacement is described in section 5.43 of this manual.

   If the CAN parameter is set to 1, the check will be based on cantilever style deflection. Let (DX1, DY1,DZ1) represent the nodal displacements (in global axes) at the node defined by DJ1 (or in the absence of DJ1, the start node of the member). Similarly, (DX2, DY2,DZ2) represent the deflection values at DJ2 or the end node of the member.

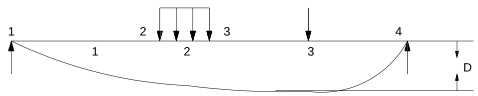
   Compute Delta = SQRT((DX2-DX1)**2 + (DY2-DY1)**2 + (DZ2-DZ1)**2)

   Compute Length = distance between DJ1 & DJ2 or, between start node and end node, as the case may be.

   Then, if CAN is specified a value 1, dff = L/Delta

Ratio due to deflection = DFF/dff

2) If CAN = 0, deflection length is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the “Deflection Length” will be equal to the length of the member. However, in some situations, the “Deflection Length” may be different. For example, refer to the figure below where a beam has been modeled using four joints and three members. The “Deflection Length” for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Also the straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured. Thus, for all three members here, DJ1 should be "1" and DJ2 should be "4".



D = Maximum local deflection for members 1 2 and 3.

EXAMPLE : PARAMETERS
DFF 300. ALL
DJ1 1 ALL
DJ2 4 ALL

3) If DJ1 and DJ2 are not used, "Deflection Length" will default to the member length and local deflections will be measured from the original member line.

4) It is important to note that unless a DFF value is specified, STAAD will not perform a deflection check. This is in accordance with the fact that there is no default value for DFF (see Table 2.1).

5) A critical difference exists between the parameters UNT/UNB and the parameters LY & LZ. UNT/UNB parameters represent the laterally unsupported length of the compression flange. It is defined in Chapter F, page 5-47 of the specifications in the AISC 1989 ASD manual as the distance between cross sections braced against twist or lateral displacement of the compression flange. UNT/UNB parameters are used to calculate the allowable compressive stress (FCZ and FCY) for behavior as a beam. LY and LZ on the other hand are the unbraced lengths

for behavior as a column and are used to calculate the KL/r ratios and the allowable axial compressive stress FA.

6) SSY and CMY are 2 parameters which are based upon 2 values defined in page 5-55, Chapter H of the AISC 9th ed. manual. SSY is a variable which allows the user to define whether or not the member is subject to sidesway in the local Y direction. CMY is a variable used for defining the expression called Cm in the AISC manual. When SSY is set to 0 (which is the default value), it means that the member is subject to sidesway in the local Y direction. When SSY is set to 1.0, it means that the member is not subject to sidesway in the local Y direction. The only effect that SSY has is that it causes the program to calculate the appropriate value of CMY. If SSY is set to 0 and CMY is not provided, STAAD will calculate CMY as 0.85. If SSY is set to 1 and CMY is not provided, STAAD will calculate CMY from the equation on page 5-55. However, if the user provides CMY, the program will use that value and not calculate CMY at all, regardless of what the user defines SSY to be.

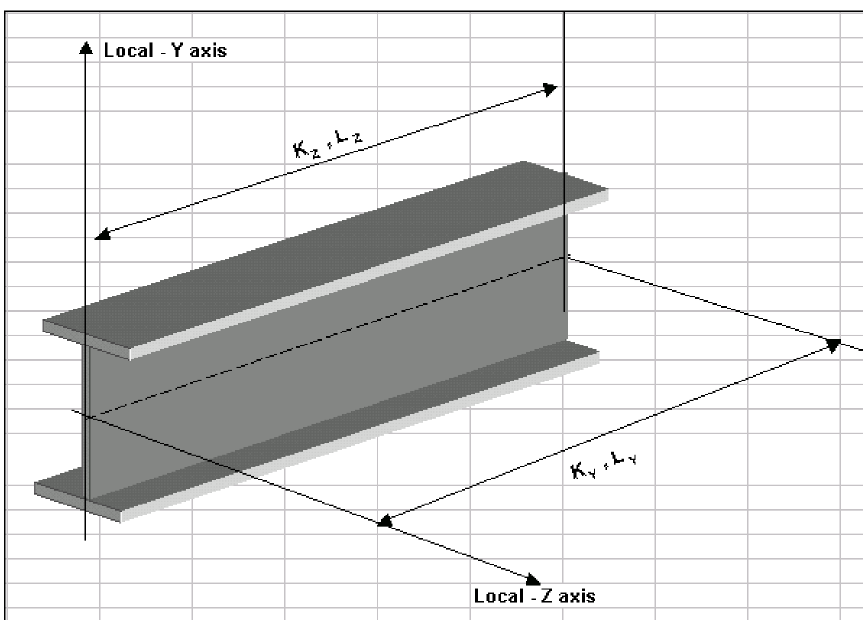


Figure 2.1 - Terms used in calculating slenderness ratios KL/r for local Y and Z axes

7) For a T shape which is cut from a parent I, W, S, M or H shapes, the PROFILE parameter should be assigned a value corresponding to the parent shape. For example, if the T desired is an American WT6, specify W12 for the PROFILE parameter.

2.5 Code Checking

See Section 5.48.2 and Example 1

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per AISC-89. Code checking is done using the forces and moments at specified sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed the code or have failed; the critical condition of the AISC code (like any of the AISC specifications or compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for a value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member) of forces in the member where the critical condition occurs.
Code checking can be done with any type of steel section listed in Section 2.2 of this manual.

2.6 Member Selection

See Section 5.48.3

STAAD is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e. the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. A wide flange will be selected to replace a wide flange, etc.. Several parameters are available to guide this selection. If the PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one. Member selection can also be constrained by the parameters DMAX and DMIN which limit the maximum and minimum depth of the members. If the PROFILE parameter is provided for specified members, DMAX or DMIN parameters will be ignored by the program in selecting these members.

Member selection can be performed with all the types of steel sections listed in Section 2.2 of this manual. For beams with cover plates, the sizes of the cover plate are kept constant while the beam section is iterated.

Selection of members, whose properties are originally input from a user created table, will be limited to sections in the user table. Member selection cannot be performed on members whose section properties are input as prismatic.

2.6.1 Member Selection by Optimization

See Section 5.48.4

Steel table properties of an entire structure can be optimized by STAAD. The method used in the optimization, which takes place if the SELECT OPTIMIZE command is specified, involves the following steps.

a. CHECK CODE ALL
b. Modify the ratios
c. SELECT ALL
d. PERFORM ANALYSIS
e. SELECT ALL

An additional step of Grouping may be performed if the FIXED GROUP and GROUP commands are provided (see section 5.49). After the last step, a re-analysis is not automatically performed, so users must ensure that they specify the analysis command following the SELECT OPTIMIZE command.

2.6.2 Deflection Check With Steel Design

This facility allows the user to consider deflection as a criteria in the CODE CHECK and MEMBER SELECTION processes. The deflection check may be controlled using the three parameters DJ1, DJ2 and DFF which are described in Table 2.1. Deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

2.7 Truss Members

As mentioned earlier, a truss member is capable of carrying only axial forces. So during the design phase, no time is wasted calculating the allowable bending or shear stresses. From this standpoint, if there is any truss member in an analysis (like bracing or strut, etc.), it is advisable to declare it as a truss member rather than as a regular frame member with both ends pinned.

2.8 Unsymmetric Sections

For unsymmetric sections like single angles, STAAD considers the smaller section modulus for calculating bending stresses.

For single angles, the “specification for allowable stress design of single-angle members”, explained in pages 5-309 to 5-314 of the AISC-ASD 9th edition manual has been incorporated.

2.9 Composite Beam Design as per AISC-ASD

In section 1.7.7 of this manual, two methods of specifying the properties of a beam as a composite section (I-shaped beam with concrete slab on top) are described. Those members can be designed as composite beams in accordance with the AISC ASD code provisions. If the properties are assigned using the explicit method as defined in section 1.7.7, the design parameters described in Table 2.2 later in this section have to be separately assigned. The CMP parameter in particular must be set a value of 1.0 or 2.0. If the properties are derived from the composite decks, the design parameters are automatically generated during the deck creation phase, and hence no separate parameters listed in Table 2.2 have to be assigned.

The following parameters have been introduced to support the composite member design, specified using the explicit definition method.

Table 2.2 – Composite Beam Design Parameters for AISC-ASD

Parameter Name	Default value	Description
CMP	0	Composite action with connectors 0 = design as a non-composite beam 1 = design as a composite beam if the slab is in bending compression throughout the span, design as a non-composite beam if the slab is in tension anywhere along the span 2 = design as a composite beam only. Ignore moments which cause tension in the slab.
DIA	0.625 in	Diameter of shear connectors
HGT	2.5 in	Height of shear connectors after welding
DR1	0.4	Ratio of moment due to dead load applied before concrete hardens to the total moment
WID	0.25 times the member length	Effective Width of concrete slab
FPC	3.0 ksi	Compressive strength of concrete at 28 days
PLT	0.0	Thickness of cover plate welded to bottom flange of composite beam
PLW	0.0	Width of cover plate welded to bottom flange of composite beam
RBH	0.0	Height of rib of form steel deck
RBW	2.5 in	Width of rib of form steel deck

Table 2.2 – Composite Beam Design Parameters for AISC-ASD

Parameter Name	Default value	Description
SHR	0	Temporary shoring during construction 0 = without shoring 1 = with shoring
THK	4.0 in	Thickness of concrete slab or the thickness of concrete slab above the form steel deck.

```
UNIT INCH
PARAMETER
CODE AISC
BEAM 1 ALL
TRACK 2 ALL
DR1 0.3135 ALL
WID 69.525 ALL
FPC 3.0 ALL
THK 4.0 ALL
CMP 1 ALL
CHECK CODE ALL
SELECT ALL
```

2.10 Plate Girders

The requirements of Chapter G – pages 5-51 through 5-53 of the AISC ASD 9th edition manual – are currently not implemented. So, if the web slenderness ratio h/tw of a section exceeds 970/sqrt (Fy), STAAD does not design that member.

2.11 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

a) MEMBER refers to the member number for which the design is performed.

b) TABLE refers to the AISC steel section name which has been checked against the steel code or has been selected.

c) RESULT prints whether the member has PASSed or FAILed. If the RESULT is FAIL, there will be an asterisk (*) mark in front of the member number.

d) CRITICAL COND refers to the section of the AISC code which governed the design.

e) RATIO prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.

f) LOADING provides the load case number which governed the design.

g) FX, MY and MZ provide the axial force, moment in local y-axis and moment in local z-axis respectively. Although STAAD does consider all the member forces and moments to perform design, only FX MY and MZ are printed since they are the ones which are of interest, in most cases.

h) LOCATION specifies the actual distance from the start of the member to the section where design forces govern.

i) If the parameter TRACK is set to 1.0, the program will block out part of the table and will print the allowable bending stresses in compression (FCY & FCZ) and tension (FTY &

FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV), all in kips per square inch. In addition, member length, area, section moduli, governing KL/r ratio and CB are also printed.

j) In the output for TRACK 2.0, the items Fey and Fez are as follows:

$$\mathrm{Fey} = \frac{12\pi^2 E}{23\left(K_Y L_Y / r_Y\right)^2}$$

$$\mathrm{Fez} = \frac{12\pi^2 E}{23\left(K_Z L_Z / r_Z\right)^2}$$

```
                    STAAD.Pro MEMBER SELECTION - (AISC 9TH EDITION)
                    **************************

|-----------------------------------------------------------------------------|
|                                                      Y        PROPERTIES    |
|*************                                         |        IN INCH UNIT  |
|           *   |=============================|   ===|===       ------------ |
|MEMBER   2  *  |                             |      |          AX =   8.85   |
|           *   |       ST  W14X30            |      |    --Z   AY =   3.39   |
|DESIGN CODE *  |                             |      |          AZ =   3.47   |
| AISC-1989  *  ===============================   ===|===       SY =   5.82   |
|           *                                                   SZ =  42.05   |
|           *   |<---LENGTH (FT)=   19.00 --->|                 RY =   1.49   |
|*************                                                  RZ =   5.73   |
|                                                                             |
|                70.5 (KIP-FEET)                                              |
|PARAMETER       |               L1  L1                         STRESSES      |
|IN KIP  INCH    |                       L1                     IN KIP  INCH  |
|---------------  +                                      L3     -------------|
| KL/R-Y= 153.21  |           L3                                FA  =   6.36  |
| KL/R-Z=  39.76  +                                             fa  =   0.45  |
| UNL   = 120.00  |       L3                  L1                FCZ =  21.68  |
| CB    =   1.00  +L1                                           FTZ =  24.00  |
| CMY   =   0.85  |   L3                              L3        FCY =  30.00  |
| CMZ   =   0.85  +                                             FTY =  30.00  |
| FYLD  =  40.00  |                               L1            fbz =  20.13  |
| NSF   =   1.00  +---+---+---+---+---+---+---+---+---|         fby =   0.00  |
| DFF   =   0.00  13.8                                          FV  =  16.00  |
| dff   =   0.00              ABSOLUTE MZ ENVELOPE              Fey =   6.36  |
|                                (WITH LOAD NO.)                 Fez =  94.46  |
|                                                                             |
|                   MAX FORCE/ MOMENT SUMMARY  (KIP-FEET)                      |
|                   -------------------------                                 |
|                                                                             |
|               AXIAL      SHEAR-Y    SHEAR-Z    MOMENT-Y    MOMENT-Z          |
|                                                                             |
|      VALUE       4.7        24.3        0.0         0.0         70.5         |
|   LOCATION       0.0        19.0        0.0         0.0          9.5         |
|    LOADING        3           1          0           0           1          |
|                                                                             |
|*****************************************************************************|
|*                                                                           *|
|*                         DESIGN SUMMARY  (KIP-FEET)                        *|
|*                         --------------                                    *|
|*                                                                           *|
|*       RESULT/     CRITICAL COND/     RATIO/         LOADING/              *|
|         FX              MY              MZ          LOCATION               |
|        =======================================================             |
|          PASS       AISC- H1-3        0.999              1                 |
|          3.99 C         0.00        -70.55            9.50                 |
|*                                                                           *|
|*****************************************************************************|
|                                                                             |
|-----------------------------------------------------------------------------|
```

Figure 2.2

2.12 Weld Design

Selected provisions of the AISC specifications for the Design, Fabrication and Erection of Steel for Buildings, 1999, and the American Welding Society D1.1 Structural Welding Code – Steel, 1998, have been implemented.

See Section 5.48.5

STAAD is able to select weld thickness for connections and tabulate the various stresses. The weld design is limited to the members having properties from wide flange, tee, single angle, single channel, pipe and tube section tables only. The parameters WELD, WMIN and WSTR (as explained in Table 2.1) govern the weld design.

Since the thickness of a weld is very small in comparison to its length, the properties of the weld can be calculated as line member. Therefore, the cross-sectional area (AZ) of the weld will actually be the length of the weld. Similarly, the units for the section moduli (SY and SZ) will be length-squared and for the polar moments of inertia (JW) will be length-cubed. The following table shows the different available weld lines, their type and their coordinate axes.

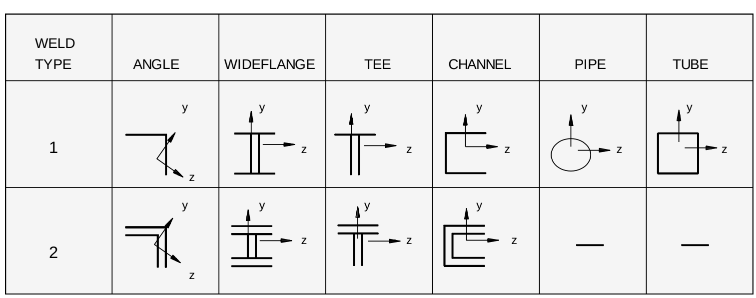


Figure 2.3

Actual stresses, calculated from the member forces, can be specified by three names, based on their directions.

Horizontal Stress - as produced by the local z-shear force and torsional moment.

Vertical Stress - as produced by the axial y-shear force and torsional moment.

Direct Stress - as produced by the axial force and bending moments in the local y and z directions.

The Combined Stress is calculated by the square root of the summation of the squares of the above three principal stresses.

Following are the equations:

Forces

MX = Torsional moment
MY = Bending in local y-axis
MZ = Bending in local z-axis
FX = Axial force
VY = Shear in local y-axis
VZ = Shear in local z-axis

Properties of Weld

AX = Area of the weld as the line member
SY = Section modulus around local y-axis
SZ = Section modulus around local z-axis
JW = Polar moment of inertia
CH = Distance of the extreme fiber for horizontal (local z) forces
CV = Distance of the extreme fiber for vertical (local y) forces

Stress Equations:

Horizontal stress, $$Fh = \frac{VZ}{AX} + \frac{CH \times MX}{JW}$$

$$\text{Vertical stress, } Fv = \frac{VY}{AX} + \frac{CV \times MX}{JW}$$

$$\text{Direct stress, } Fd = \frac{FX}{AX} + \frac{MZ^{*}}{SZ} + \frac{MY^{*}}{SY}$$

* The moments MY and MZ are taken as absolute values, which may result in some conservative results for asymmetrical sections like angle, tee and channel.

$$\text{Combined force } F_{comb} = \sqrt{F_h^2 + F_v^2 + F_d^2}$$

$$\text{Weld thickness} = \frac{F_{comb}}{F_w}$$

where F_w = Allowable weld stress, default value is 0.4 FYLD (Table 2.1).

The thickness t is rounded up to the nearest 1/16th of an inch and all the stresses are recalculated. The tabulated output prints the latter stresses. If the parameter TRACK is set to 1.0, the output will include the weld properties. The program does not calculate the minimum weld thickness as needed by some codes, but checks only against the minimum thickness as provided by the user (or 1/16th inch if not provided).

When the TRUSS qualifier is used with SELECT WELD command, the program will design the welds required for truss angle and double angle members that are attached to gusset plates. The program reports the number of welds (two for single angles, four for double angles), and the length required for each weld. The thickness of the weld is taken as 1/4 inch (6 mm) for members up to 1/4 inch (6 mm) thick, and 1/16 inch (1.5 mm) less than the angle thickness for members greater than 1/4 inch (6 mm) thick. Minimum weld length is taken as four times weld thickness.

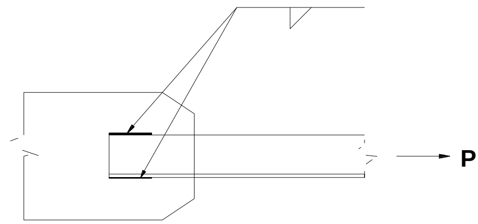


Figure 2.4 - Weld design for SELECT WELD TRUSS.

2.13 Steel Design per AASHTO Specifications

Design to AASHTO Standard Specifications for Highway Bridges utilizing the ASD and LRFD approaches is implemented in STAAD. These are described in the following two sections.

To utilize the ASD method, specify the commands

PARAMETER
CODE AASHTO

Or

PARAMETER
CODE AASHTO ASD

To utilize the LRFD method, specify the commands

PARAMETER
CODE AASHTO LRFD

2.13.1 AASHTO ASD

The design of structural steel members in accordance with the AASHTO Standard Specifications for Highway Bridges, 17th edition has been implemented.

General Comments

The section of the above code implemented in STAAD is Chapter 10, Part C – Service Load design Method, Allowable Stress design. Sections 10.32.1.A and 10.36 are implemented. As per the AASHTO committee, this is the last edition for this code (the ASD approach) and only technical errors will be fixed in the future for this code.

In general, the concepts followed in MEMBER SELECTION and CODE CHECKING procedures are similar to that of the AISC based design. It is assumed that the user is familiar with the basic concepts of steel design facilities available in STAAD. Please refer to Section 2 of the STAAD Technical Reference Manual for detailed information on this topic. This section specifically addresses the implementation of steel design based on the AASHTO specifications.

Design is available for all standard sections listed in the AISC ASD 9th edition manual, namely, Wide Flanges, S, M, HP, Tees, Channels, Single Angles, Double Angles, Tubes and Pipes. The design of HSS sections (those listed in the 3rd edition AISC LRFD manual) and Composite beams (I shapes with concrete slab on top) are not supported.

Allowable Stresses per AASHTO Code

The member design and code checking in STAAD is based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. It is beyond the scope of this manual to describe every aspect of structural steel design per AASHTO specifications because of practical reasons. This section will discuss the salient features of the allowable stresses specified by the AASHTO code. Table 10.32.1A of the AASHTO code specifies the allowable stresses.

Axial Stress

Allowable tension stress, as calculated in AASHTO is based on the net section. This tends to produce a slightly conservative result. Allowable tension stress on the net section is given by,

$$F_t = 0.55F_y$$

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$F_a = \frac{F_y}{F.S.} \quad \frac{(1 - (Kl/r)^2 F_y}{4\pi^2 E} \qquad \text{when}(Kl/r) < C_c$$

$$F_a = \frac{\pi^2 E}{F.S.(Kl/r)^2} \qquad \text{when } (Kl/r) > C_c$$

$$\text{with } C_c = \ (2\pi^2 E/F_y)^{1/2} \quad \text{and F.S.} = 2.12$$

It should be noted that AASHTO does not have a provision for increase in allowable stresses for a secondary member and when 1/r exceeds a certain value.

Bending Stress

Allowable stress in bending compression for rolled shape girders and built-up sections whose compression flanges are supported laterally through their full length by embedment in concrete is given by:-

$$F_b = 0.55F_y$$

For similar members with unsupported or partially supported flange lengths, the allowable bending compressive stress is given by

$$F_b = 0.55F_y\,(1 - \frac{(1/r)^2 F_y}{4\pi^2 E}) \qquad \text{when } (kl/r) < C_c$$

$$\text{with}(r^2) = b^2/12$$

Due to inadequate information in the AASHTO Code, the allowable tensile stresses due to bending for both axes are set to be the same as the corresponding allowable bending compressive stresses.

Shear Stress

Allowable shear stress on the gross section is given by:-

$$F_v = 0.33F_y$$

For shear on the web, the gross section is defined as the product of the total depth and the web thickness. The AASHTO code does not specify any allowable stress for shear on flanges. The program assumes the same allowable for shear stress (0.33Fy) for both shear on the web and shear on the flanges. For shear on the flanges, the gross section is taken as 2/3 times the total flange area.

Bending-Axial Stress Interaction

Members subjected to both axial and bending stresses are proportioned according to section 10.36 of the AASHTO steel code. All members subject to bending and axial compression are required to satisfy the following formula:

$$\frac{f_a}{F_a} + \frac{C_{mx} f_{bx}}{(1 - f_a / F_{ex}) F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a / F_{ey}) F_{by}} < 1.0$$

at intermediate points, and

$$\frac{f_a}{.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} < 1.0$$

at support points.

The start and end nodes of a member are treated as support points.

For members subject to axial tension and bending, the following equations are checked.

$$\frac{f_a}{F_a} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} < 1.0$$

at intermediate points, and

$$\frac{f_a}{0.472 F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} < 1.0$$

at ends of member.

AASHTO ASD Design Parameters

The following table outlines the parameters that can be used with the AASHTO (ASD) code along with the default values used if not explicitly specified.

Table 2.3 - AASHTO ASD Design Parameters

Parameter Name	Default Value	Description
FYLD	36 KSI	Yield strength of steel in current units.
FU	Depends on FYLD	Ultimate tensile strength of steel in current units.
RATIO	1.0	Permissible ratio of the actual to allowable stresses.
KY	1.0	K value in local y-axis. Usually, this is minor axis.
KZ	1.0	K value in local z-axis. Usually, this is major axis.
UNF	1.0	Unsupported length factor of the compression flange for calculating the allowable bending compressive strength.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y axis.
LZ	Member Length	Same as above except in local z-axis.
UNL	Member Length	Unsupported length of compression flange for calculating allowable bending compressive stress.
NSF	1.0	Ratio of 'Net cross section area' / 'Gross section area' for tension member design.
SSY	0.0	0.0 =Sidesway in local y-axis. 1.0 =No sidesway in local y-axis
SSZ	0.0	0.0 =Sidesway in local z-axis. 1.0 =No sidesway in local z-axis.
CMY CMZ	0.85 for sidesway and calculated for no sidesway	Cm value in local y & z axes

Table 2.3 - AASHTO ASD Design Parameters

Parameter Name	Default Value	Description
MAIN	0.0	0.0 =check for slenderness 1.0 =suppress slenderness check
BEAM	1.0	0.0 = Design at ends and those locations specified by the SECTION command. 1.0 = Design at ends and every 1/12th point along the member length.
CB	1.0	Cb value as used in the calculation of Fb 0.0 = Cb value to be calculated Any other value will be used in the calculations.
DMAX	1000.0	Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command.
DMIN	0.0	Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.
PROFILE	None	Used in member section. See section 5.47.1 of Technical Reference Manual for more details.
PUNCH		1.0 = K-Overlap 2.0 = K-Gap 3.0 = T and Y 4.0 = Cross with diaphragms 5.0 = Cross without diaphragms
STIFF	Greater of member length or depth of beam.	Spacing of stiffeners for plate girder design in current length units.

Table 2.3 - AASHTO ASD Design Parameters

Parameter Name	Default Value	Description
<u>TRACK</u>	0	Level of detail in Output File:- 0 = Print the design output at the minimum detail level. 1 = Print the design output at the intermediate detail level. 2 = Print the design output at maximum detail level..
<u>DFF</u>	None. (Mandatory for a deflection check)	"Deflection length" / Maximum allowable local axis deflection.
<u>DJ1</u>	Start joint of member	Joint No. denoting starting point for calculating "Deflection Length".
<u>DJ2</u>	End joint of member	Joint No. denoting ending point for calculating "Deflection Length".
<u>WSTR</u>	0.4 x FYLD	Allowable welding stress
<u>WMAX</u>	See Section 2.12	Maximum welding thickness
<u>WMIN</u>	See Section 2.12	Minimum welding thickness
<u>WELD</u>	1 for open sections 2 for closed sections	Weld type as described in section 2.11. 6.0 = Welding on one side except for wide-flange or tee sections where the web is always assumed welded on both sides. 7.0 Welding on both sides. Closed sections such as pipes and tubes will only be considered as welded on one side.

2.13.2 AASHTO LRFD

The following outlines the implementation of the AASHTO Standard Specifications for Highway Bridges (LRFD, 1998) which has been implemented in STAAD.Pro.

General Comments

The design philosophy embodied in the Load and Resistance Factor Design (LRFD) Specification is built around the concept of limit state design, the current state-of-the-art in structural engineering. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of AASTHO-LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the AASTHO-LRFD specifications as implemented in STAAD steel design.

Capacities per AASHTO LRFD Code

Axial Strength

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF. STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly

$Pr = \varphi_y P_{ny} = \varphi_y F_y A_g$

$Pr = \varphi_u P_{nu} = \varphi_u F_u A_n U$

P_{ny} = Nominal tensile resistance for yielding in gross section (kip)

F_y = Yield strength (ksi)

A_g = Gross cross-sectional area of the member (in^2)

P_{nu} = Nominal tensile resistance foe the fracture in the net section (kip)

F_u = Tensile strength (ksi)

A_n = Net area of the member

U = reduction factor to account for shear lag.

φ_y = resistance factor for yielding of tension member

φ_u = resistance factor for fracture of tension members

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$\lambda = \left(\frac{Kl}{r_s \pi}\right)^2 \frac{F_y}{E}$$

if $\lambda \le 2.25$

Nominal compressive resistance, $Pn = 0.66^{\lambda} F_y A_s$

if $\lambda > 2.25$

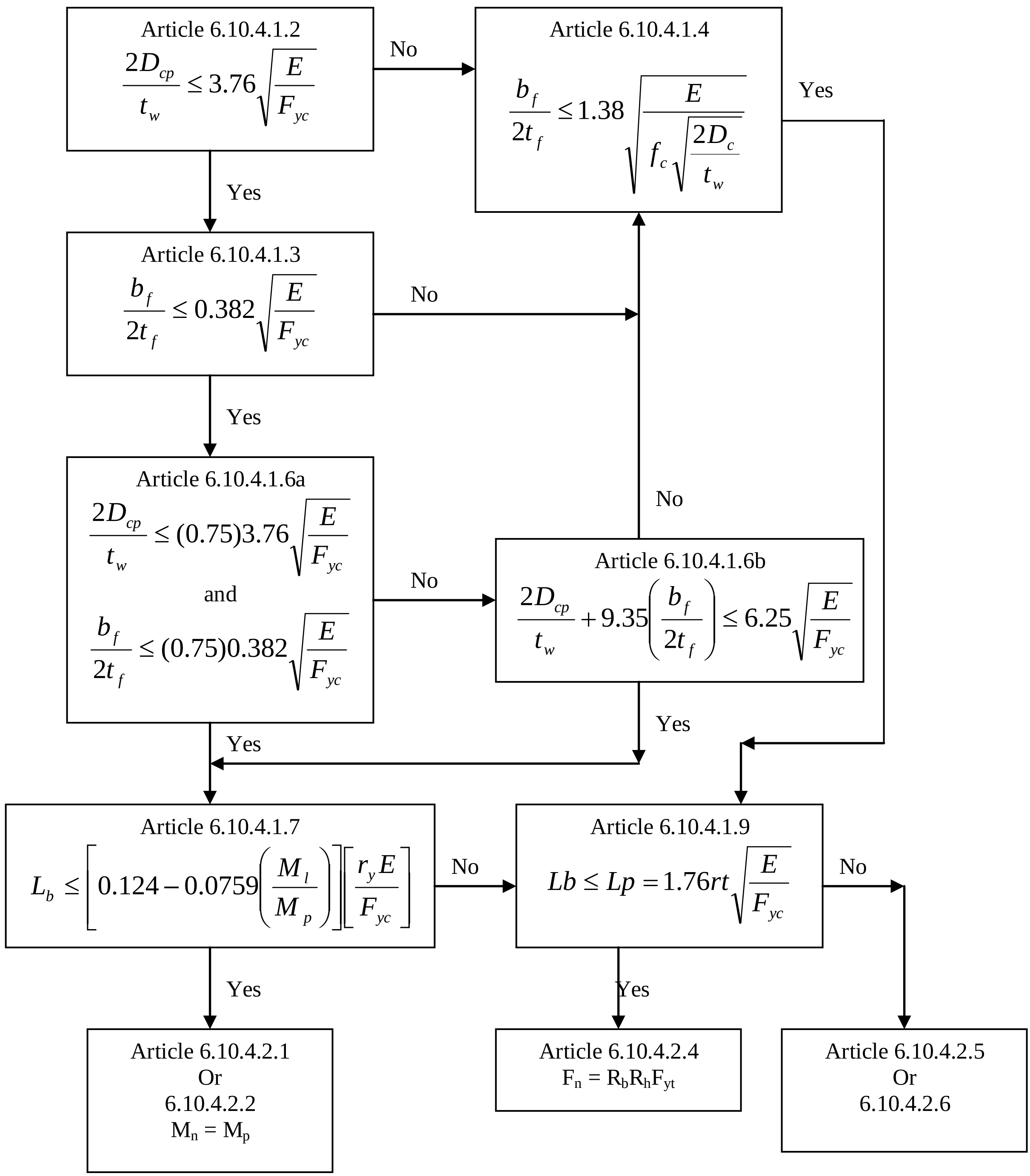
Nominal compressive resistance, $Pn = \dfrac{0.88 F_y A_s}{\lambda}$

A_s = Gross sectional area

The Factored resistance, $Pr = \varphi_c\ Pn$

Bending Strength

The flow to calculate the allowable bending strength for rolled shape girders and built-up sections is given by the flowing flow chart.


Article 6.10.4.1.2
$\frac{2D_{cp}}{t_w} \le 3.76\sqrt{\frac{E}{F_{yc}}}$
No
Article 6.10.4.1.4
$\frac{b_f}{2t_f} \le 1.38\sqrt{\frac{E}{f_c\sqrt{\frac{2D_c}{t_w}}}}$
Yes
Yes
Article 6.10.4.1.3
$\frac{b_f}{2t_f} \le 0.382\sqrt{\frac{E}{F_{yc}}}$
No
Yes
No
Article 6.10.4.1.6a
$\frac{2D_{cp}}{t_w} \le (0.75)3.76\sqrt{\frac{E}{F_{yc}}}$
and
$\frac{b_f}{2t_f} \le (0.75)0.382\sqrt{\frac{E}{F_{yc}}}$
No
Article 6.10.4.1.6b
$\frac{2D_{cp}}{t_w} + 9.35\left(\frac{b_f}{2t_f}\right) \le 6.25\sqrt{\frac{E}{F_{yc}}}$
Yes
Yes
Article 6.10.4.1.7
$L_b \le \left[0.124 - 0.0759\left(\frac{M_l}{M_p}\right)\right]\left[\frac{r_y E}{F_{yc}}\right]$
No
Article 6.10.4.1.9
$Lb \le Lp = 1.76rt\sqrt{\frac{E}{F_{yc}}}$
No
Yes
Yes
Article 6.10.4.2.1
Or
6.10.4.2.2
$M_n = M_p$
Article 6.10.4.2.4
$F_n = R_b R_h F_{yt}$
Article 6.10.4.2.5
Or
6.10.4.2.6

Shear Strength

The nominal shear resistance of un-stiffened webs of homogeneous girders shall be calculated as.

If $\frac{D}{t_w} \leq 2.46\sqrt{\frac{E}{F_{yw}}}$, then

$V_n = V_p = 0.58F_{yw}Dt_w$

If $2.46\sqrt{\frac{E}{F_{yw}}} < \frac{D}{t_w} \leq 3.07\sqrt{\frac{E}{F_{yw}}}$, then

$V_n = 1.48t_w^2\sqrt{EF_{yw}}$

If $\frac{D}{t_w} > 3.07\sqrt{\frac{E}{F_{yw}}}$, then

Bending-Axial Interaction

Members subjected to both axial forces and bending moments are proportioned according to section 6.9.2.2 of the AASHTO steel code. All members subject to bending and axial compression or axial tension are required to satisfy the following formula:

If $\frac{Pu}{\Pr} < 0.2$, then

$$\frac{Pu}{2.0\Pr} + \left(\frac{Mux}{Mrx} + \frac{Muy}{Mry}\right) \leq 1.0$$

If $\frac{Pu}{\mathrm{Pr}} \geq 0.2$, then

$$\frac{Pu}{\mathrm{Pr}} + \frac{8.0}{9.0}\left(\frac{Mux}{Mrx} + \frac{Muy}{Mry}\right) \leq 1.0$$

AASHTO LRFD Design Parameters

The following table outlines the parameters that can be used with the AASHTO (LRFD) code along with the default values used if not explicitly specified.

Table 2.4 - AASHTO LRFD Design Parameters

Parameter Name	Default Value	Description
GRADE	1	Grade of Steel: 1: Grade 36 2: Grade 50 3 Grade 50W 4: Grade 70W 5: Grade 100/100W Refer to AASHTO LRFD, Table 6.4.1-1
LY	Member Length	Length to calculate slenderness ratio for buckling about the local Y axis.
LZ	Member Length	Same as above except in local z-axis.
KY	1.0	K value in local y-axis. Usually, this is the minor axis.
KZ	1.0	K value in local z-axis. Usually, this is the major axis.
NSF	1.0	Net Section Factor. Ratio of (Net Area)/(Gross Area)

Table 2.4 - AASHTO LRFD Design Parameters

Parameter Name	Default Value	Description
TRACK	0	Level of detail in Output File:- 0 = Print the design output at the minimum detail level. 1 = Print the design output at the intermediate detail level. 2 = Print the design output at maximum detail level..
NSF	1.0	Net section factor for tension members.
UNT	Member Length	Unsupported length of top flange. Used for calculating the moment of resistance when top of beam is in compression.
UNB	Member Length	Unsupported length of bottom flange. Used for calculating the moment of resistance when the bottom of beam is in compression.
BEAM	1.0	Identify where beam checks are performed: 0 = Perform design at ends and those locations specified in the SECTION command. 1 = Perform design at ends and 1/12th section locations along member length.
MAIN	0.0	Flag for checking slenderness limit: 0.0 =check for slenderness 1.0 =suppress slenderness check
DMAX	1000	Maximum allowed section depth (in current length units) for a section to be selected with the SELECT command.
DMIN	0	Minimum allowed section depth (in current length units) for a section to be selected with the SELECT command.
DFF	0	"Deflection Length"/Max allowable local deflection If set to 0, (default) then no deflection check is performed.

Table 2.4 - AASHTO LRFD Design Parameters

Parameter Name	Default Value	Description
DJ1	Start joint of member	Joint No. denoting starting point for calculating "Deflection Length".
DJ2	End joint of member	Joint No. denoting ending point for calculating "Deflection Length".

2.14 Steel Design per AISC/LRFD Specification

The 2nd and 3rd editions of the American LRFD code have been implemented. The commands to access those respective codes are:

For the 3rd edition code –

```
PARAMETER
CODE LRFD

or

PARAMETER
CODE LRFD3
```

For the 2nd edition –

```
PARAMETER
CODE LRFD2
```

2.14.1 General Comments

The design philosophy embodied in the Load and Resistance Factor Design (LRFD) Specification is built around the concept of limit state design. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized--ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the LRFD specifications as implemented in STAAD steel design. A detailed description of the design process along with its underlying concepts and assumptions is available in the LRFD manual. However, since the design philosophy is drastically different from the conventional Allowable Stress Design (ASD), a brief description of the fundamental concepts is presented here to initiate the user into the design process.

2.14.2 LRFD Fundamentals

The primary objective of the LRFD Specification is to provide a uniform reliability for all steel structures under various loading conditions. This uniformity can not be obtained with the allowable stress design (ASD) format.

The ASD method can be represented by the inequality

$$\sum Q_i < R_n / F.S.$$

The left side is the required strength, which is the summation of the load effects, Qi (forces and moments). The right side, the design strength, is the nominal strength or resistance, R_n, divided by a factor of safety. When divided by the appropriate section property (area or section modulus), the two sides of the inequality become the actual stress and allowable stress respectively. ASD, then, is characterized by the use of unfactored "working" loads in conjunction with a single factor of safety applied to the resistance. Because of the greater variability and, hence, unpredictability of the live load and other loads in comparison with the dead load, a uniform reliability is not possible.

LRFD, as its name implies, uses separate factors for each load and resistance. Because the different factors reflect the degree of uncertainty of different loads and combinations of loads and of the accuracy of predicted strength, a more uniform reliability is possible. The LRFD method may be summarized by the inequality

$$y_i Q_i < R_n \varnothing$$

On the left side of the inequality, the required strength is the summation of the various load effects, Q_i, multiplied by their respective load factors, y_i. The design strength, on the right side, is the nominal strength or resistance, R_n, multiplied by a resistance factor, Ø.

In the STAAD implementation of LRFD, it is assumed that the user will use appropriate load factors and create the load combinations necessary for analysis. The design portion of the program will take into consideration the load effects (forces and moments) obtained from analysis. In calculation of resistances of various elements (beams, columns etc.), resistance (nominal strength) and applicable resistance factor will be automatically considered.

2.14.3 Analysis Requirements

The types of construction recognized by AISC specification have not changed, except that both "simple framing" (formerly Type 2) and "semi-rigid framing" (formerly Type 3) have been combined into the same category, Type PR (partially restrained). "Rigid Framing" (formerly Type 1) is now Type FR (fully restrained). Type FR construction is permitted unconditionally. Type PR construction may necessitate some inelastic, but self-limiting, deformation of a structural steel element. Thus, when specifying Type PR construction, the designer should take into consideration the effects of partial restraint on the stability of the structure, lateral deflections and second order bending moments. As stated in Sect. C1 of the LRFD specification, an analysis of second order effects is required. Thus, when using LRFD code for steel design, the user must use the P-Delta analysis feature of STAAD.

2.14.4 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, noncompact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD is capable of determining the section classification for the standard shapes and user specified shapes and design accordingly.

2.14.5 Axial Tension

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF (see Table 2.2). STAAD calculates the tension capacity of a given member based on these two limit states and proceeds with member selection or code check accordingly.

2.14.6 Axial Compression

The column strength equations have been revised in LRFD to take into account inelastic deformation and other recent research in column behavior. Two equations governing column strength are available, one for inelastic buckling and the other for elastic or Euler buckling. Both equations include the effects of residual stresses and initial out-of-straightness. Compression strength for a particular member is calculated by STAAD according to the procedure outlined in Chapter E of the LRFD specifications. For slender elements, the procedure described in Appendix B5.3 is used.

Singly symmetric and unsymmetric compression members are designed on the basis of the limit states of flexural-torsional and torsional buckling. The procedure of Appendix E3 is implemented for the determination of design strength for these limit states.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ and/or LY, LZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

2.14.7 Flexural Design Strength

In LRFD, the flexural design strength of a member is determined by the limit state of lateral torsional buckling. Inelastic bending is allowed and the basic measure of flexural capacity is the plastic moment capacity of the section. The flexural resistance is a function of plastic moment capacity, actual laterally unbraced length, limiting laterally unbraced length, buckling moment and the bending coefficient. The limiting laterally unbraced length L_r and buckling moment M_r are functions of the section geometry and are calculated as per the procedure of Chapter F. The purpose of bending coefficient C_b is to account for the influence of the moment gradient on lateral-torsional buckling. This coefficient can be specified by the user through the use of parameter CB (see Table 2.2) or may be calculated by the program (if CB is specified as 0.0). In the absence of the parameter CB, a default value of 1.0 will be used. The procedure for calculation of design strength for flexure also accounts for the presence of residual stresses of rolling. To specify laterally unsupported length, the parameters UNT/UNB (see Table 2.2) can be used.

2.14.8 Combined Axial Force And Bending

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by formulas H1-1a and H1-1b. These interaction formulas cover the general case of biaxial bending combined with axial force. They are also valid for uniaxial bending and axial force.

2.14.9 Design for Shear

The procedure of Sect. F2 of the LRFD Specification is used in STAAD to design for shear forces in members. Shear strength as calculated in LRFD is governed by the following limit states: Eq. F2-1a by yielding of the web; Eq. F2-2a by inelastic buckling of the web; Eq. F2-3a by elastic buckling of the web. Shear in wide flanges and channel sections is resisted by the area of the web, which is taken as the overall depth times the web thickness.

2.14.10 Design Parameters

Design per LRFD specifications is requested by using the CODE parameter (see Section 5.48). Other applicable parameters are summarized in Table 2.5. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

See Table 2.2 and Section 5.48.1

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

The parameters DMAX and DMIN are applicable for member selection only.

Table 2.5 - AISC LRFD (2nd and 3rd editions) Design Parameters

Parameter Name	Default Value	Description
KX	1.0	K value for flexural-torsional buckling.
KY	1.0	K value in local Y-axis. Usually this is the minor axis.
KZ	1.0	K value in local Z-axis. Usually this is the major axis.
LX	Member Length	Length for flexural-torsional buckling.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y-axis.
LZ	Member Length	Length to calculate slenderness ratio for buckling about local Z-axis.
FYLD	36.0 ksi	Yield strength of steel.
STP	1	Section Type to determine Fr (compressive residual stress in Flange) per Third Ed. LRFD specs. Page 16.1-97 1 = Rolled section (Fr-10 ksi) 2 = Welded section (Fr-16.5 ksi)
FU	60.0 ksi	Ultimate tensile strength of steel.
NSF	1.0	Net section factor for tension members.
UNT	Member Length	Unsupported length (L_b) of the top* flange for calculating flexural strength. Will be used only if flexural compression is on the top flange.
UNB	Member Length	Unsupported length (L_b) of the bottom* flange for calculating flexural strength. Will be used only if flexural compression is on the bottom flange.
STIFF	Member Length or depth whichever is greater	Spacing of stiffeners for beams for shear design.
CB **	1.0	Coefficient C_b per Chapter F. If C_b is set to 0.0, it will be calculated by the program. Any other value will be directly used in design.

Table 2.5 - AISC LRFD (2^{nd} and 3^{rd} editions) Design Parameters

Parameter Name	Default Value	Description
TRACK	0.0	0.0 = Suppress all design strengths. 1.0 = Print all design strengths. 2.0 = Print expanded design output.
DMAX	45.0 in.	Maximum allowable depth.
DMIN	0.0 in.	Minimum allowable depth.
DFF	None (Mandatory for deflection check)	"Deflection Length" / Maxm. allowable local deflection
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (See Note 1 of Table 2.1)
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (See Note 1of Table 2.1)
CAN	0	0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = deflection check based on the principle that maximum deflection is of the cantilever type
RATIO	1.0	Permissible ratio of actual load effect to design strength.
BEAM	1.0	0.0 = design at ends and those locations specified by SECTION command. 1.0 = design at ends and at every $1/12^{th}$ point along member length. (Default)
PROFILE	None	Used in member selection. See section 5.47.1 for details.
AXIS	1	1 - Design single angles for bending based on principal axis. 2 - Design single angles for bending based on geometric axis.
FLX	1	1 – Single Angle Member is not fully braced against lateral torsional buckling. 2 - Single Angle Member is fully braced against lateral torsional buckling. 3 - Single Angle Member is braced against lateral torsional buckling at the point of maximum moment.

Table 2.5 - AISC LRFD (2nd and 3rd editions) Design Parameters

Parameter Name	Default Value	Description
MAIN	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check Any value greater than 1 = Allowable KL/r in compression
TMAIN	300	Any value greater than 1 = Allowable KL/r in tension

*Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if SET Z UP is used).

Note: For a description of the deflection check parameters DFF, DJ1, DJ2 and CAN, see the Notes section of Table 2.1 of this manual.

The STIFF parameter represents the term “a” as defined in Section F2, page 6-113 of the LRFD 2nd edition manual.

** Non-default values of CB must be re-entered before every subsequent CHECK CODE or SELECT command.

2.14.11 Code Checking and Member Selection

Both code checking and member selection options are available in STAAD LRFD implementation. For general information on these options, refer to Sections 2.5 and 2.6. For information on specification of these commands, refer to Section 5.47.1.

Example for the LRFD-2001 code

```
UNIT KIP INCH
PARAMETER
CODE LRFD

or

CODE LRFD3
FYLD 50 ALL
UNT 72 MEMBER 1 TO 10
UNB 72 MEMB 1 TO 10
MAIN 1.0 MEMB 17 20
SELECT MEMB 30 TO 40
CHECK CODE MEMB 1 TO 30
```

Example for the LRFD-1994 code

```
UNIT KIP INCH
PARAMETER
CODE LRFD2
FYLD 50 ALL
UNT 72 MEMBER 1 TO 10
UNB 72 MEMB 1 TO 10
MAIN 1.0 MEMB 17 20
SELECT MEMB 30 TO 40
CHECK CODE MEMB 1 TO 30
```

2.14.12 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format. A detailed discussion of the format is provided in Section 2.11. Following exceptions may be noted: CRITICAL COND refers to the section of the LRFD specifications which governed the design.

If the TRACK is set to 1.0, member design strengths will be printed out.

2.14.13 Composite Beam Design per the American LRFD 3rd edition code

The design of composite beams per the 3rd edition of the American LRFD code has been implemented. The salient points of this feature are as follows:

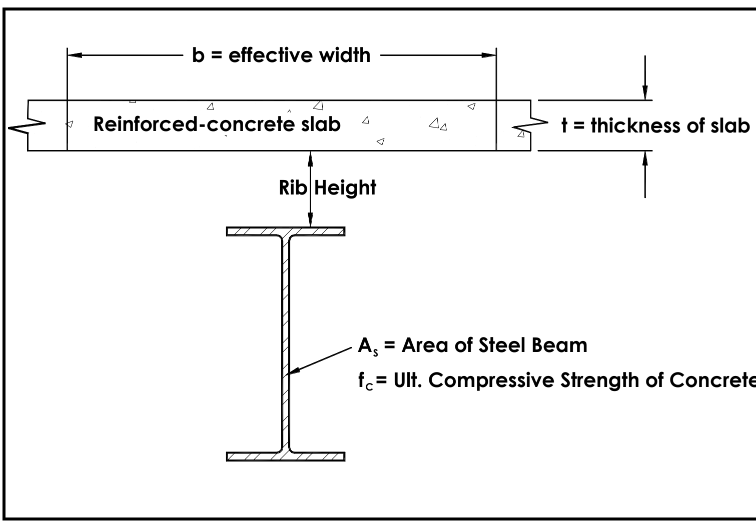


Figure 2.6

Theoretical basis:

1. Find the maximum compressive force carried by concrete as:

$$0.85\ f_c\ .\ b\ .\ t$$

2. Find the maximum tensile force carried by the steel beam as:

$$A_s\ .\ f_y$$

Tensile strength of concrete is ignored.

3. If step 1 produces a higher value than step 2, plastic neutral axis (PNA) is in the slab. Else, it is in the steel beam.

CASE 1: PNA IN SLAB

Find the depth of PNA below the top of slab as:

$$0.85\ f_c\ .\ b\ .\ a\ .=A_s\ .\ f_y$$

$$a=\frac{A_s\ .f_y}{0.85\,f_c\ .b}$$

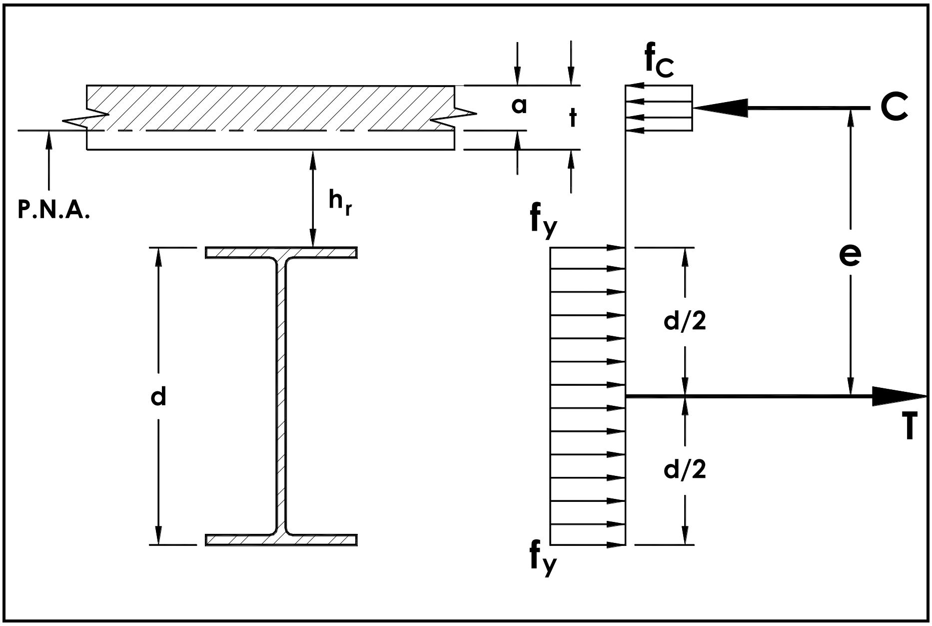


Figure 2.7

$$\text{Lever arm } e = \frac{d}{2} + h_r + t - \frac{a}{2}$$

$$\text{Moment capacity} = \phi_b \left(A_s \, . \, f_y\right). e$$

CASE 2: PNA IN STEEL BEAM

Define:

C_s = Compressive force in slab = 0.85 . f_c . b . t
C_b = Compressive force in steel beam
T_b = Tensile force in steel beam

$C_s + C_b = T_b$

Since magnitude of C_b + magnitude of $T_b = A_s \,.\, f_y$

Substituting for T_b as ($A_s \,.\, f_y - C_b$), we get:

$C_s + C_b = A_s \,.\, f_y - C_b$

$C_b = (A_s \,.\, f_y - C_s) \times 0.5$

Determine whether the PNA is within the top flange of steel beam, or inside its web.

C_f = Maximum compressive force carried by flange
$= A_f \,.\, f_y$

where, A_f = Area of flange

If $C_f >= C_b$, PNA is in the flange.

If $C_f < C_b$, PNA is in the web.

CASE 2A: PNA IN FLANGE OF STEEL BEAM

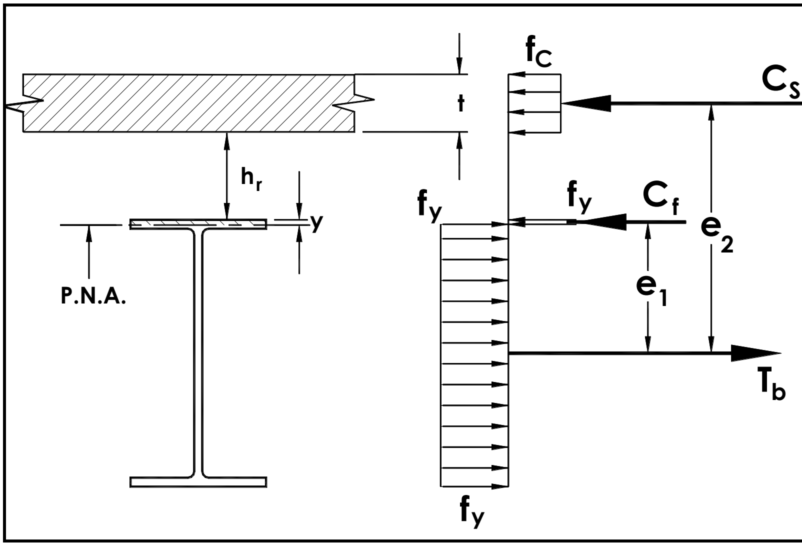


Figure 2.8

Calculate:

$$y = C_f / (b_f \, . f_y)$$

where, b_f = width of flange

The point of action of the tensile force is the centroid of the steel area below the PNA. After finding that point, e_1 and e_2 can be calculated.

$$\text{Moment Capacity} = \phi_b \left(C_f \, . e_1 + C_s \, . e_2 \right)$$

CASE 2B: PNA IN WEB OF STEEL BEAM

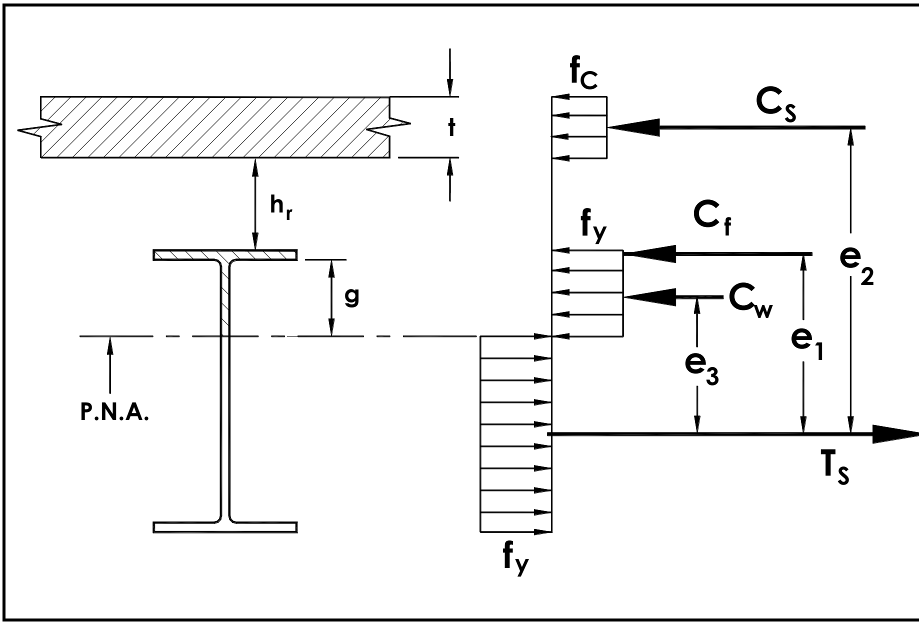


Figure 2.9

C_f = Compressive force in flange = $A_f \,.\, f_y$

C_w = Compressive force in web = $C_b - C_f$

$g = C_w / (t_w \,.\, f_y)$ where, t_w = thickness of web

Point of action of the tensile force is the centroid of the steel area below the PNA. After finding that point, e_1, e_2 and e_3 can be calculated.

Moment Capacity = $\phi_b \left(C_s \,.\, e_2 + C_f \,.\, e_1 + C_w \,.\, e_3\right)$

$\boxed{\text{Utilization Ratio = Applied Moment / Moment Capacity}}$

Notes

1. Rib Height is the distance from top of flange of steel beam to lower surface of concrete.
2. If the slab is flush on top of the steel beam, set the Rib Height to zero.

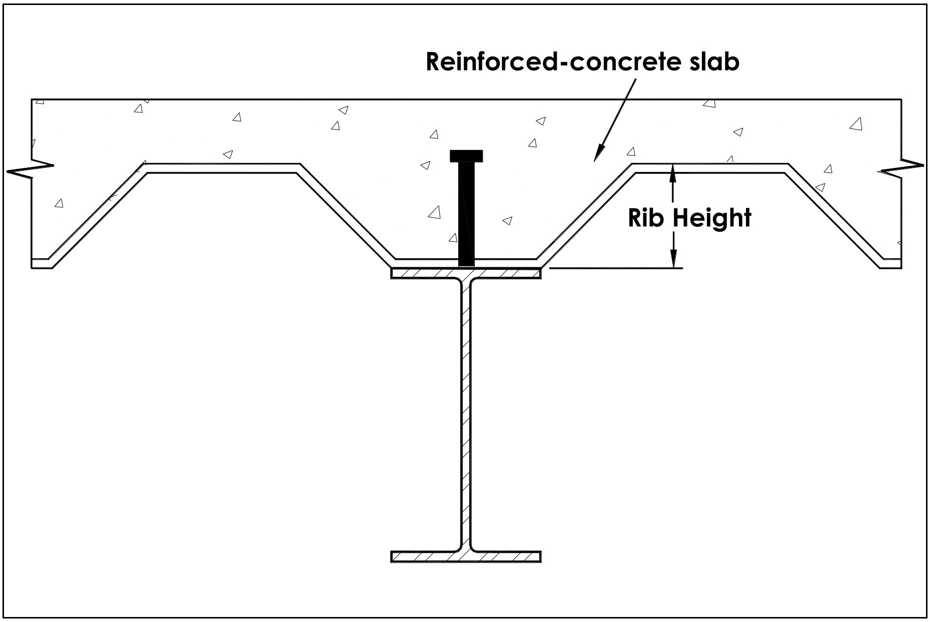


Figure 2.10

3. For moments which cause tension in the slab (called positive moments in STAAD convention), design of the beam is presently not carried out.
4. Shear connectors are presently not designed.
5. Member selection is presently not carried out.
6. In order to design a member as a composite beam, the member property specification during the analysis phase of the data must contain the "CM" attribute. See section 5.20.1 of the STAAD Technical Reference manual for details.

TABLE 2.6 – COMPOSITE BEAM DESIGN PARAMETERS FOR AISC-LRFD		
Name	**Default value**	**Description**
RBH	0.0 inches	Rib Height
EFFW	Value used in analysis	Effective width of slab
FPC	Value used in analysis	Ultimate compressive strength of concrete

Example

```
STAAD SPACE
...
...
MEMBER PROPERTY
1 TA CM W12X26 CT 6.0 FC 4.0 CW 40.0
...
...
PERFORM ANALYSIS
...
...
PARAMETER
CODE LRFD
RBH 5.0 MEMB 1
CHECK CODE MEMB 1
FINISH
```

2.15 Design per American Cold Formed Steel Code

General

Provisions of the AISI Specification for the Design of Cold-Formed Steel Structural Members, 1996 Edition have been implemented. The program allows design of single (non-composite) members in tension, compression, bending, shear, as well as their combinations using the LRFD Method. For flexural members, the Nominal Section Strength is calculated on the basis of initiation of yielding in the effective section (Procedure I). Strength increase from Cold Work of Forming is a user selectable option.

Cross-Sectional Properties

The user specifies the geometry of the cross-section by choosing one of the section shape designations from the STAAD Steel Tables for cold-formed sections, which mirror the Gross Section Property Tables published in the "Cold-Formed Steel Design Manual", AISI, 1996 Edition.

The Tables are currently available for the following shapes:

- Channel with Lips
- Channel without Lips
- Angle with Lips
- Angle without Lips
- Z with Lips
- Z without Lips
- Hat

Shape selection may be done using the member property pages of the graphical user interface (GUI) or by specifying the section designation symbol in the input file. Details of the latter are explained below.

The AISI Steel Section Library:

The command-line syntax for assigning steel sections from the AISI library is as explained below :

C-Section With Lips

```
20 TO 30 TA ST 14CS3.75X135
33 36 TA ST 12CS1.625X102
42 43 TA ST 4CS4X060
```

C-Section Without Lips

```
50 TO 60 TA ST 10CU1.25X071
32 33 TA ST 3CU1.25X057
21 28 TA ST 1.5CU1.25X035
```

Z-Section With Lips

```
1 3 4 TA ST 12ZS3.25X135
33 45 TA ST 10ZS3X060
12 13 TA ST 6ZS2X048
```

Z-Section Without Lips

```
2 3 TA ST 8ZU1.25X105
4 5 TA ST 4ZU1.25X036
6 7 TA ST 1.5ZU1.25X048
```

Equal Leg Angles With Lips

```
8 9 TA ST 4LS4X105
10 11 TA ST 3LS3X060
12 13 TA ST 2LS2X075
```

Equal Leg Angles Without Lips

```
1 5 TA ST 4LU4X135
7 8 TA ST 2.5LU2.5X105
4 9 TA ST 2LU2X060
```

Hat Sections Without Lips

```
4 8 TA ST 10HU5X075
5 6 TA ST 6HU9X105
1 7 TA ST 3HU4.5X135
```

Current Limitations :

At the present time, only standard single sections are available for specification. Options such as double angles, double channels, and user provided sections including pipes and tubes will be available at a later date. Additionally, combination sections, such as an angle placed on top of a channel, or a plate welded to the top, bottom or side of one of the above shapes, are not available at this time.

STAAD.Pro uses unreduced section properties in the structure analysis stage. Both unreduced and effective section properties are used in the design stage, as applicable.

Design Procedure

The following two design modes are available:

1. Code Checking

   The program compares the resistance of members with the applied load effects, in accordance with the LRFD Method of the AISI code. Code checking is carried out for locations specified by the user via the SECTION command or the BEAM parameter. The results are presented in a form of a PASS/FAIL identifier and a RATIO of load effect to resistance for each member checked. The user may choose the degree of detail in the output data by setting the TRACK parameter.

2. Member Selection

   The user may request that the program search the cold formed steel shapes database (AISI standard sections) for alternative members that pass the code check and meet the least weight criterion. In addition, a minimum and/or maximum acceptable depth of the member may be specified. The program will then evaluate all database sections of the type initially specified (i.e., channel, angle, etc.) and, if a suitable replacement is found, present design results for that section. If no section satisfying the depth restrictions or lighter than the initial one can be found, the program leaves the member unchanged, regardless of whether it passes the code check or not.

The program calculates effective section properties in accordance with the following Sections:

- B2.1, Uniformly Compressed Stiffened Elements
- B2.3, Webs and Stiffened Elements with Stress Gradient
- B3.1, Uniformly Compressed Unstiffened Elements
- B3.2, Unstiffened Elements and Edge Stiffeners with Stress Gradient
- B4.2, Uniformly Compressed Elements with an Edge Stiffener

Cross-sectional properties of members are checked for compliance with the following Sections:

- B1.1(a), Maximum Flat-Width-to-Thickness Ratios, and
- B1.2, Maximum Web Depth-to-Thickness Ratio

The program checks member strength in accordance with Chapter C of the specification as follows:

1. Tension Members.

   Resistance is calculated in accordance with Section C2.

2. Flexural Members.

   a) C3.1, Strength for bending only:

      - C3.1.1, Nominal Section Strength, Procedure I
      - C3.1.2, Lateral Buckling Strength

   b) C3.2, Strength for Shear Only

   c) C3.3, Strength for Combined Bending and Shear

3. Concentrically Loaded Compression Members.

   a) C4.1, Sections not subject to Torsional or Torsional-Flexural Buckling, and

   b) C4.2, Doubly or Singly Symmetric sections subject to Torsional or Torsional-Flexural Buckling.

4. Combined Axial Load and Bending.

   a) C5.1, Combined Tensile Axial Load and Bending, and

   b) C5.2, Combined Compressive Axial Load and Bending.

The following table contains the input parameters for specifying values of design variables and selection of design options.

Table 2.7 - AISI Cold Formed Steel Design Parameters

Parameter Name	Default Value	Description
BEAM	1.0	When this parameter is set to 1.0 (default), the adequacy of the member is determined by checking a total of 13 equally spaced locations along the length of the member. If the BEAM value is 0.0, the 13 location check is not conducted, and instead, checking is done only at the locations specified by the SECTION command (See STAAD manual for details). If neither the BEAM parameter nor any SECTION command is specified, STAAD will terminate the run and ask the user to provide one of those 2 commands. This rule is not enforced for TRUSS members.
CMZ	1.0	End moment coefficient for bending about Z axis. See AISI C5.2.2. Used for combined axial load and bending design. Values range from 0.4 to 1.0.
CMY	0.0	End moment coefficient for bending about Y axis. See AISI C5.2.2. Used for combined axial load and bending design. Values range from 0.4 to 1.0.
CWY	0	Specifies whether the cold work of forming strengthening effect should be included in resistance computation. See AISI A7.2. Values: 0 – effect should not be included 1 – effect should be included

Table 2.7 - AISI Cold Formed Steel Design Parameters

Parameter Name	Default Value	Description
DMAX	1000.0	Maximum depth permissible for the section during member selection. This value must be provided in the current units.
DMIN	0.0	Minimum depth required for the section during member selection. This value must be provided in the current units.
FLX	1	Specifies whether torsional-flexural buckling restraint is provided or is not necessary for the member. See AISI C4.1 Values: 0 – Section subject to torsional flexural buckling and restraint not provided 1 – restraint provided or unnecessary
FU	58 ksi	Ultimate tensile strength of steel in current units.
FYLD	36 ksi	Yield strength of steel in current units.
KT	1.0	Effective length factor for torsional buckling. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.
KY	1.0	Effective length factor for overall column buckling about the local Y-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.

Table 2.7 - AISI Cold Formed Steel Design Parameters		
Parameter Name	**Default Value**	**Description**
KZ	1.0	Effective length factor for overall column buckling in the local Z-axis. It is a fraction and is unit-less. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
LT	Member length	Unbraced length for twisting. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from torsional buckling) to any user specified large value. It is used to compute the KL/R ratio for twisting for determining the capacity in axial compression.
LY	Member length	Effective length for overall column buckling in the local Y-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the capacity in axial compression.
LZ	Member length	Effective length for overall column buckling in the local Z-axis. It is input in the current units of length. Values can range from 0.01 (for a column completely prevented from buckling) to any user specified large value. It is used to compute the KL/R ratio for determining the allowable stress in axial compression.
NSF	1.0	Net section factor for tension members, See AISI C2.

Table 2.7 - AISI Cold Formed Steel Design Parameters		
Parameter Name	**Default Value**	**Description**
STIFF	Member length	Spacing in the longitudinal direction of shear stiffeners for reinforced web elements. It is input in the current units of length. See section AISI C3.2
TRACK	0	This parameter is used to control the level of detail in which the design output is reported in the output file. The allowable values are: 0 - Prints only the member number, section name, ratio, and PASS/FAIL status. 1 - Prints the design summary in addition to that printed by TRACK 1 2 - Prints member and material properties in addition to that printed by TRACK 2.
TSA	1	Specifies whether bearing and intermediate transverse stiffeners are present. If true, the program uses the more liberal set of interaction equations in AISI C3.3.2. Values: 0 – Beams with un-reinforced webs 1 – Beams with transverse web stiffeners

2.16 Castellated Beams

STAAD.Pro comes with the non-composite castellated beam tables supplied by the steel products manufacturer SMI Steel Products. Details of the manufacture and design of these sections may be found at

http://www.smisteelproducts.com/English/About/design.html

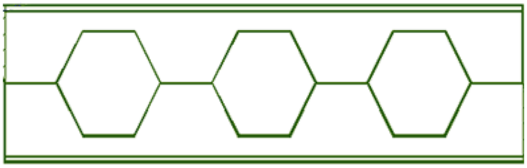


Figure 2.11

According to the manufacturer, castellated beams are manufactured by cutting a wide flange along the web in a zig-zag pattern, offsetting the two halves, and welding the two halves together, as shown in the next figure. As a result, the underlying steel section is a wide flange (W shapes) in the AISC table or a B shape. STAAD currently supports only the ones derived from W shapes.

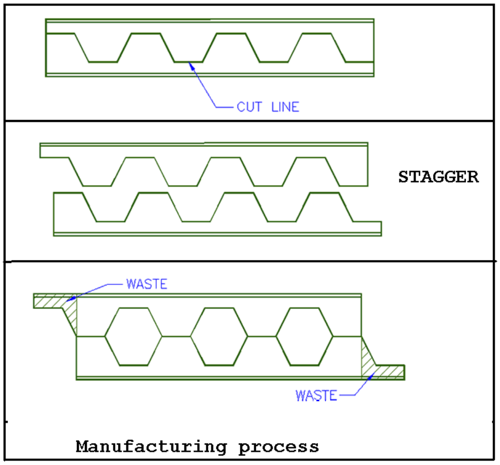


Figure 2.12

Analysis and Design criteria

The local axis system (local X, local Y and local Z) of a castellated beam is identical to that for a wide flange, and is shown in section 1.5.2 of the Technical Reference manual.

Users have to recognize that there are two basic issues to be understood with regard to these members a) analysis b) steel design.

We first explain the design issues because only then will their relationship with the analysis issues become apparent. Design of a castellated beam is done only for FY (shear along the web) and MZ (moment about the major axis which is the Z axis). If at the start of the design process, the program detects that the beam has axial force (FX), shear along local-Z (FZ), torsion (MX) or moment about the minor axis (MY), design of that member will be terminated.

And now we come to how these design limitations have a bearing on the analysis issues. If the user intends to design these members, as a result of the above restrictions, he/she must model it in such a way that none of the 4 unacceptable degrees of freedom end up with a non-zero value anywhere along the length of the member. That means, if the member ends are defined as supports, the support conditions must be defined with the above in mind. Similarly, if the castellated member is attached to other members, its end conditions (MEMBER RELEASES) must be modeled taking the above facts into consideration.

The design limitations also have a bearing on the type of loads that are applied to the member. Loads which cause any of the above-mentioned four degrees of freedom to end up with a non-zero value will cause the member design to be terminated.

However, <u>if the user wishes to only analyze the structure, and is not interested in performing a steel design, the above described restrictions for supports, member end conditions or loading are not applicable.</u>

The design method is the allowable stress method, using mainly the rules stated in the AISC ASD 9th edition code. Only code checking is currently available for castellated beams. Member selection is not.

Note:

STAAD does not multiply the analysis moment by 1.7 for ASD method. It is upto the user to multiply the dead and live loads by 1.7 in load combination and using this load case in design. The reason is that if program internally multiplies the analysis moment by 1.7 for ASD method (it is 1.2 for dead and 1.6 for live loads for LRFD method) then user must ensure that the analysis moment is the unfactored moment. If by mistake user uses factor 1.7 during load combination and designs I-beam with web opening with this load, the program will further increase the load by 1.7.

Hence, it has been intentionally kept at the hands of the user to use whatever load factor he wants to use before designing I-beam with opening with that factored load case.

Design parameters:

The following table contains a list of parameters and their default values.

Table 2.8 American Castellated Beam Design Parameters

Parameter	Default Value	Description
SOPEN	1.5e + b is the minimum allowable value. Any user-specified value higher than or equal to this minimum will be used by the program. “e” and “b” are as described in the next figure.	Distance from the start of the member to the center of the first hole.

Table 2.8 American Castellated Beam Design Parameters

Parameter	Default Value	Description
EOPEN	1.5e + b is the minimum allowable value. Any user-specified value higher than or equal to this minimum will be used by the program. “e” and “b” are as described in the next figure.	Distance from the center of the last hole to the end of the member.
UNL	Member length	Unsupported length of compression flange for calculating allowable bending stress.
FYLD	36 ksi	Yield Stress of Steel
CB	1.0	Cb value used for computing the allowable bending stress per Chapter F of AISC specifications.
CMZ	0.85	Cm value in local Z axis. Used in the interaction equations in Chapter H of AISC specifications.
TRACK	0	Parameter used to control the level of description of design output. Permissible values are 0 and 1 .
RATIO	1.0	Permissible maximum ratio of actual load to section capacity. Any input value will be used to change the right hand side of governing interaction equations in Chapter H and elsewhere.

References:

STAAD.Pro's design of castellated beams is based on the information gathered from the following sources:

a. Design of Welded Structures -- Omer W. Blodget, published by The James Lincoln Arc Welding Foundation, pages 4.7-8 and 4.7-9

b. AISC 9th edition manual – Allowable stress design

c. ASCE Journal of Structural Engineering 124:10, October 1998 – castellated beam web buckling in shear, R.G. Redwood and S. Demirdjian

Design procedure:

Cross-section checks:

The first check that is carried out is a verification whether the member properties satisfy certain basic requirements. If the member fails these checks, the remainder of the checks are not performed.

The cross section checks are the following:

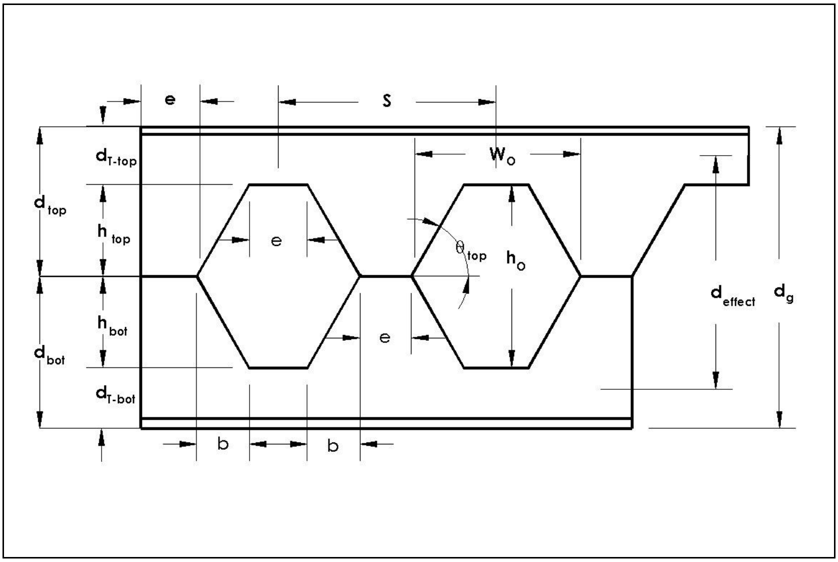


Figure 2.13

1. Web Post Width (e) should be at least 3.0 inches
2. Tee Depth (d_T-top and d_T-bot) should be greater than the thickness of flange plus one inch.
3. Angle θ should be between 45 and 70 degrees.
4. In order for the program to determine the number of holes which are admissible for the beam, the parameters SOPEN and EOPEN need to be assigned. In the figure above, there is a term shown as S. This value is part of the section tables supplied with STAAD.Pro, so it retrieves that value from there. It then computes the number of holes, and the remainder of the terms shown in the above diagram.
5. SOPEN and EOPEN (see the parameter table shown earlier) have to be at least 1.5e + b, with “e” and “b” as shown in the earlier figure. If the user inputs a value less than these minima, the minimum values are used.

Checking the member for adequacy in carrying the applied loading:

This consists of five different checks:

1. Global Bending
2. Vierendeel Bending
3. Horizontal Shear
4. Vertical Shear
5. Web Post Buckling

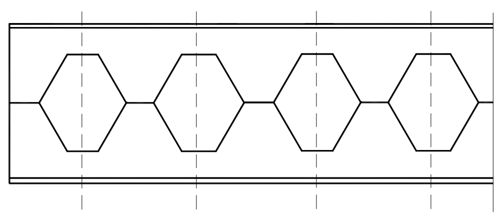
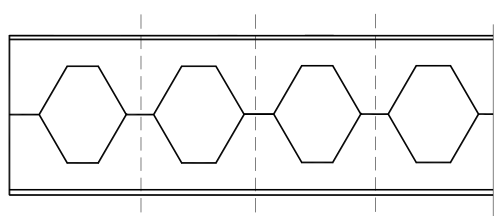
Design for	**Section considered in the design (shown with the vertical dotted lines)**
Vierendeel Bending	
Global Bending Vertical Shear Horizontal Shear Web Post Buckling	

Figure 2.14

1. <u>Global Bending</u>:

   Global bending check is done at the web post section. This is the region of the member where the full cross section is active, without interference of the holes.

   The actual bending stress is computed at the middle of the web post location and is obtained by dividing the moment by the section modulus of the full section.

   For computing the allowable bending stress, the compactness of the section is first determined in accordance with Table B5.1 in the Chapter B of the AISC 9th edition specifications. The rules applicable to I-shaped sections are used for this. Following this, the allowable bending stress is computed per chapter F of the same.

   The ratio is computed by dividing the actual stress by the allowable stress.

2. <u>Vierendeel Bending</u>:

   This is checked at the middle of the hole locations. The effective cross section at these locations is a Tee. The overall moment (Mz) at the span point corresponding to the middle of the hole is converted to an axial force and a moment on the Tee.

   The actual stress is computed at the top and bottom of each Tee section.

   fa = M / (deffect * At)

   where At is the area of the Tee section

   fb = V * e * a / (2 S)

   where a is the area factor. For the top Tee section, a = Area of Top Tee / (Area of Top Tee + Area of Bottom Tee)

Allowable Stresses for vierendeel bending:

- Axial Stress: The allowable axial stress is computed as per the Chapter E of the AISC specifications. The unsupported length for column buckling is equal to e.

- Bending Stress: The allowable bending stress is computed for the top and bottom Tee section as per the Chapter F of the AISC manual.

The axial stress plus bending stress is computed at the top and bottom of each tee section. If it is compressive then it is checked against equations H1-1 and H1-2 of Chapter H of the AISC manual. If it is tensile then it is checked against equation H2-1.

3. Horizontal Shear:

Allowable Shear stress is computed as 0.4 Fy.

Actual Stress: Please refer to pages 4.7-8 and 4.7-9 of the reference book on welded structures mentioned under Item (a) earlier.

4. Vertical Shear:

Allowable Shear stress is computed as 0.4 Fy.

The actual shear stress is computed at the middle of the web post location.

5. Web Post buckling:

Please refer to pages 1202-1207 of the ASCE journal mentioned under Item (c) earlier.

The command syntax in the STAAD input file for assigning castellated beams is:

MEMBER PROPERTY AMERICAN

Member-list TABLE ST section-name

Example

MEMBER PROPERTY AMERICAN
2 TABLE ST CB12x28

Assigning Design parameters

Under the PARAMETERS block on input, the code name must be specified as:

CODE AISC CASTELLATED

Example

```
PARAMETER
CODE AISC CASTELLATED
UNL 0.01 MEMB 25 31
FYLD 50 MEMB 25 31
SOPEN 11.124 MEMB 25 31
……
CHECK CODE MEMB 25 31
```

Steel Design Output:

A typical TRACK 2 level output page from the STAAD output file is as shown.

```
             STAAD.PRO CODE CHECKING - (AISC CASTELLATED)
             ********************************************

ALL UNITS ARE - Kip and Inches (UNLESS OTHERWISE NOTED)

 Castellated Steel Design for Member      2
 ==========================================

 Section Name  ST  NCB27X40

 Design Results
 --------------
 Design Status: Pass

 Check for Global Bending
 -------------------------
 Load =     3        Section =    260.874
 Fy =       0.76          Mz =   -3034.03
 Fb top =      33.00   Fb Bot =     33.00
 fb  =      26.95
 Ratio =  0.82

 Check for Vierendeel Bending
 ----------------------------
 Load =     3        Section =    196.124
 Fy =       6.16          Mz =   -2810.10
 Fa =      29.91          Fb =      30.00
 Klr =       1.46         Fe =   69606.88
 fa =       5.63          fb =      12.12
 Ratio =  0.97

 Check for Vertical Shear
 -------------------------
 Load =     3        Section =      0.000
 Fy =      22.50          Mz =       0.00
 Fv =      20.00          fv =       2.62
 Ratio =  0.13

 Check For Horizontal Shear ( Web Post )
 ---------------------------------------
 Load =     3        Section =    519.874
 Fy =     -20.82          Mz =    -435.96
 Fv =      20.00          fv =      14.73
 Ratio =  0.74

 Check for Web Post Buckling
 ---------------------------
 Load =     3        Section =    519.874
 Fy =     -20.82          Mz =    -435.96
 Mallow =     141.32    Mact =     189.47
 Ratio =  0.75
```

Figure 2.15

Viewing the design results in the graphical screens:

After the analysis and design is completed, double click on the castellated member. This feature, known as member query, brings up a dialog box, one of whose tabs will be **Castellated Beam Design** as shown.

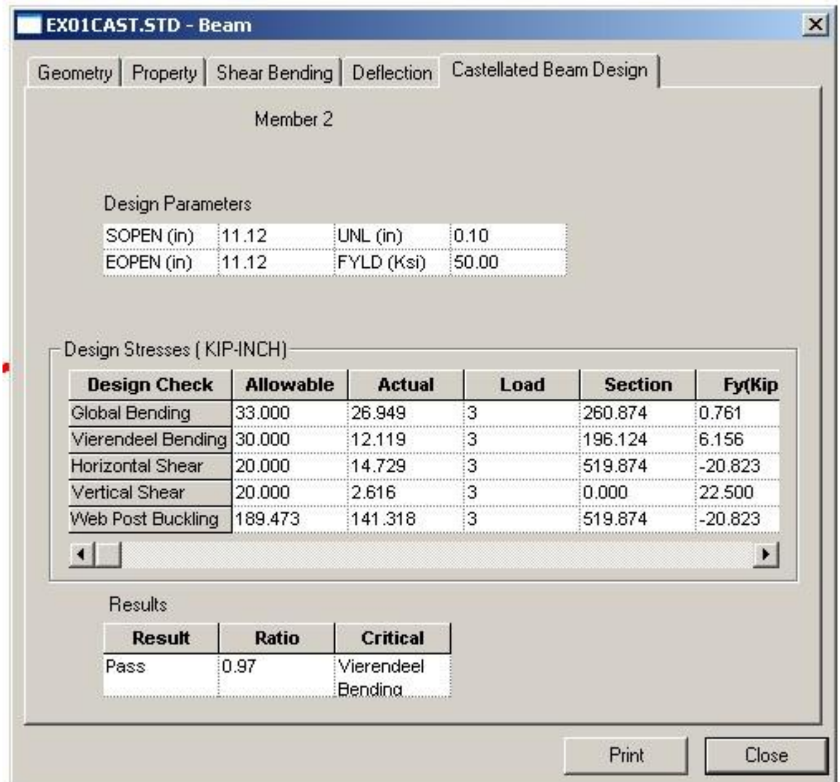


Figure 2.16

Example Problem :

```
STAAD PLANE EXAMPLE PROBLEM FOR
*CASTELLATED BEAM DESIGN

UNIT FT KIP
JOINT COORDINATES
1 0. 0. ; 2 45 0
3 0 15; 4 45 15

MEMBER INCIDENCE
1 1 3; 2 3 4; 3 4 2

MEMBER PROPERTY AMERICAN
2 TA ST CB27x40
1 3 TA ST W21X50

UNIT INCH
CONSTANTS
E STEEL ALL
DEN STEEL ALL
POISSON STEEL ALL

MEMBER RELEASE
2 START MX MY MZ
2 END MY MZ

UNIT FT
SUPPORT
1 2 FIXED

LOADING 1 DEAD AND LIVE LOAD
MEMB LOAD
2 UNI Y -0.4
```

```
LOADING 2  WIND FROM LEFT
MEMBER LOAD
2 UNI Y -0.6

LOAD COMB 3
1 1.0 2 1.0

PERFORM ANALYSIS
LOAD LIST 3
PRINT MEMBER FORCES
PRINT SUPPORT REACTION

UNIT KIP INCH
PARAMETER
CODE AISC CASTELLATED
UNL 0.01 MEMB 2
FYLD 50 MEMB 2
CMZ 0.85 MEMB 2
CB 1.1 MEMB 2
TRACK 2.0 ALL
SOPEN 11.124 MEMB 2
EOPEN 11.124 MEMB 2
CHECK CODE MEMB 2
FINISH
```

2.17 Steel Design per the AISC 360-05 Steel Design Specification

The specifications of ANSI AISC 360-05 have been implemented with effect from STAAD.Pro 2006. These specifications are published as part of the AISC "Steel Construction Manual", 13th Edition. Since the ASD and the LRFD method are both addressed in those specifications, they are referred to as UNIFIED.

The command to access those specifications is:

CODE AISC UNIFIED

2.17.1 General Comments on Design as per AISC 360-05 specifications (Unified Code)

Both methods are implemented in STAAD. The selection of the method can be done through the METHOD parameter explained in the parameter list.

Design for Strength Using Load and Resistance Factor Design (LRFD)

Design according to the provisions for Load and Resistance Factor Design (LRFD) satisfies the requirements of the 360-05 Code Specification, when the design strength of each structural component equals or exceeds the required strength determined on the basis of the LRFD load combinations.

Design shall be performed in accordance with Equation B3-1 of the Code:

$Ru \leq \varphi Rn$

Where,

Ru = Required Strength (LRFD)
Rn = Nominal Strength,
φ = Resistance Factor,
φRn = Design Strength

Design for Strength Using Allowable Strength Design (ASD)

Design according to the provisions for Allowable Strength Design (ASD) satisfies the requirements of the Unified Code Specification when the allowable strength of each structural component equals or exceeds the required strength determined on the basis of the ASD load combinations.
Design shall be performed in accordance with Equation B3-2 of the Code:

$Ra \leq Rn/\Omega$

Where,

Ra = Required Strength (ASD)
Rn = Nominal Strength,
Ω = Safety Factor,
Rn/Ω = Allowable Strength

2.17.2 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, non-compact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD is capable of determining the section classification for the standard shapes and design accordingly.

The Section Classification is done as per section B4 and Table B4.1, for Stiffened and Un-Stiffened Elements of a section.

2.17.3 Axial Tension

The criteria governing the capacity of tension members are based on:

- Tensile Yielding in Gross Section and
- Tensile Rupture of Net Section.

The limit state of yielding in the gross section is intended to prevent excessive elongation of the member, and the corresponding check is done as per section D2-(a) of the code.

The second limit state involves fracture at the section with the minimum effective net area, and the corresponding check is done as per section D2-(b) of the code.

STAAD calculates the tension capacity of a given member based on these two limit states.

The Net Section Area may be specified by the user through the use of the parameter NSF (see Table 2.2). The Effective Net Area of tension members can be determined by using the Shear Lag Factor. The user can also input the shear lag factor through the use of the parameter SLF.

2.17.4 Axial Compression

The Design Compressive Strength (LRFD), φc.Pn, and the Allowable Compressive Strength (ASD), Pn/Ωc, are calculated by the program.

The Nominal Compressive Strength, Pn, shall be the minimum value obtained according to the Limit States of -

- Flexural Buckling,
- Torsional Buckling, and
- Flexural-Torsional Buckling.

The Nominal Compressive Strength, Pn, for a particular member is calculated by STAAD according to the procedure outlined in Chapter E, section E3 to E5, of the unified code specifications. For slender elements, the procedure described in section E7 is used.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.

2.17.5 Flexural Design Strength

The Design Flexural Strength (LRFD), φb.Mn, and the Allowable Flexural Strength (ASD), Mn/Ωb, are being calculated by the program.

The Nominal Flexural Strength, Mn, is determined according to Sections F2 through F12 of unified code specifications, for different types of rolled sections.

The Nominal Flexural Strength of a member is determined by the limit states of Yielding (Y), Lateral-Torsional Buckling (LTB), Flange Local Buckling (FLB), Web Local Buckling (WLB), Tension Flange Yielding (TFY), Leg Local Buckling (LLB), and Local Buckling (LB).

The program internally calculates the Lateral-Torsional Buckling Modification Factor (Cb) for non-uniform moment diagrams when both ends of the unsupported segment are braced. The purpose of this factor is to account for the influence of the moment gradient on lateral-torsional buckling.

To specify the laterally unsupported length, the parameters UNT and UNB can be used, which by default take the value of the member length.

2.17.6 Design for Shear

The Design Shear Strength (LRFD), φv.Vn, and the Allowable Shear Strength (ASD), Vn/Ωv, are calculated by the program, as per section G2 of the unified code specifications.

The Nominal Shear Strength, Vn, of un-stiffened or stiffened webs, is calculated taking care of limit states of shear yielding and shear buckling. The sections G4 to G7 of the code specifications are used to evaluate Nominal Shear Strength, Vn for different types of rolled sections.

2.17.7 Design for Combined Forces and Torsion

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by formulas H1 and H3. These interaction formulas cover the general case of biaxial bending combined with axial force and torsion. They are also valid for uni-axial bending and axial force.

2.17.8 Design Parameters

Design per AISC 360-05 specifications is requested by using the CODE parameter. Other applicable parameters are summarized in the following table. These parameters communicate design decisions from the engineer to the program and thus allow the user to control the design process to suit an application's specific needs.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Table 2.9 AISC 360-05 Design Parameters

Parameter Name	Default Value	Description
METHOD	LRFD	Options: LRFD, ASD
KX	1.0	K value for Flexural-Torsional Buckling.
KY	1.0	K value in local-Y axis
KZ	1.0	K value in local-Z axis
FYLD	250 MPa	Yield Strength of Steel.
FU	400 MPa	Ultimate Tensile Strength of Steel
NSF	1.0	Net Section Factor for Tension Members
SLF	1.0	Shear Lag Factor for determination of Net Effective Area of Tension Members.
MAIN	200	Allowable Slenderness limit for Compression Members.
TMAIN	300	Allowable Slenderness limit for Tension Members.
UNT	Member length	Unsupported Length of the top* flange for calculating flexural strength. Will be used only if compression is in the top flange.

Table 2.9 AISC 360-05 Design Parameters		
Parameter Name	**Default Value**	**Description**
UNB	Member length	Unsupported Length of the bottom* flange for calculating flexural strength. Will be used only if compression is in the bottom flange.
RATIO	1.0	Permissible Ratio of Actual Load to Allowable Strength.
DMAX	1000.0 mm	Maximum Allowable Depth for Member Selection.
DMIN	0.0 mm	Minimum Allowable Depth for Member Selection.
PROFILE	None	Used in Member election. Refer Section 5.47.1 for details.
DFF	None	This is a mandatory parameter for Deflection Check, and defined by "DeflectionLength"/Maximum Allowable Local Deflection.
DJ1	Start Joint of the Member	Joint number denoting starting point for calculation of "Deflection Length".
DJ2	End Joint of the Member	Joint number denoting end point for calculation of "Deflection Length".
TRACK	0	0 = Suppress all member capacities 1 = Print all member capacities
LX	Member Length	Length for flexural-torsional buckling.
LY	Member Length	Length to calculate slenderness ratio for buckling about local Y-axis.
LZ	Member Length	Length to calculate slenderness ratio for buckling about local Z-axis.
CAN	0	0 = deflection check based on the principle that maximum deflection occurs within the span between DJ1 and DJ2. 1 = deflection check based on the principle that maximum deflection is of the cantilever type

Table 2.9 AISC 360-05 Design Parameters

Parameter Name	Default Value	Description
CB **	1.0	Coefficient C_b per Chapter F. If C_b is set to 0.0, it will be calculated by the program. Any other value will be directly used in design.
STP	1	Section Type to determine Fr (compressive residual stress in Flange) 1 = Rolled section (Fr-10 ksi) 2 = Welded section (Fr-16.5 ksi)
AXIS	1	1 - Design single angles for bending based on principal axis. 2 - Design single angles for bending based on geometric axis.

* Top and Bottom represent the positive and negative side of the local Y axis (local Z axis if SET Z UP is used).
** Non-default values of CB must be re-entered before every subsequent CHECK CODE or SELECT command.

Notes:

For a description of the deflection check parameters DFF, DJ1, DJ2 see the Notes section of Table 2.1 of this manual.

NSF is the Net Section Factor as used in most of the Steel Design Codes of STAAD. It is defined as the Ratio of 'Net cross section area' / 'Gross section area' for tension member design. The default value is 1.0. For the AISC 360-05 code, it is described in section D.3.2.

SLF is the Shear Lag Factor, as used in Section D.3.3 of the AISC 360-05 code. This factor is used to determine the Effective Net Area by multiplying this factor with Net Area of the cross section. Please refer to Table D3.1 of the 360-05 code for a list of acceptable SLF values. In STAAD, the default value for SLF is 1.0. The Effective Net Area is used to determine the Tensile Strength for Tensile rupture in the Net Section, as per equation D.2.2.

To summarize, the “Gross Area” (Ag) is multiplied by NSF to get the “Net Area” (An) of the section. The “Net Area” (An) is again multiplied by SLF to get the “Effective Net Area” (Ae) of the section.

2.17.9 Code Checking and Member Selection

Code Checking and Member Selection options are both available in the AISC Unified Code implementation in STAAD.Pro. For general information on these options, refer to Sections 2.5 and 2.6. For information on specification of these commands, refer to Section 5.47.1.

2.17.10 Tabulated Results of Steel Design

Results of Code Checking and Member Selection are presented in the output file.

The following details are presented on Code Checking of any member:

- Result of Code Checking (Pass / Fail) for the member Number.
- Critical Condition which governed the design and the corresponding Ratio and Location.
- Loads corresponding to the Critical Condition at the Critical Location.
- Section Classification
- Slenderness check report
- Section Capacities in Axial Tension, Axial Compression, Bending and Shear in both the directions.

Notes

Notes

American Concrete Design

Section 3

3.1 Design Operations

STAAD has the capabilities for performing concrete design. It will calculate the reinforcement needed for the specified concrete section. All the concrete design calculations are based on the current ACI 318.

Three versions of the code are currently implemented, the 1999, 2002 and 2005 edition.

To access the 1999 edition, specify the commands:

```
START CONCRETE DESIGN
CODE ACI 1999
```

To access the 2002 edition, specify the commands:

```
START CONCRETE DESIGN
CODE ACI 2002
```

To access the 2005 edition, specify the commands:

```
START CONCRETE DESIGN
CODE ACI

or

CODE ACI 2005
```

3.2 Section Types for Concrete Design

The following types of cross sections can be defined for concrete design.

For Beams	Prismatic (Rectangular & Square), Trapezoidal and T-shapes
For Columns	Prismatic (Rectangular, Square and Circular)
For Slabs Walls/Plates	Finite element with a specified thickness.

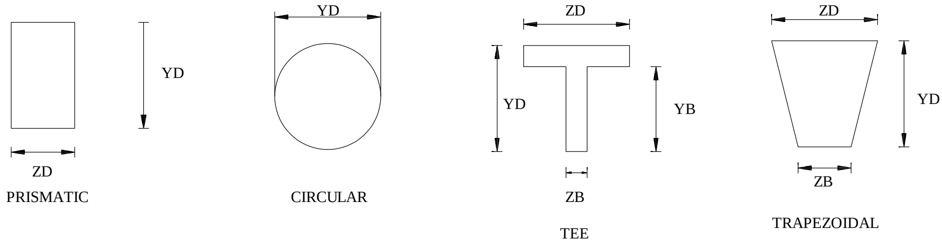


Figure 3.1

3.3 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the MEMBER PROPERTY command. The following example shows the required input:

```
UNIT INCH
MEMBER PROPERTY
1 3 TO 7 9 PRISM YD 18. ZD 12. IZ 2916 IY 1296
11 13 PR YD 12.
14 TO 16 PRIS YD 24. ZD 48. YB 18. ZB 12.
17 TO 19 PR YD 24. ZD 18. ZB 12.
```

In the above input, the first set of members are rectangular (18 inch depth and 12 inch width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 12 inch diameter. Note that no area (AX) is provided for these members. For concrete design, this property must not be provided. If shear areas and moments of inertias are not provided, the program calculates these values from YD and ZD. Notice that in the above example the IZ and IY values provided are actually 50% of the values calculated using YD and ZD. This is a conventional practice which takes into consideration revised section parameters due to cracking of section.

The third and the fourth set of members in the above example represent a T-shape and a TRAPEZOIDAL shape respectively. Depending on the properties (YD, ZD, YB, ZB, etc.) provided, the program will determine whether the section is rectangular, trapezoidal or T-shaped and the BEAM design will be done accordingly.

3.4 Design Parameters

The program contains a number of parameters which are needed to perform design by the ACI code. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 3.1 is a complete list of the available parameters and their default values.

See Section 5.51

Section 5.51.2 of this manual describes the commands required to provide these parameters in the input file. For example, the values of SFACE and EFACE (parameters that are used in shear design), the distances of the face of supports from the end nodes of a beam, are assigned values of zero by default but may be changed depending on the actual situation. Similarly, beams and columns are designed for moments directly obtained from the analyses without any magnification. The factor MMAG may be used for magnification of column moments. For beams, the user may generate load cases which contain loads magnified by the appropriate load factors.

Table 3.1 – ACI 318 Design Parameters

Parameter Name	Default Value	Description
FYMAIN	* 60,000 psi	Yield Stress for main reinforcing steel.
FYSEC	* 60,000 psi	Yield Stress for secondary steel.
FC	* 4,000 psi	Compressive Strength of Concrete.
CLT	* 1.5 inch	Clear cover for top reinforcement.
CLB	* 1.5 inch	Clear cover for bottom reinforcement.
CLS	* 1.5 inch	Clear cover for side reinforcement.
MINMAIN **	Number 4 bar	Minimum main reinforcement bar size. (Number 4 - 18)
MINSEC **	Number 4 bar	Minimum secondary reinforcement bar size.
MAXMAIN **	Number 18 bar	Maximum main reinforcement bar size
SFACE	*0.0	Face of support location at start of beam. If specified, the shear force at start is computed at a distance of SFACE+d from the start joint of the member.
EFACE	*0.0	Face of support location at end of beam. (Note: Both SFACE & EFACE are input as positive numbers) If specified, the shear force at end is computed at a distance of EFACE+d from the end joint of the member.
REINF	0.0	Tied Column. A value of 1.0 will mean spiral.
MMAG	1.0 (for columns only)	A factor by which the column design moments will be magnified.
WIDTH	*ZD	Width of concrete member. This value defaults to ZD as provided under MEMBER PROPERTIES.

Table 3.1 – ACI 318 Design Parameters

Parameter Name	Default Value	Description
DEPTH	*YD	Depth of concrete member. This value defaults to YD as provided under MEMBER PROPERTIES.
NSECTION ***	12	Number of equally-spaced sections to be considered in finding critical moments for beam design.
TRACK	0.0	**BEAM DESIGN:** With TRACK set to 0.0, Critical Moment will not be printed out with beam design report. A value of 1.0 will mean a print out. A value of 2.0 will print out required steel areas for all intermediate sections specified by NSECTION. **COLUMN DESIGN:** TRACK 0.0 prints out detailed design results. TRACK 1.0 prints out column interaction analysis results in addition to TRACK 0.0 output. TRACK 2.0 prints out a schematic interaction diagram and intermediate interaction values in addition to all of above.
RHOMN	0.01 (indicates 1%)	Minimum reinforcement (as the fractional number) required in a concrete column. Enter a value between 0.0 and 0.08 where 0.08 stands for 8%, the max allowed by the ACI code.

* These values must be provided in the current unit system being used.

** When using metric units for ACI design, provide values for these parameters in actual 'mm' units instead of the bar number. The following metric bar sizes are available: 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm and 60 mm.

*** NSECTION should have no member list since it applies to all members. The minimum value allowed is 12, the maximum is 20. If more than one NSECTION entered, then highest value is used.

3.5 Slenderness Effects and Analysis Consideration

Slenderness effects are extremely important in designing compression members. The ACI-318 code specifies two options by which the slenderness effect can be accommodated (Section 10.10 & 10.11 ACI-318). One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed-end moments, the effect of deflections on moments and forces, and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD has been written to allow the use of both the options. To perform the first type of analysis, use the command PDELTA ANALYSIS instead of PERFORM ANALYSIS. This analysis method will accommodate the requirements as specified in Section 10.10 of the ACI-318 Code, except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration. If it is desired, STAAD can also accommodate any arbitrary moment magnification factor (second option) as an input, in order to provide some safety due to the effects of the duration of loads.

Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, moment-magnification is based on empirical formula and assumptions on sidesway.

Considering all this information, it is our belief, that a PDELTA ANALYSIS, as performed by STAAD, is most appropriate for the design of concrete members. However, the user must note, to take advantage of this analysis, all combinations of loadings must be provided as primary load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a primary load case is revised during the pdelta analysis based on the deflections. Also note that the proper factored loads (such as 1.4 for DL etc.) should be provided by the user. STAAD does not factor the loads automatically.

3.6 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to locate the possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with an NSECTION parameter. All of these equally spaced sections are scanned to determine moment and shear envelopes.

3.6.1 Design for Flexure

Reinforcement for positive and negative moments are calculated on the basis of the section properties provided by the user. If the section dimensions are inadequate to carry the applied load, that is if the required reinforcement is greater than the maximum allowable for the cross section, the program reports that beam fails in maximum reinforcement. Effective depth is chosen as Total depth - (Clear cover + diameter of stirrup + half the dia. of main reinforcement), and a trial value is obtained by adopting proper bar sizes for the stirrups and main reinforcements. The relevant clauses in Sections 10.2 to 10.6 of ACI 318 are utilized to obtain the actual amount of steel required as well as the maximum allowable and minimum required steel. These values are reported as ROW, ROWMX and ROWMN in the output and can be printed using the parameter TRACK 1.0 (see Table 3.1). In addition, the maximum, minimum and actual bar spacing are also printed.

It is important to note that beams are designed for flexural moment MZ only. The moment MY is not considered in the flexural design.

3.6.2 Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear forces are calculated at a distance (d+SFACE) and (d+EFACE) away from the end nodes of the beam. SFACE and EFACE have default values of zero unless provided under parameters (see Table 3.1). Note that the value of the effective depth "d" used for this purpose is the update value and accounts for the actual c.g. of the main reinforcement calculated under flexural design. Clauses 11.1 through 11.6 of ACI 318 are used to calculate the reinforcement for shear forces and torsional moments. Based on the total stirrup reinforcement required, the size of bars, the spacing, the number of bars and the distance over which they are provided are calculated. Stirrups are always assumed to be 2-legged.

3.6.3 Design for Anchorage

In the output for flexural design, the anchorage details are also provided. At any particular level, the START and END coordinates of the layout of the main reinforcement is described along with the information whether anchorage in the form of a hook or continuation is required or not at these START and END points. Note that the coordinates of these START and END points are obtained after taking into account the anchorage requirements. Anchorage length is calculated on the basis of the Clauses described in Chapter 12 of ACI 318.

Example for beam design per the ACI 318-2002 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2002

or

CODE ACI
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN BEAM 1 7 10
END CONCRETE DESIGN
```

Example for beam design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN BEAM 1 7 10
END CONCRETE DESIGN
```

3.6.4 Description of Output for Beam Design

Table 3.2 shows a sample output of an actual reinforcement pattern developed by the program. The following annotations apply to the Table 3.2:

1) LEVEL Serial number of bar level which may contain one or more bar group

2)	HEIGHT	Height of bar level from the bottom of beam.
3)	BAR INFO	Reinforcement bar information specifying number of bars and bar size.
4)	FROM	Distance from the start of the beam to the start of the reinforcement bar.
5)	TO	Distance from the start of the beam to the end of the reinforcement bar.
6)	ANCHOR (STA/END)	States whether anchorage, either a hook or continuation, is needed at start (STA) or at the end.
7)	ROW	Actually required flexural reinforcement (As/bd) where b = width of cross section (ZD for rectangular and square section) and d = effective depth of cross section (YD - distance from extreme tension fiber to the c.g. of main reinforcement).
8)	ROWMN	Minimum required flexural reinforcement (Amin/bd)
9)	ROWMX	Maximum allowable flexural reinforcement (Amax/bd)
10)	SPACING	Distance between centers of adjacent bars of main reinforcement
11)	Vu	Factored shear force at section.
12)	Vc	Nominal shear strength provided by concrete.
13)	Vs	Nominal shear strength provided by shear reinforcement.
14)	Tu	Factored torsional moment at section
15)	Tc	Nominal torsional moment strength provided by concrete.
16)	Ts	Nominal torsional moment strength provided by torsion reinforcement.

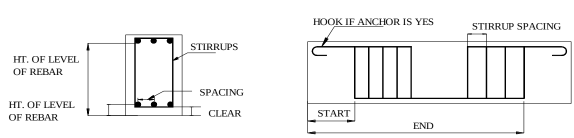


Figure 3.2

Table 3.2
(Actual Output from Design)

```
=====================================================================

      B E A M  N O.  14  D E S I G N  R E S U L T S -  FLEXURE

   LEN -  20.00FT.  FY - 60000.  FC - 4000.  SIZE - 15.00 X 21.00 INCHES

   LEVEL    HEIGHT     BAR INFO       FROM           TO         ANCHOR
          FT.   IN.               FT.     IN.     FT.     IN.   STA  END
   ----------------------------------------------------------------------

     1    0 + 2-5/8    3-NUM.9      2 + 4-5/8     20 + 0-0/0    NO   YES
   |----------------------------------------------------------------|
   |   CRITICAL POS MOMENT=    189.77 KIP-FT  AT 12.00 FT, LOAD   1 |
   |   REQD STEEL=  2.48 IN2, ROW=0.0090, ROWMX=0.0214 ROWMN=0.0033 |
   |   MAX/MIN/ACTUAL BAR SPACING=  7.50/ 2.26/ 4.94 INCH           |
   |   BASIC/REQD. DEVELOPMENT LENGTH = 37.95/ 31.42 INCH           |
   |----------------------------------------------------------------|

     2    1 + 6-1/8    4-NUM.11     0 + 0-0/0     18 +10-0/0    YES  NO
   |----------------------------------------------------------------|
   |   CRITICAL NEG MOMENT=    369.86 KIP-FT  AT  0.00 FT, LOAD   1 |
   |   REQD STEEL=  5.47 IN2, ROW=0.0198, ROWMX=0.0214 ROWMN=0.0033 |
   |   MAX/MIN/ACTUAL BAR SPACING=  7.50/ 2.82/ 3.20 INCH           |
   |   BASIC/REQD. DEVELOPMENT LENGTH = 59.20/153.91 INCH           |
   |----------------------------------------------------------------|

     3    1 + 6-3/8    3-NUM.6     16 + 8-1/4     20 + 0-0/0    NO   YES
   |----------------------------------------------------------------|

   |   CRITICAL NEG MOMENT=    105.87 KIP-FT  AT 20.00 FT, LOAD   1 |
   |   REQD STEEL=  1.31 IN2, ROW=0.0047, ROWMX=0.0214 ROWMN=0.0033 |
   |   MAX/MIN/ACTUAL BAR SPACING=  7.50/ 1.75/ 5.13 INCH           |
   |   BASIC/REQD. DEVELOPMENT LENGTH = 16.70/ 27.75 INCH           |
   |----------------------------------------------------------------|

      B E A M  N O.  14  D E S I G N  R E S U L T S - SHEAR

    AT START SUPPORT - Vu=   83.90 KIP  Vc=   35.10 KIP  Vs=   63.61 KIP
                         PROVIDE NUM. 4  BARS AT  7.0 IN. C/C FOR  108. IN.
    AT END   SUPPORT - Vu=   57.51 KIP  Vc=   35.10 KIP  Vs=   32.56 KIP
                         PROVIDE NUM. 4  BARS AT  9.3 IN. C/C FOR   84. IN.
```

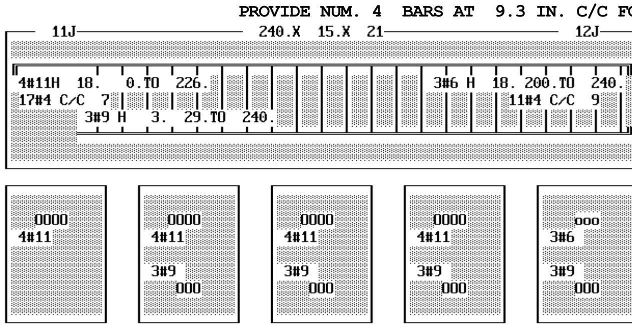


Figure 3.3

3.6.5 Cracked Moment of Inertia – ACI Beam Design

When beam design is done per ACI 318, STAAD will report the moment of inertia of the cracked section at the location where the design is performed. The cracked section properties are calculated in accordance with the equations shown below.

Rectangular sections

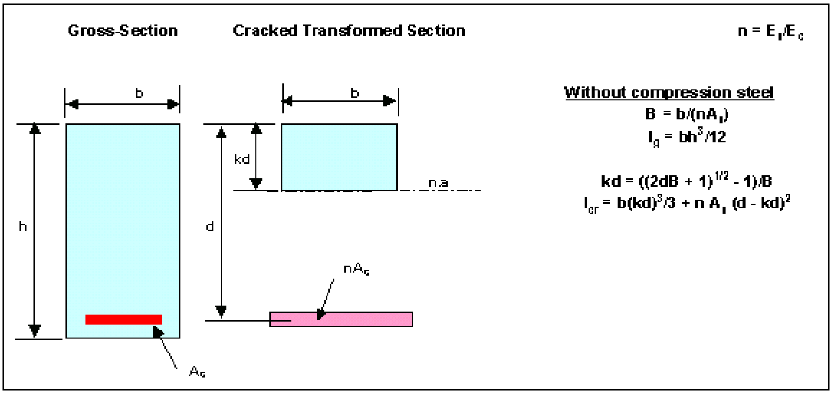


Figure 3.4

Tee shaped sections

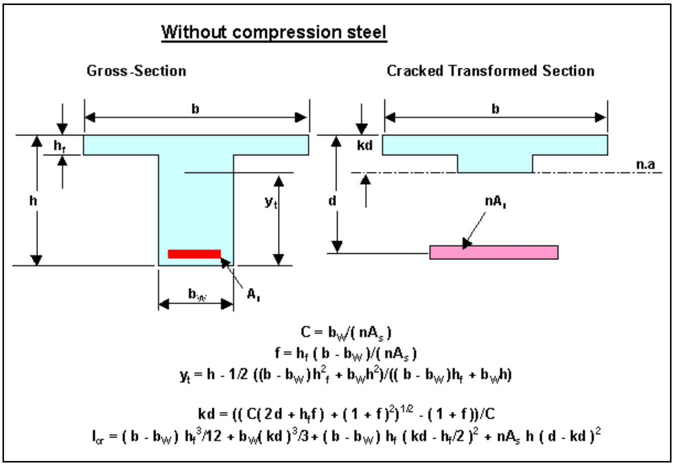


Figure 3.5

A typical screen from the STAAD beam design output, showing the cracked moment of inertia value, is shown below.

```
  2    1 + 6-1/8    4-NUM.11     0 + 0-0/0     16 + 2-0/0    YES  NO
 |----------------------------------------------------------------|
 |   CRITICAL NEG MOMENT=    371.02 KIP-FT  AT   0.00 FT, LOAD   1 |
 |   REQD STEEL=  5.38 IN2, ROW=0.0184, ROWMX=0.0214 ROWMN=0.0033 |
 |   MAX/MIN/ACTUAL BAR SPACING= 10.59/ 2.82/ 3.53 INCH           |
 |   BASIC/REQD. DEVELOPMENT LENGTH = 59.20/153.91 INCH           |
 |----------------------------------------------------------------|

  Cracked Moment of Inertia Iz at above location = 8238 inch^4
```

Figure 3.6

3.7 Column Design

Columns design in STAAD per the ACI code is performed for axial force and uniaxial as well as biaxial moments. All active loadings are checked to compute reinforcement. The loading which produces the largest amount of reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and circular sections, reinforcement is always assumed to be equally distributed on all faces. This means that the total number of bars for these sections will always be a multiple of four (4). If the MMAG parameter is specified, the column moments are multiplied by the MMAG value to arrive at the ultimate moments on the column. Since the ACI code no longer requires any minimum eccentricity conditions to be satisfied, such checks are not made.

Method used: Bresler Load Contour Method

Known Values: Pu, Muy, Muz, B, D, Clear cover, Fc, Fy
Ultimate Strain for concrete : 0.003

Steps involved:

1. Assume some reinforcement. Minimum reinforcement (1%) is a good amount to start with.

2. Find an approximate arrangement of bars for the assumed reinforcement.
3. Calculate PNMAX = 0.85 Po, where Po is the maximum axial load capacity of the section. Ensure that the actual nominal load on the column does not exceed PNMAX. If PNMAX is less than Pu/PHI, (PHI is the strength reduction factor) increase the reinforcement and repeat steps 2 and 3. If the reinforcement exceeds 8%, the column cannot be designed with its current dimensions.
4. For the assumed reinforcement, bar arrangement and axial load, find the uniaxial moment capacities of the column for the Y and the Z axes, independently. These values are referred to as MYCAP and MZCAP respectively.
5. Solve the interaction equation:

$$\left(\frac{M_{ny}}{M_{ycap}}\right)^{\alpha} + \left(\frac{M_{nz}}{M_{zcap}}\right)^{\alpha} \leq 1.0$$

   Where α = 1.24

   If the column is subjected to a uniaxial moment, α is chosen as 1.0.
6. If the Interaction equation is satisfied, find an arrangement with available bar sizes, find the uniaxial capacities and solve the interaction equation again. If the equation is satisfied now, the reinforcement details are written to the output file.
7. If the interaction equation is not satisfied, the assumed reinforcement is increased (ensuring that it is under 8%) and steps 2 to 6 are repeated.
8. The maximum spacing of reinforcement closest to the tension force, for purposes of crack control, is given by

$$s = 15\left(40000\frac{40000}{f_s}\right) - 2.5cc \leq 12\left(\frac{40000}{f_s}\right)$$

   with fs in psi and is permitted to be taken equal to (2/3) fy, rather than 60 percent of fy, as in ACI 318-02.

9. Section 10.9.3 has been modified to permit the use of spiral reinforcement with specified yield strength of up to

100,000 psi. For spirals with fyt greater than 60,000 psi, only mechanical or welded splices may be used.

Column Interaction

The column interaction values may be obtained by using the design parameter TRACK 1.0 or TRACK 2.0 for the column member. If a value of 2.0 is used for the TRACK parameter, 12 different Pn-Mn pairs, each representing a different point on the Pn-Mn curve are printed. Each of these points represents one of the several Pn-Mn combinations that this column is capable of carrying about the given axis, for the actual reinforcement that the column has been designed for. In the case of circular columns, the values are for any of the radial axes. The values printed for the TRACK 1.0 output are:

P0 = Maximum purely axial load carrying capacity of the column (zero moment).

Pnmax = Maximum allowable axial load on the column (Section 10.3.5 of ACI 318).

P-bal = Axial load capacity at balanced strain condition.

M-bal = Uniaxial moment capacity at balanced strain condition.

e-bal = M-bal / P-bal = Eccentricity at balanced strain condition.

M0 = Moment capacity at zero axial load.

P-tens = Maximum permissible tensile load on the column.

Des. Pn = Pu/PHI where PHI is the Strength Reduction Factor and Pu is the axial load for the critical load case.

Des. Mn = Mu*MMAG/PHI where PHI is the Strength Reduction Factor and Mu is the bending moment for the appropriate axis for the critical load case. For circular columns,

$$M_u = \sqrt{M_{uy}^2 + M_{uz}^2}$$

e/h = (Mn/Pn)/h where h is the length of the column.

Example for column design per the ACI 318-2005 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2005
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Example for column design per the ACI 318-2002 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2002
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Example for column design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN COLUMN 23 25
END CONCRETE DESIGN
```

Column Design Output

The following table illustrates different levels of the column design output.

Table 3.3

The following output is generated without any TRACK specification, thus uses the default of TRACK 0.0.

```
====================================================================

     C O L U M N   N O.   5   D E S I G N   R E S U L T S

  FY - 60000  FC - 4000 PSI,  SQRE SIZE - 12.00 X 12.00 INCHES, TIED

        AREA OF STEEL REQUIRED =   7.888  SQ. IN.

  BAR CONFIGURATION       REINF PCT.   LOAD   LOCATION   PHI
  ----------------------------------------------------------

   8 - NUMBER 9            5.556        2      STA     0.700
  (PROVIDE EQUAL NUMBER OF BARS AT EACH FACE)
```

TRACK 1.0 generates the following additional output.

```
 COLUMN INTERACTION: MOMENT ABOUT Z -AXIS (KIP-FT)

 ----------------------------------------------------------
   P0         Pn max     P-bal.     M-bal.    e-bal.(inch)
   897.12     717.70     189.56     158.50    10.03
    M0        P-tens.    Des.Pn     Des.Mn      e/h
   137.46    -432.00     323.12       9.88     0.003
 ----------------------------------------------------------

    COLUMN INTERACTION: MOMENT ABOUT Y -AXIS (KIP-FT)

 ----------------------------------------------------------
   P0         Pn max     P-bal.     M-bal.    e-bal.(inch)
   897.12     717.70     189.56     158.50    10.03
    M0        P-tens.    Des.Pn     Des.Mn      e/h
   137.46    -432.00     323.12     128.08     0.033
 ----------------------------------------------------------
```

TRACK 2.0 generates the following output in addition to all of the above.

```
                                     Pn       Mn       Pn       Mn    (@ Z )
              |                   210.44   166.60   156.52   170.63
          P0  |*                  202.41   168.67   149.93   169.69
              | *                 194.25   169.97   143.69   168.61
      Pn,max|__*                  185.95   170.45   137.80   167.44
              |    *              178.93   171.48   132.25   166.20
      Pn      |     *             163.81   171.74   127.39   165.41
 NOMINAL      |      *               Pn       Mn       Pn       Mn    (@ Y )
  AXIAL       |       *           210.44   166.60   156.52   170.63
 COMPRESSION|          *          202.41   168.67   149.93   169.69
          Pb|-------*Mb           194.25   169.97   143.69   168.61
              |       *           185.95   170.45   137.80   167.44
___________|____*_______          178.93   171.48   132.25   166.20
              |   * M0    Mn,     163.81   171.74   127.39   165.41
              | *    BENDING
      P-tens|*        MOMENT
              |
```

3.8 Designing elements, shear walls, slabs

STAAD currently provides facilities for designing 2 types of entities associated with surface type of structures.

a. Individual plate elements – these are designed from the standpoint that one element is independent of the next element. See section 3.8.1 for details.
b. Shear Walls – Structural components modelled using the SURFACE INCIDENCE command can be designed as shear walls. See section 3.8.2 for details.

3.8.1 Element Design

Element design will be performed only for the moments MX and MY at the center of the element. Design will not be performed for SX, SY, SXY, SQX, SQY or MXY. Also, design is not performed at any other point on the surface of the element.

A typical example of element design output is shown in Table 3.4. The reinforcement required to resist Mx moment is denoted as longitudinal reinforcement and the reinforcement required to resist My moment is denoted as transverse reinforcement (Figure 3.1). The parameters FYMAIN, FC, CLT and CLB listed in Table 3.1 are relevant to slab design. Other parameters mentioned in Table 3.1 are not applicable to slab design.

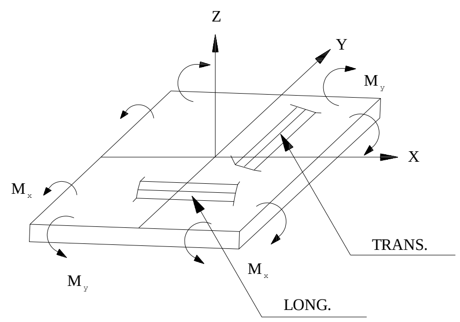


Figure 3.7

Table 3.4
(Actual Output from Design)

```
ELEMENT STRESSES     FORCE,LENGTH UNITS= KIP   FEET
----------------

         STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

ELEMENT   LOAD        SQX          SQY          MX          MY          MXY
                      VONT         VONB         SX          SY          SXY
                      TRESCAT      TRESCAB

   47       1         1.72         0.45       -11.60      -14.83        1.41
                    331.08       327.84        -1.19       -1.68        0.51
                    370.59       366.68
     TOP :  SMAX=  -266.63  SMIN=  -370.59  TMAX=    51.98  ANGLE=  20.7
     BOTT:  SMAX=   366.68  SMIN=   264.81  TMAX=    50.94  ANGLE=  20.4


 ELEMENT DESIGN SUMMARY
 ----------------------

 ELEMENT      LONG. REINF      MOM-X /LOAD      TRANS. REINF      MOM-Y /LOAD
              (SQ.IN/FT)      (K-FT/FT)         (SQ.IN/FT)      (K-FT/FT)

   47 TOP : Longitudinal direction - Only minimum steel required.
   47 TOP : Transverse direction   - Only minimum steel required.
   47 TOP :      0.130          0.00 /    0        0.130          0.00 /    0
      BOTT:      0.562         11.60 /    1        0.851         14.83 /    1
```

Example for element design per the ACI 318-2002 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 2002

or

CODE ACI
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN
```

Example for element design per the ACI 318-1999 code

```
UNIT KIP INCH
START CONCRETE DESIGN
CODE ACI 1999
FYMAIN 58 ALL
MAXMAIN 10 ALL
CLB 2.5 ALL
DESIGN ELEMENT 43
END CONCRETE DESIGN
```

3.8.2 Shear Wall Design

Purpose

Design of shear walls in accordance with ACI 318-02 has been implemented. Shear walls have to be modelled using the Surface element. The program implements provisions of chapters 10, 11 and 14 of ACI 318-02.

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces incidences -	5.13.3
Openings in surfaces -	5.13.3
Local coordinate system for surfaces -	1.6.3
Specifying sections for stress/force output -	5.13.3
Property for surfaces -	5.21.2
Material constants -	5.26.3
Surface loading -	5.32.3.4
Stress/Force output printing -	5.42
Shear Wall Design -	3.8.2, 5.54

Description

The program implements the provisions of ACI 318-02, except Chapter 21, for the design of shear walls. It performs in-plane shear, compression, as well as in-plane and out-of-plane bending design of reinforcing. The shear wall is modeled by a single or a combination of Surface elements. The use of the Surface element enables the designer to treat the entire wall as one entity. It greatly simplifies the modeling of the wall and adds clarity to the analysis and design output. The results are presented in the context of the entire wall rather than individual finite elements thereby allowing users to quickly locate required information.

The definition of the shear wall starts with specification of the Surface element(s). The boundary of the wall should include all corner joints as well as any additional joints required for connections with other elements of the building, such as beams. All geometry and material properties must be specified for surface elements separately from specifications for other element types.

The wall may be loaded indirectly through members/elements attached to it, or directly by application of nodal or uniformly distributed loads. All wall elements must be of constant thickness.

The program reports shear wall design results for each load case/combination for a user specified number of sections given by the SURFACE DIVISION (default value is 10) command. The wall is designed at these horizontal sections. The output includes the required horizontal and vertical distributed reinforcing, the concentrated (in-plane bending) reinforcing, and the links required to resist out-of-plane shear.

General format:

START SHEARWALL DESIGN
CODE ACI
FYMAIN f1
FC f2
HMIN f3
HMAX f4
VMIN f5
VMAX f6
EMIN f7
EMAX f8
LMIN f9
LMAX f10
CLEAR f11
TWOLAYERED f12
KSLENDER f13
DESIGN SHEARWALL LIST shearwall-list
END

The following table explains parameters used in the shear wall design command block above. All reinforcing bar sizes are English designation (#).

Table 3.5 - SHEAR WALL DESIGN PARAMETERS

Parameter Name	Default Value	Description
FYMAIN	60.0 ksi	Yield strength of steel, in current units.
FC	4.0 ksi	Compressive strength of concrete, in current units.
HMIN	3	Minimum size of horizontal reinforcing bars (range 3 - 18).
HMAX	18	Maximum size of horizontal reinforcing bars (range 3 - 18).
VMIN	3	Minimum size of vertical reinforcing bars (range 3 - 18).
VMAX	18	Maximum size of vertical reinforcing bars (range 3 - 18).
EMIN	3	Minimum size of vertical reinforcing bars located in edge zones (range 3 - 18).
EMAX	18	Maximum size of vertical reinforcing bars located in edge zones (range 3 - 18).

Table 3.5 - SHEAR WALL DESIGN PARAMETERS		
Parameter Name	**Default Value**	**Description**
LMIN	3	Minimum size of links (range 3 - 18)
LMAX	18	Maximum size of links (range 3 - 18)
CLEAR	3.0 in	Clear concrete cover, in current units.
TWOLAYERED	0	Reinforcement placement mode: 0 - single layer, each direction 1 - two layers, each direction
KSLENDER	1.5	Slenderness factor for finding effective height

Example

```
.
.
.
SET DIVISION 12

SURFACE INCIDENCES
2 5 37 34 SUR 1
19 16 65 68 SUR 2
11 15 186 165 SUR 3
10 6 138 159 SUR 4
.
.
.
SURFACE PROPERTY
1 TO 4 THI 18

SUPPORTS
1 7 14 20 PINNED
```

```
2 TO 5 GEN PIN
6 TO 10 GEN PIN
11 TO 15 GEN PIN
19 TO 16 GEN PIN
.
.
.
SURFACE CONSTANTS
E 3150
POISSON 0.17
DENSITY 8.68e-005
ALPHA 5.5e-006
.
.
.
START SHEARWALL DES
CODE ACI
FC 4
FYMAIN 60
TWO 1
VMIN 5
HMIN 5
EMIN 8
DESIGN SHEA LIST 1 TO 4
END
```

Notes regarding the above example:

1. Command **SET DIVISION 12** indicates that the surface boundary node-to-node segments will be subdivided into 12 fragments prior to finite element mesh generation.
2. Four surfaces are defined by the **SURFACE INCIDENCES** command.
3. The **SUPPORTS** command includes the support generation routine. For instance, the line **2 TO 5 GEN PIN** assigns pinned supports to all nodes between nodes 2 and 5. As the node-to-node distances were previously subdivided by the **SET DIVISION 12** command, there will be an additional 11 nodes between nodes 2 and 5. As a result, all 13 nodes will be assigned pinned supports. Please note that the additional 11 nodes are not individually accessible to the user. They are created by the program to enable the finite element mesh generation and to allow application of boundary constraints.
4. Surface thickness and material constants are specified by the **SURFACE PROPERTY** and **SURFACE CONSTANTS**, respectively.
5. The shear wall design commands are listed between lines **START SHEARWALL DES** and **END**. The **CODE** command selects the design code that will be the basis for the design. The **DESIGN SHEARWALL LIST** command is followed by a list of previously defined Surface elements intended as shear walls and/or shear wall components. Refer to the beginning of this section for references to all related commands.

Technical Overview

The program implements provisions of Chapter 14 of ACI-318-02 and relevant provisions from Chapters 10 and 11, as referenced therein, for all active load cases. The wall is designed as an unbraced reinforced wall. The following steps are performed for each of the horizontal sections of the wall set using the SURFACE DIVISION command whose default value is 10.

Design for in-plane shear (denoted using Fxy in the shear wall force output) per Section 11.10 of ACI 318

- Extreme compression fiber to centroid of tension (concentrated) reinforcement distance, d, is taken as 0.8 horizontal length of the wall (ACI - 11.10.4),
- Limit on the nominal shear strength, Vn, is calculated (ACI - 11.10.3),
- Nominal shear strength of concrete is computed (11.10.6),
- If the factored shear force does not exceed ½ of the design strength of concrete, the minimum ratios of shear (distributed) reinforcing are reported, in accordance with 14.3.2 and 14.3.3. Otherwise, the reinforcing ratios are established in accordance with 11.10.9.
- If the factored shear force is greater than ½ but does not exceed the design strength of concrete, the ratios reported are the minima calculated in accordance with 11.10.9.2 and 11.10.9.4.
- If the factored shear force exceeds the design strength of concrete, the distributed reinforcing is calculated based on 11.1.1 and 11.10.9.
- Number of distributed reinforcing layers and reinforcing allocation between layers reflect requirements of 14.3.4.
- Rebar spacing is given c/c and meets the requirements of 14.3.5, 11.10.9.3, and 11.10.9.5 of ACI 318, as applicable.

Design for in-plane bending (denoted by Mz in the shear wall force output) per Section 14.4 of ACI 318

- Walls are assumed to be cantilever beams fixed at their base and carrying loads to the foundation.
- Strength reduction factor is established in accordance with Section 9.3.2.
- Minimum reinforcing is calculated in accordance with 10.5.1 or 10.5.3, whichever produces a smaller ratio.

- Extreme compression fiber to centroid of tension reinforcement distance, d, is taken as 0.8 horizontal length of the wall (11.10.4 of ACI 318).
- Flexural design of the wall is carried out in accordance with provisions of Chapter 10.
- The flexural (concentrated) reinforcing is located at both ends (edges) of the length of the wall. Rebar layout conforms to the spacing requirements of Section 7.6.

Design for compression and out-of-plane bending (Section 14.8)

- The design is based on the Alternative Design of Slender Walls procedure.
- The procedure requires that the wall panel be designed as simply supported (at top and bottom), axially loaded with out-of-plane uniform lateral load, with maximum moments and deflections occurring at mid-height.
- Minimum distributed reinforcing ratio is controlled by the in-plane shear design calculations.
- The reinforcing amount required by 14.8 is over and above any reinforcing required due to in-plane shear.

3.8.3 Slabs and RC Designer

Contact the technical support department for further information.

3.8.4 Design of I-shaped beams per ACI-318

I-shaped sections can be designed as beams per the ACI 318 code. The property for these sections must be defined through a user table, I-section, or using the tapered specification. Information on assigning properties in this manner is available in sections 5.19 (I-section type) and 5.20.3 (Tapered I shape) of the Technical Reference manual.

From the standpoint of the analysis – determining member forces, nodal displacements and support reactions – the same set of facilities and rules which are applicable for any normal reinforced concrete frames or other structures can be used when I-sections or tapered concrete members are specified. In other words, there isn't anything unique or special to account for in the analysis model simply because I-shaped concrete beams are part of it.

From the standpoint of design, the following rules are applicable:

1. The member can be designed as a beam using the general principles explained in Chapter 3 of the Technical Reference manual. It currently cannot be designed as a column. Design as a beam is done for flexure (MZ), shear (FY) and torsion (MX) just like that for rectangular, tee or trapezoidal beams. Axial forces (FX) are used during the capacity computations in shear and torsion. At each section along the length that the member is designed at, the depth at that section location is used for effective depth computation.
2. The program performs the following tests on the section dimensions before starting the design:

- If the thickness of the web is the same as the width of the top and bottom flanges, the member is designed as a rectangular section.
- If the thickness of the web is the same as the width of one of the flanges but not the other, the member is designed as a T-section or a rectangular section, depending on which side the compression due to bending is at.

- If the web thickness does not match the width of either flange, design is done using the rules applicable for T-beams – one flange is in compression, the other in tension, and tensile capacity of concrete on the tensile side of the neutral axis is ignored.
- The program is also able to design the beam as a doubly reinforced section if it is unable to design it as a single-reinforced section.

3. The parameters for designing these members are as shown in Table 3.1 of this manual. Detailed output on design at individual section locations along the member length may be obtained by setting the TRACK parameter to 3.0.

An example for I-beam design is shown below.

```
STAAD PLANE I BEAM CONCRETE DESIGN PER ACI-318

UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 10 0 0

MEMBER INCIDENCES
1 1 2

UNIT INCHES KIP
MEMBER PROPERTY
1 TAPERED 18 10 18 15 2.5

CONSTANTS
E 3300 ALL
DENSITY CONCRETE ALL
POISSON CONCRETE ALL

SUPPORTS
1 2 PINNED

UNIT FEET KIP
LOAD 1 DEAD LOAD
MEMBER LOAD
```

```
1 UNI GY -5.76

LOAD 2 LIVE LOAD
MEMBER LOAD
1 UNI GY -7.04

LOAD COMB 3 ACI 318-02
1 1.4 2 1.7

PERFORM ANALYSIS

LOAD LIST 3
START CONCRETE DESIGN
CODE ACI 2002
UNIT INCHES KIP
MINMAIN 9 ALL
FC 4 ALL
FYMAIN 60 ALL
TRACK 2.0 ALL
DESIGN BEAM ALL
END CONCRETE DESIGN
FINISH
```

*N
o
t
e
s*

Timber Design

Section 4

4.1 Timber Design

STAAD.Pro supports timber design per two codes – 1985 AITC code and 1994 AITC code. The implementation of both the codes is explained below.

1994 AITC code implementation

The salient aspects of design in accordance with the 4th edition (1994) of the Timber Construction Manual published by the American Institute of Timber Construction are:

1. Design can be performed for two types of timber sections - dimensional timber sections, also known as sawn lumber, and, glulaminated sections. The design facilities of the 1985 code implementation in STAAD was limited to glulaminated sections.
2. The program includes a database of dimensional timber sections with this code.

Implementation of Dimensional Lumber Properties

The database of sawn lumber sections, listed in Table 8.1 of the 1994 AITC Manual, is implemented in the program. Some of the key aspects of this implementation are:

In the property tables in the AITC manual, one will find that, for any particular species of timber, the Modulus of Elasticity (E) and allowable stresses may vary with the cross-section size. For example, a 2x4 Douglas Fir-Larch, Select Structural member has an E of 1900 ksi and an allowable bending stress, F_b, of 1450 psi. A 5x5 Douglas Fir-Larch, Select Structural, Beam or Stringer member has an E of 1600 ksi and an allowable bending stress, F_b, of 1600 psi. And a 5x5 Douglas Fir-Larch, Select Structural, Post or Timbers member has an E of 1600 ksi and an allowable bending stress, F_b, of 1750 psi.

So, in the STAAD timber database for sawn lumber, for each species and grade of timber, the section size, or properties are associated with the Modulus of Elasticity and allowable stresses for the cross-section. When a section is assigned, its E and allowable stresses are automatically fetched along with its properties. The material properties of Southern Pine members were taken from Table 8.4 of the 1994 AITC manual. For all other species with section sizes 2"-4" wide, the material properties have been taken from Table 8.3. For all non-Southern Pine species with section sizes greater than 5"x5", the material properties are obtained from Table 8.6 of the 1994 AITC Manual.

Please note that not all section sizes listed in Table 8.1 are available in every species. Some sizes are not produced for particular species. For example, the Aspen species only produces sizes from 2"-4" wide. It does not produce sizes 5"x5" and larger. This can be observed by comparing Table 8.3, where Aspen is listed as an available species, to Table 8.6, where Aspen is not listed as an available species. Also note that although 1" wide members are listed in Table 8.1, there are no values available in the species properties tables; Table 8.4, Table 8.5, and Table 8.6. AITC does not allow for the structural design of these small members.

Naming convention in STAAD.Pro for Dimensional Lumber sections

As can be seen from Tables 8.3 through 8.6 of the AITC 1994 manual, one or more of the following attributes have to be considered while choosing a section :

- Species
- Commercial Grade
- Size classification
- Nominal size of the section
- Grading rules agency

STAAD uses a naming convention that incorporates all of the above. Shown below is the name of a section that has characteristics as shown. It may be found on page 8-637 of the AITC 1994 manual.

Species: Douglas Fir Larch
Commercial Grade: Dense Select Structural
Size Classification: Beams
Nominal size: 5”x5”
Grading Rules Agency: WCLIB

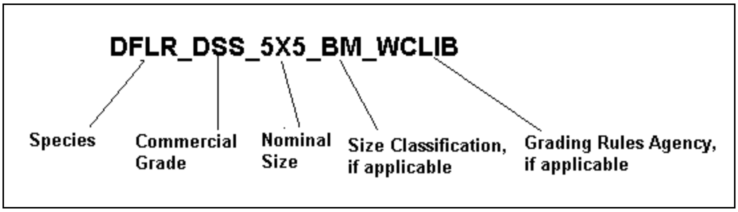


Figure 4.1

Implementation of Glulam Properties

For Glulam sections, each glulam designation has an associated value of Modulus of Elasticity and a set of allowable stresses. However, these values are not dependent on the size of the cross section. For example, a 3-1/8”x 6” 24F-V8 DF/DF beam and a 6-

3/4” x 30” 24F-V8 DF/DF beam both have an E of 1600 ksi and an allowable bending stress in the tension zone, F_{bx}, of 2400 psi. Therefore, in STAAD’s glulam database, the section sizes are not linked to the glulam type. Users may specify any cross-section size they choose and pick the desired glulam type. The Modulus of Elasticity and allowable stresses associated with that glulam are assigned to the member. The material properties for the Glulam database are taken from Table 1 of AITC 117-93 – Design Standard Specifications for Structural Glued Laminated Timber of Softwood Species. This publication has been reproduced in the AITC 1994 manual starting from page 8-843.

Naming convention in STAAD.Pro for Glulam sections

The STAAD name for glulam sections incorporates

Combination Symbol
Species-Outer Laminations/Core Laminations

Shown here is a typical section listed in page 8-854 of the AITC manual.

GLT-24F-V11_DF/DFS

Implementation of Material Constants

As explained in the previous paragraphs, for sawn lumber as well as glulam sections, E is built into the database and gets automatically assigned to the member along with the section dimensions. Density, Poisson’s ratio and Alpha (coefficient of thermal expansion) have to be assigned separately. If they are not assigned, the analysis engine will use default values for those.

A typical set of input data for section properties and constants is as shown below.

Example for Dimensional Timber:

```
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC DFLN_SS_4X4
E 273600
POISSON 0.15
DENSITY 0.025
ALPHA 5.5e-006
END DEFINE MATERIAL

MEMBER PROPERTY AITC
3 4 7 8 TABLE ST DFLN_SS_4X4

CONSTANTS
MATERIAL DFLN_SS_4X4 MEMB 3 4 7 8
```

Example for Glulam Timber:

```
UNIT FEET KIP
DEFINE MATERIAL START
ISOTROPIC GLT-24F-V8_WET_DF/DF
E 191923
POISSON 0.15
DENSITY 0.025
ALPHA 5.5e-006
END DEFINE MATERIAL

MEMBER PROPERTY AITC
8 PRIS YD 1.5 ZD 0.427083

CONSTANTS
MATERIAL GLT-24F-V8_WET_DF/DF MEMB 8
```

Assigning the input

Please see the Graphical User Interface manual for the procedure for assigning the properties, glulam types and material constants.

Design parameters

The timber design parameters for the AITC 4th Edition are listed below.

Table 4.1 - AITC 1994 Timber Design Parameters

Parameter name referred to in AITC 1994 code document	Name used in STAAD	Default value and units if applicable	Description
C_b	CB	1.0	Bearing Area Factor, Table 4.13
C_F	CFB	1.0	Size Factor for Allowable Bending Stress, see Table 8.3, 8.4, 8.5, 8.6, 8.7
C_F	CFC	1.0	Size Factor for Allowable Compression Parallel to Grain, see Table 8.3, 8.4, 8.5, 8.6, 8.7
C_F	CFT	1.0	Size Factor for Allowable Tension Parallel to Grain, see Table 8.3, 8.4, 8.5, 8.6, 8.7
C_{fu}	CFU	1.0	Flat Use Factor, see Table 4.9
C_H	CSS	1.0	Shear Stress Factor, Section 4.5.14
C_M	CMB	1.0	Wet service Factor for Allowable Bending Stress, see Table 4.8
C_M	CMC	1.0	Wet service Factor for Allowable Compression Parallel to Grain, see Table 4.8
C_M	CME	1.0	Wet service Factor for Modulus of Elasticity, see Table 4.8
C_M	CMP	1.0	Wet service Factor for Allowable Compression Perpendicular to Grain, see Table 4.8
C_M	CMT	1.0	Wet service Factor for Allowable Tension Parallel to Grain, see Table 4.8
C_M	CMV	1.0	Wet service Factor for Allowable Shear Stress Parallel to Grain, see Table 4.8

Table 4.1 - AITC 1994 Timber Design Parameters

Parameter name referred to in AITC 1994 code document	Name used in STAAD	Default value and units if applicable	Description
C_r	CR	1.0	Repetitive Member Factor, see Section 4.5.10
C_F	CSF	1.0	Form Factor, see Section 4.5.12
C_t	CTM	1.0	Temperature Factor, see Table 4.11
C_T	CTT	1.0	Buckling Stiffness Factor, see Section 4.5.15
K_b	KB	1.0	Buckling Length Coefficient to calculate Effective Length
K_{bd}	KBD	1.0	Buckling Length Coefficient for Depth to calculate Effective Length
K_{bE}	KBE	0.609	Euler Buckling Coefficient for Beams, see Section 5.4.11
K_{cE}	KCE	1.0	Euler Buckling Coefficient for Columns, see Section 5.8.2
K_{ey}	KEY	1.0	Buckling Length Coefficient in Y Direction
K_{ez}	KEZ	1.0	Buckling Length Coefficient in Z Direction
K_l	KL	1.0	Load Condition Coefficient, Table 4.10
LZ	LZ	Member Length	Effective shear length in the z direction for Column Stability Check, Le=Ke*L
LY	LY	Member Length	Effective shear length in the y direction for Column Stability Check, Le=Ke*L
LUZ	LUZ	Member Length	Member length in the z direction for Beam Stability Check, Lu=Kb*l+Kbd*d
LUY	LUY	Member Length	Member length in the y direction for Beam Stability Check, Lu=Kb*l+Kbd*d
CDT	CDT	1.0	Load Duration Factor
CCR	CCR	1.0	Curvature factor (Section 4.5.11)
INDEX	INDEX	10	Exponent value in the Volume Factor Equation (Section 4.5.6)
CV	CV	1.0	Volume Factor (Section 4.5.6)
CC	CC	0.8	Variable in Column Stability Factor, Cp (Section 5.8.2, Eqn 5-14)

Table 4.1 - AITC 1994 Timber Design Parameters

Parameter name referred to in AITC 1994 code document	**Name used in STAAD**	**Default value and units if applicable**	**Description**
SRC	SRC	1.0	Slenderness ratio of Compression member
SRT	SRT	1.0	Slenderness ratio of Tension member
	RATIO	1.0	Permissible ratio of actual to allowable stress
	BEAM	1.0	0 = Design for end forces or locations specified by section command. 1 = Calculate moments at 12 pts along the beam and use the maximum for design.

Design commands

Only code checking is available for timber members. Member selection is currently not available. It will be incorporated in a future build of the program. The command is specified as

CHECK CODE memb-list

The code name command for timber design is treated by STAAD in the following manner :

Command in STAAD	**Interpretation by the program**
CODE AITC	AITC 1994 edition
CODE AITC 1994	AITC 1994 edition
CODE AITC 1984	AITC 1984 edition
CODE TIMBER	AITC 1984 edition

Example for Dimensional lumber:

```
STAAD PLANE EXAMPLE FOR DIMENSIONAL LUMBER

UNIT FEET POUND
JOINT COORDINATES
1 0 0 0; 2 6 0 0; 3 12 0 0; 4 18 0 0;
5 24 0 0; 6 6 3 0; 7 12 6 0; 8 18 3 0;

MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 1 6; 6 6 7; 7 7 8; 8 8 5;
9 2 6; 10 3 7; 11 4 8; 12 6 3; 13 3 8;

UNIT FEET POUND
DEFINE MATERIAL START
ISOTROPIC DFLR_SS_2X4
E 2.736e+008
POISSON 0.15
DENSITY 25
ALPHA 5.5e-006

ISOTROPIC DFLR_SS_3X6
E 2.736e+008
POISSON 0.15
DENSITY 25
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY AITC
1 TO 4 9 TO 11 TABLE ST DFLR_SS_2X4
5 TO 8 12 13 TABLE ST DFLR_SS_3X6

CONSTANTS
MATERIAL DFLR_SS_2X4 MEMB 1 TO 4 9 TO 11
MATERIAL DFLR_SS_3X6 MEMB 5 TO 8 12 13

MEMBER RELEASE
9 TO 13 START MP 0.99
9 TO 13 END   MP 0.99
6 END MP 0.99
```

```
7 START MP 0.99

SUPPORTS
1 PINNED
5 FIXED BUT FX MZ

UNIT FEET POUND

LOAD 1 DEAD+LIVE LOAD
SELFWEIGHT Y -1
MEMBER LOAD
1 TO 4 UNI GY -30
5 TO 8 UNI GY -40

LOAD 2 SNOW LOAD
MEMBER LOAD
5 TO 8 UNI GY -50

LOAD 3 WIND LOAD
MEMBER LOAD
5 6 UNI Y -30
7 8 UNI Y 25

LOAD COMB 11 D+L+SNOW
1 1.0 2 1.0

LOAD COMB 12 D+L+SNOW+WIND
1 1.0 2 1.0 3 1.0

PERFORM ANALYSIS PRINT STATICS CHECK

PARAMETER
CODE AITC
BEAM 1.0 ALL
CHECK CODE ALL

FINISH
```

Example for Glulaminated lumber:

```
STAAD PLANE EXAMPLE FOR GLULAM DESIGN

INPUT WIDTH 79

UNIT FEET KIP
JOINT COORDINATES
1 0 0 0; 2 12 0 0; 3 24 0 0; 4 36 0 0; 5 0 12 0; 6 6 10 0; 7 18
6 0; 8 30 2 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4; 4 5 6; 5 6 7; 6 7 8; 7 8 4; 8 1 5; 9 2 6; 10
3 7; 11 1 6;
12 2 7; 13 3 8;

UNIT INCHES KIP
DEFINE MATERIAL START
ISOTROPIC GLT-24F-V8_DF/DF
E 1600
POISSON 0.15
DENSITY 1.44676e-005
ALPHA 5.5e-006
END DEFINE MATERIAL
MEMBER PROPERTY
1 TO 7 PRIS YD 16.5 ZD 10.75
8 TO 13 PRIS YD 10.5 ZD 8.75

CONSTANTS
MATERIAL GLT-24F-V8_DF/DF MEMB 1 TO 13

SUPPORTS
1 4 PINNED

UNIT POUND FEET

LOAD 1  DEAD
SELFWEIGHT Y -1

LOAD 2  LIVE
```

```
MEMBER LOAD
1 TO 3 UNI GY -100
4 TO 7 UNI GY -100

LOAD COMB 3
1 1.0 2 1.0

PERFORM ANALYSIS PRINT STATICS CHECK

PARAMETER
CODE AITC
CMT 1 ALL
RATIO 0.9 ALL
CHECK CODE ALL

FINISH
```

1985 AITC code implementation

STAAD's Timber design module per the 1985 AITC code (Timber Construction Manual, 3rd. Edition, 1985) allows design of Glulam timber sections. It also conforms to the National Design Specification for Wood Construction and Supplement (NDS) and building codes like Uniform Building Code (UBC), Basic/National Building Code and Standard Building Code. Some of the main features of the program are:

1. This feature is for Glulam Timber only.
2. Code check and design of members as per TCM - AITC.
3. Design values for Structural Glued Laminated Timber tables are in-built into the program. The program accepts Table no., Combination and Species specifications as inputs (e.g, 1:16F-V3-SP/SP) and reads design values from in-built tables.
4. Incorporates all the following Allowable stress modifiers:
   i) Duration of Load Factor
   ii) Size Factor

iii) Form Factor

iv) Lateral stability of Beams and Columns

v) Moisture Content Factor

vi) Temperature and Curvature factors.

The allowable stresses for bending, tension, compression, shear and Moduli of elasticities are modified accordingly .

5. Determines slenderness for beams and columns (Short, intermediate and long) and checks for min. eccentricity, lateral stability, buckling, bending and compression, bending and tension and horizontal shear against both axes.
6. The output results show sections provided or chosen, actual and allowable stresses, governing condition and ratios of interaction formulae and the relevant AITC clause nos. etc for each individual member.

4.2 Design Operations

Explanation of terms and symbols used in this section

Symbols	Description
f_a	Actual compression or tension stress (in PSI). For tension, the axial load is divided by net sectional area (i.e., NSF x X-area).
FA	Allowable design value for compression or tension (in PSI) modified with applicable modifiers or calculated based on slenderness in case of compression.
f_{bz}, f_{by}	Actual bending stresses about local Z and Y axis (in PSI).
FBZ, FBY	Allowable design values for bending stresses about local Z and Y axis (in PSI) modified by the applicable modifiers.
JZ, JY	Modifier for P-DELTA effect about the Z and Y axis respectively as explained in formula 5-18 of TCM.

f_{vz}, f_{vy}	Actual horizontal shear stresses.
FVZ, FVY	Allowable horizontal shear stresses.
VZ, VY	Shear in local Z and local Y direction.
ZD, YD	Depth of section in local Z and Y axis.
EZ, EY	Minimum eccentricity along Z and Y axis.
CFZ, CFY	CFZ and CFY are values of the size factors in the Z-axis and Y-axis respectively.
CLZ, CLY	CLZ and CLY represent the factors of lateral stability for beams about Z-axis and Y-axis respectively.
RATIO	Permissible ratio of the stresses as provided by the user. The default value is 1.

Combined Bending and Axial Stresses

Bending and Axial Tension:

The following interaction formulae are checked :

i) f_a/FA + f_{bz}/(FBZ x CFZ) + f_{by}/(FBY x CFY) =< RATIO

ii) Lateral stability check with Net compressive stress:
- f_a/FA + f_{bz}/(FBZ x CLZ) + f_{by}/(FBY x CLY) =< RATIO

Bending and Axial Compression:

i) f_a/FA + f_{bz}/(FBZ-JZ x f_a) + f_{by}/(FBY-JY x f_a) =< RATIO

Applicability of the size factor:

a) When CF < 1.00,
if fa > FBZ x (1-CFZ), FBZ is not modified with CFZ. if fa > FBY x (1-CFY), FBY is not modified with CFY.

if fa < FBZ x (1-CFZ) FBZ is taken as FBZ x CFZ + fa but shall not exceed FBZ x CLZ

if fa < FBY x (1-CFY) FBY is taken as FBY x CFY + fa but shall not exceed FBY x CLY

b) When CF >= 1.00, the effect of CF and CL are cumulative FBZ is taken as FBZ x CFZ x CLZ FBY is taken as FBY x CFY x CLY

Min. Eccentricity:

The program checks against min. eccentricity in following cases:

a) The member is a FRAME member and not a truss member and under compression.
b) The value of actual axial compressive stress does not exceed 30% of the allowable compressive stress.
c) The actual moments about both axes are less than moments that would be caused due to min. eccentricity. In this approach, the moment due to min. eccentricity is taken as the compressive load times an eccentricity of 1 in. or 0.1 x depth whichever is larger.

In case of min. eccentricity,

f_{bz} is taken as f_a x (6+1.5 x JZ)/(EZ/ZD) f_{by} is taken as f_a x (6+1.5 x JY)/ (EY/YD)

the following conditions are checked :

f_a/FA + f_{bz}/(FBZ-JZ x f_a) =< RATIO and f_a/FA + f_{by}/(FBY-JY x f_a) =< RATIO

Shear Stresses:

Horizontal stresses are calculated and checked against allowable values:

f_{vz} = 3 x VY /(2 x Area x NSF) =< FVZ/f_{vy} = 3 x VZ /(2 x Area x NSF) =< FVY

4.3 Input Specification

A typical set of input commands for STAAD TIMBER DESIGN is listed below:

```
UNIT KIP INCH
PARAMETER
CODE TIMBER
GLULAM 1:16F-V3-DF/DF MEMB 1 TO 14
GLULAM 1:24F-V5-SP/SP MEMB 15 TO 31
GLULAM 20F-V1-DF/WW MEMB 32 TO 41
LAMIN 1.375 LY 168.0 MEMB  5  9  15 TO 31
LZ 176.0 MEMB 1 TO 4  6  7  8  10 TO 14
LUZ 322.6 ALL
LUY 322.6 ALL
WET 1.0 ALL
CDT 1.33
NSF 0.85
BEAM 1.0 ALL
CHECK CODE 1 TO 14
SELECT MEMB 15 TO 31
```

Input Commands and Parameters Explained

Specify PARAMETER and then CODE TIMBER to start TIMBER DESIGN before specifying the input parameters. The user must provide the timber grade (GLULAM GRADE) for each member he intends to design. The parameters can be specified for all or specified list of members. If a parameter is not specified, the default value is assigned to it. See following INPUT PARAMETERS LIST TABLE for description and default values of the parameters.

Glulam Grade & Allowable Stresses from Table

The allowable stresses for GLULAM members are read in from Table-1 and Table-2 of AITC for design values for Structural

Glued Laminated Timber. The structural members are to be specified in the following manner:

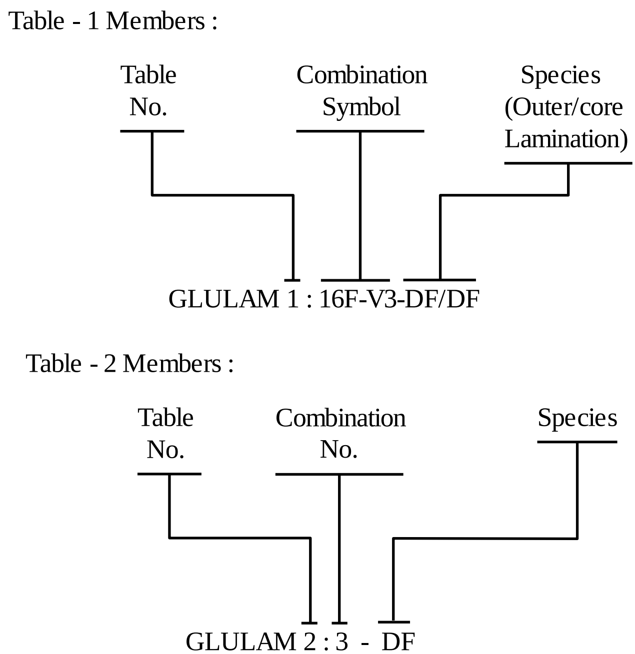


Figure 4.2

For TABLE-2 members, the applicable stress values are selected based on the depth and the number of laminations. Please note here that lamination thickness (in inch) can be provided by the user and in case it is not provided the default is taken as 1.5 inch. Usually, it is either 1.5 inch or 1.375 inch.

4.4 Code Checking

The CHECK CODE command enables the user to check the adequacy of the size (YD X ZD) provided in the MEMBER PROPERTIES for the most critical forces and moments. The program prints whether the member has PASSed or FAILed, the critical conditions and the value of the ratio.

4.5 Orientation of Lamination

Laminations are always assumed to lie along the local Z-plane of the member. The user may please note that in the MEMBER PROPERTIES section, YD always represents the depth of the section across the grain and ZD represents the width along the grain. This is in accordance with the sign convention conforming to “SET Y UP”.

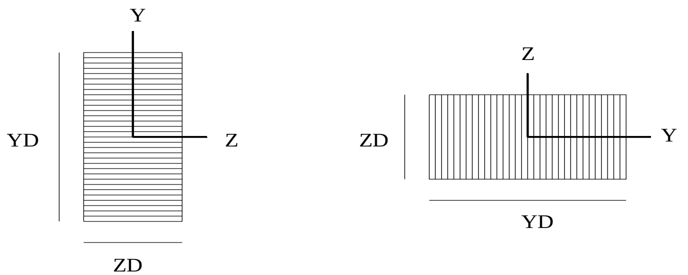


Figure 4.3

4.6 Member Selection

The SELECT MEMBER command starts with the min. permissible depth (or min. depth provided thru DMIN parameter) and checks the code. If the member fails with this depth, the thickness is increased by one lamination thickness and the codal requirements are checked again. The process is continued till the section passes all the codal requirements. This ensures the least weight section for the member. If the depth of the section reaches max. allowable or available depth and the member still fails, the user can have the following options for redesign:

i) Change the width or increase the max. allowable depth (DMAX)
ii) Change the timber grade
iii) Change the design parameters.

Table 4.2 - AITC 1985 Timber Design Parameters

Parameter Name	Default Value	Description
LZ	Length of the Member(L)	Effective length of the column in z-axis.
LY	-DO-	Same as above in y-axis.
LUZ	1.92*L	Unsupported effective length for beam in z.
LUY	1.92*L	Unsupported effective length for beam in y.
WET	0.0	0.0 - dry condition 1.0 - wet condition wet use factors are in-built
NSF	1.0	Net section factor for tension members. (both shear and tension stresses are based on sectional area x nsf)
CDT	1.0	Duration of load factor
CSF	1.0	Form factor
CTM	1.0	Temp. factor
CCR	1.0	Curvature factor.
RATIO	1.0	Permissible ratio of actual to allowable stresses.
LAMINATION	1.50 inch	Thickness of lamination in inch (1.50 or 1.375)
BEAM	1.0	0.0 = design for end forces or at locations specified by section command. 1.0 = calculate moments at twelfths sections along the beam and use the max. for design. (Default)

Note:

i. In case the column buckling is restrained in Y and/or Z direction provide LY and/or LZ as zero(s). Similarly, lateral beam buckling in Y and/or Z direction could be restrained by providing LUY and/or LUZ as zeros.

ii. Size Factor, lateral stability and moisture content factors and few others are either calculated or read from tables within the program.

SAMPLE OUTPUT RESULTS

```
                    STAAD CODE CHECKING - (AITC)
                    ***********************

 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE NOTED)

 MEMBER     TABLE        RESULT/   CRITICAL COND/     RATIO/     LOADING/
                           FX            MY             MZ       LOCATION
 =======================================================================


    2 PR  8.000X15.000  FAIL      TCM:CL. 5-18        1.205         2
                      2.24 C          0.00          45.38     0.0000
|-----------------------------------------------------------------------|
|  MEMB-    2 GLULAM GRADE:16F-V1-DF/WW     LAM.=1.500 UNITS: POUND-INCH  |
| LZ=240.00 LY=240.00 LUZ=240.00 LUY=240.00 JZ =0.370 JY =1.000 CDT=1.000  |
| CSF=1.00 WET=0.0 CCR=1.00 CTM=1.00 CFZ=0.98 CFY=1.00 CLZ=1.000 CLY=1.000 |
|ACTUAL STRESSES: fa=  18.67,fbz=1815.06,fby=   0.00,fvz= 49.37,fvy=  0.00 |
|ALLOW. STRESSES: FA= 366.67,FBZ=1579.49,FBY= 950.00,FVZ=140.00,FVY=130.00 |
|-----------------------------------------------------------------------|


    3 PR  8.000X15.000  FAIL      TCM:CL. 5-18        1.239         2
                     10.64 C          0.00          43.94     0.0000
|-----------------------------------------------------------------------|
|  MEMB-    3 GLULAM GRADE:16F-V1-DF/WW     LAM.=1.500 UNITS: POUND-INCH  |
| LZ=180.00 LY=180.00 LUZ=180.00 LUY=180.00 JZ =0.074 JY =0.997 CDT=1.000  |
| CSF=1.00 WET=0.0 CCR=1.00 CTM=1.00 CFZ=0.98 CFY=1.00 CLZ=1.000 CLY=1.000 |
|ACTUAL STRESSES: fa=  88.68,fbz=1757.49,fby=   0.00,fvz= 36.61,fvy=  0.00 |
|ALLOW. STRESSES: FA= 652.22,FBZ=1600.00,FBY= 950.00,FVZ=140.00,FVY=130.00 |
|-----------------------------------------------------------------------|


                  STAAD MEMBER SELECTION - (AITC)
                  **************************

 ALL UNITS ARE - KIP  FEET (UNLESS OTHERWISE NOTED)

 MEMBER     TABLE        RESULT/   CRITICAL COND/     RATIO/     LOADING/
                           FX            MY             MZ       LOCATION
 =======================================================================

   2 PR  8.000X18.000  PASS      TCM:CL. 5-18        0.860         2
                      2.24 C          0.00          45.38     0.0000
|-----------------------------------------------------------------------|
|  MEMB-    2 GLULAM GRADE:16F-V1-DF/WW     LAM.=1.500 UNITS: POUND-INCH  |
| LZ=240.00 LY=240.00 LUZ=240.00 LUY=240.00 JZ =0.173 JY =1.000 CDT=1.000  |
| CSF=1.00 WET=0.0 CCR=1.00 CTM=1.00 CFZ=0.96 CFY=1.00 CLZ=1.000 CLY=1.000 |
|ACTUAL STRESSES: fa=  15.56,fbz=1260.46,fby=   0.00,fvz= 41.14,fvy=  0.00 |
|ALLOW. STRESSES: FA= 366.67,FBZ=1545.08,FBY= 950.00,FVZ=140.00,FVY=130.00 |
|-----------------------------------------------------------------------|


    3 PR  8.000X18.000  PASS      TCM:CL. 5-18        0.876         2
                     10.64 C          0.00          43.94     0.0000
|-----------------------------------------------------------------------|
|  MEMB-    3 GLULAM GRADE:16F-V1-DF/WW     LAM.=1.500 UNITS: POUND-INCH  |
| LZ=180.00 LY=180.00 LUZ=180.00 LUY=180.00 JZ =0.000 JY =0.997 CDT=1.000  |
| CSF=1.00 WET=0.0 CCR=1.00 CTM=1.00 CFZ=0.96 CFY=1.00 CLZ=1.000 CLY=1.000 |
|ACTUAL STRESSES: fa=  73.90,fbz=1220.48,fby=   0.00,fvz= 30.51,fvy=  0.00 |
|ALLOW. STRESSES: FA= 652.22,FBZ=1600.00,FBY= 950.00,FVZ=140.00,FVY=130.00 |
|-----------------------------------------------------------------------|
```

Figure 4.4

Output Results and Parameters Explained

For CODE CHECKING and/or MEMBER SELECTION the output results are printed as shown in the previous section. The items are explained as follows:

a) MEMBER refers to the member number for which the design is performed.

b) TABLE refers to the size of the PRISMATIC section (B X D or ZD X YD).

c) RESULT prints whether the member has PASSed or FAILed .

d) CRITICAL COND refers to the CLAUSE or FORMULA NO. from the TIMBER CONSTRUCTION MANUAL (3rd. Edition, AITC-1985) which governed the design. See following table:

Critical Condition	**Governing Criteria**
CLAUSE 5-19	Axial Compression and Bending with MINIMUM ECCENTRICITY.
CLAUSE 5-18	Axial Compression and Bending
CLAUSE 5-42	Axial Tension and Bending
CLAUSE 5-24	Horizontal Shear
CLAUSE 5-40	Lateral stability for net compressive stress in case of Tension and Bending.

e) RATIO prints the ratio of the actual stresses to allowable stresses for the critical condition. This ratio is usually the cumulative ratio of stresses in the interaction formula. In case of shear governing the design, it means the ratio of the actual shear stress to allowable shear stress. If this value exceeds the allowable ratio (default 1.0) the member is FAILed.

f) LOADING provides the load case number that governed.

g) FX, MY and MZ provide the design axial force, moment in local Y axes and moment in local Z axes respectively. FX value is followed by a letter C or T to denote COMPRESSION or TENSION.

h) LOCATION specifies the actual distance from the start of the member to the section where design forces govern in case BEAM command or SECTION command is specified.

OUTPUT parameters that appear within the box are explained as follows:

a) MEMB refers to the same member number for which the design is performed.

b) GLULAM GRADE refers to the grade of the timber.

c) LAM refers to lamination thickness provided in the input or assumed by the program. See INPUT PARAMETERS section.

d) LZ, LY, LUZ and LUY are the effective lengths as provided or calculated. See INPUT PARAMETERS section.

e) JZ and JY are the modifiers for the P-DELTA effect about Z-axis and Y-axis respectively. These are calculated by the program.

f) CDT, CSF, WET, CCR, CTM are the allowable stress modifiers explained in the INPUT PARAMETERS section.

g) CFZ and CFY are values of the size factors in the Z-axis and Y-axis respectively. CLZ and CLY represent the factors of lateral stability for beams about Z-axis and Y-axis respectively. These values are printed to help the user see the intermediate design values and re-check the design calculations.

h) f_a, f_{bz}, f_{by}, f_{vz} and f_{vy} are the actual axial stress, bending stresses about Z and Y axes and horizontal shear stresses about Z and Y axes respectively. If the bending moments about both axes are less then the eccentric moments based on min. eccentricity then bending stresses are calculated based on the min. eccentricity . Refer DESIGN OPERATIONS section for details.

i) FA, FBZ, FBY, FVZ and FVY are the final allowable axial, bending (Z and Y axes) and horizontal shear (Z and Y axes) stresses. Refer DESIGN OPERATIONS section for details.

STAAD Commands and Input Instructions

Section 5

This section of the manual describes in detail various commands and related instructions for STAAD. The user utilizes a command language format to communicate instructions to the program. Each of these commands either supplies some data to the program or instructs it to perform some calculations using the data already specified. The command language format and conventions are described in Section 5.1. This is followed by a description of the available commands.

Although the STAAD input can be created through the Modeling mode, it is important to understand the command language. With the knowledge of this language, it is easy to understand the problem and add or comment data as necessary. The general sequence in which the commands should appear in an input file should ideally follow the same sequence in which they are presented in this section. The commands are executed in the sequence entered. Obviously then the data needed for proper execution of a command must precede the command (e.g. Print results after Perform Analysis). Otherwise, the commands can be provided in any order with the following exceptions.

i) All design related data can be provided only after the analysis command.

ii) All load cases and load combinations must be provided together, except in a case where the CHANGE command is used. Additional load cases can be provided in the latter part of input.

All input data provided is stored by the program. Data can be added, deleted or modified within an existing data file.

In STAAD.Pro 2006 and earlier, all analytical calculations such as joint displacements, eigenvalues and eigenvectors are calculated using an analysis engine which for identification purposes is known as the Standard STAAD Engine. This engine has been able to handle the analytical requirements for a vast majority of STAAD models that users have created in the recent years.

As computer resources such as processor speed, memory and disk space have grown, the users are also creating larger models. As a result, numerically faster algorithms and solution techniques have become necessary. Also, new features such as pushover analysis and buckling analysis which are outside the scope of the standard engine have made it necessary to introduce a new engine which is known as the Advanced Solver. This new Solver is available as an alternative engine effective from STAAD.Pro 2007, Build 1001.

The Advanced Solver

As described above, the Advanced solver is a new addition to the STAAD Analysis Engine (*) which can be used for solving both static and dynamic problems. It is part of the STAAD engine with no special command required to run it. It is automatically activated if a suitable license is available (**), however, this can be turned off and the standard solver used by including the option:-

```
SET STAR 0
```

This command must be included in the header information block at the start of the file and before the first JOINT command block.

The engine can operate in two modes, **in-core** and **out-of-core**. The in-core solver will be used for models with under 20000 joints and the out-of-core solver for models over 20000 joints. In most situations, the in-core mode will provide the quickest solution, but where there is insufficient memory available, then the engine will use the out-of-core mode. Again, selection of the mode is automatically chosen by the analysis, but can be over-ridden.

The full set of overrides for the advanced engine is:-

SET STAR -3	use in-core solver regardless of size
SET STAR 4	use out-of-core solver regardless of size
SET STAR 3	default
SET STAR 0	use STAAD standard solver

Notes:-

(*) The Advanced Solver is NOT available for use with a Stardyne Analysis.

(**) To use this feature requires access to a 'STAAD Advanced' license. If you do not currently have this feature, please contact your account manager.

(***) Global Euler Buckling analysis is different between the two solvers.

Input Instructions

5.1 Command Language Conventions

This section describes the command language used in STAAD. First, the various elements of the language are discussed and then the command format is described in detail.

5.1.1 Elements of The Commands

a) **Integer Numbers:** Integer numbers are whole numbers written without a decimal point. These numbers are designated as i_1, i_2, etc., and should not contain any decimal point. Negative signs (-) are permitted in front of these numbers. Omit the sign for positive. No spaces between the sign and the number.

b) **Floating Point Numbers:** These are real numbers which may contain a decimal portion. These numbers are designated as f_1, f_2... etc.. Values may have a decimal point and/or exponent. When specifying numbers with magnitude less than 1/100, it is advisable to use the E format to avoid precision related errors. Negative signs (-) are permitted in front of these numbers. Omit the sign for positive. No spaces between the sign and the number. Limit these to 24 characters.

Example

```
5055.32   0.73   -8.9   732
5E3   -3.4E-6
etc.
```

The decimal point may be omitted if the decimal portion of the number is zero.

c) **Alphanumeric:** These are characters, which are used to construct the names for data, titles or commands. Alphabetic characters may be input in upper or lower case letters. No quotation marks are needed to enclose them.

Example

```
MEMBER PROPERTIES
1 TO 8 TABLE ST W8X35
```

d) **Repetitive Data:** Repetitive numerical data may be provided in some (but not all) input tables such as joint coordinates by using the following format:

n*f

where n = number of times data has to be repeated
f = numeric data, integer and floating point

Example

```
JOINT COORDINATES
1  3*0.
```

This joint coordinate specification is same as:

```
1  0.  0.  0.
```

5.1.2 Command Formats

a) **Free-Format Input:** All input to STAAD is in free-format style. Input data items should be separated by blank spaces (not commas) from the other input data items. Quotation marks are never needed to separate any alphabetic words such as data, commands or titles. Limit a data item to 24 characters.

b) **Commenting Input:** For documentation of a STAAD data file, the facility to provide comments is available. Comments can be included by providing an asterisk (*) mark as the first non-blank character in any line. The line with the comment is "echoed" in the output file but not processed by the program.

Example

```
JOINT LOAD
* THE FOLLOWING IS AN EQUIPMENT LOAD
2 3 7 FY 35.0
etc.
```

c) **Meaning of Underlining in the Manual:** Exact command formats are described in the latter part of this section. Many words in the commands and data may be abbreviated. The full word intended is given in the command description with the portion actually required (the abbreviation) underlined.

For example, if the word <u>MEMB</u>ER is used in a command, only the portion MEMB need be input. It is clearer for others reading the output if the entire word is used, but an experienced user may desire to use the abbreviations.

d) **Meaning of Braces and Parenthesis:** In some command formats, braces enclose a number of choices, which are arranged vertically. One and only one of the choices can be selected. However, several of the listed choices may be selected if an asterisk (*) mark is located outside the braces.

Example

$$\left\{\begin{array}{c}\underline{\mathbf{XY}}\\ \underline{\mathbf{YZ}}\\ \underline{\mathbf{XZ}}\end{array}\right\}$$

In the above example, the user must make a choice of XY or YZ or XZ.

Example

$$*\left\{\begin{array}{c}\underline{\mathbf{FX}}\\ \underline{\mathbf{FY}}\\ \underline{\mathbf{FZ}}\end{array}\right\}$$

Here the user can choose one or all of the listing (FX, FY and FZ) in any order. Parentheses, ( ), enclosing a portion of a command indicate that the enclosed portion is optional. The presence or absence of this portion affects the meaning of the command, as is explained in the description of the particular command.

Example

```
PRINT (MEMBER) FORCES
PERFORM ANALYSIS (PRINT LOAD DATA)
```

In the first line, the word MEMBER may be omitted with no change of the meaning of the command. In the second line,

```
PRINT LOAD DATA
```

command may also be omitted, in which case the load data will not be printed.

e) **Multiple Data Separator:** Multiple data can be provided on a single line, if they are separated by a semicolon (;) character. One restriction is that a semicolon can not separate consecutive commands. They must appear on separate lines.

Example

```
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4
etc.
Possible Error:
PRINT FORCES; PRINT STRESSES
```

In the above case, only the PRINT FORCES command is processed and the PRINT STRESSES command is ignored.

f) **Listing Data:** In some STAAD command descriptions, the word "list" is used to identify a list of joints, members/elements or loading cases. The format of a list can be defined as follows:

$$\textbf{list} = \ ^{*}\left\{ \begin{array}{l} i_1 \ \ i_2 \ \ i_3 \ \ldots\ldots \\ i_1 \ \underline{\textbf{TO}} \ i_2 \ (\underline{\textbf{BY}} \ i_3) \\ \underline{\textbf{X}} \ \textbf{or} \ \underline{\textbf{Y}} \ \textbf{or} \ \underline{\textbf{Z}} \end{array} \right\}$$

TO means all integers from the first (i_1) to the second (i_2) inclusive. BY means that the numbers are incremented by an amount equal to the third data item (i_3). If BY i_3 is omitted, the increment will be set to one. Sometimes the list may be too long to fit on one line, in which case the list may be continued to the next line by providing a hyphen preceded by a blank. Also, only a list may be continued and not any other type of data.

Instead of a numerical list, a single group-name may be entered if that group was previously defined.

Instead of a numerical list, the specification X (or Y or Z) may be used. This specification will include all MEMBERs parallel to the global direction specified. Note that this is not applicable to JOINTs or ELEMENTs.

ALL, BEAM, PLATE, SOLID. Do not use these unless the documentation for a command specifically mentions them as available for that command. ALL means all members and elements, BEAM means all members, etc.

g) **Continuing a command to the next line:** Only lists may be continued to the next line by ending the line with a blank and hyphen (see above) with few exceptions: Multilinear spring supports, Supports, Master/Slave. Others have special types of continuations. Please follow the command descriptions.

Example

```
2 4 7 TO 13 BY 2 19 TO 22 -
28 31 TO 33 FX 10.0
This list of items is the same as:
2 4 7 9 11 13 19 20 21 22 28 31 32 33 FX 10.0
Possible Error:
3 5 TO 9 11 15 -
FX 10.0
```

In this case, the continuation mark for list items is used when list items are not continued. This will result in an error message or possibly unpredictable results.

5.1.3 Listing of Joints/Members/Elements by Specification of Global Ranges

This command allows the user to specify lists of joints/members/elements by providing global ranges. The general format of the specification is as follows.

General format:

$$\left\{\begin{array}{l}\underline{XR}ANGE\\ \underline{YR}ANGE\\ \underline{ZR}ANGE\end{array}\right\} \quad f_1, f_2$$

where,

XRANGE, YRANGE, ZRANGE = direction of range (parallel to global X, Y, Z directions respectively)

f1, f2 = values (in current unit system) that defines the specified range.

Notes

1) Only one range direction (XRANGE, YRANGE etc.) is allowed per list. (Exceptions: Area/Floor load and Master/Slave).
2) No other items may be in the list.
3) The values defining the range (f1, f2) must be in the current unit system.

Example

```
MEMBER TRUSS
XRANGE 20. 70.
CONSTANTS
E STEEL YRANGE 10. 55.
```

In the above example, a XRANGE is specified with values of 20. and 70. This range will include all members lying entirely within a range parallel to the global X-axis and limited by X=20 and X=70.

STAAD Commands

5.2 Problem Initiation And Title

Purpose

This command initiates the STAAD run, allows the user to specify the type of the structure and an optional title.

General format:

$$\textbf{\underline{STA}AD} \quad \left\{ \begin{array}{l} \textbf{\underline{PLA}NE} \\ \textbf{\underline{SPA}CE} \\ \\ \textbf{\underline{TRU}SS} \\ \textbf{\underline{FLO}OR} \end{array} \right\} \quad \textbf{(any title } \mathbf{a_1}\textbf{)}$$

Description

Any STAAD input has to start with the word STAAD. Following type specifications are available:

See Section 1.3

PLANE = Plane frame structure
SPACE = Space frame structure
TRUSS = Plane or space truss structure
FLOOR = Floor structure

a_1 = Any title for the problem. This title will appear on the top of every output page. To include additional information in the page header, use a comment line containing the pertinent information as the second line of input.

Notes

1) The user should be careful about choosing the type of the structure. The choice is dependent on the various degrees of freedom that need to be considered in the analysis. The following figure illustrates the degrees of freedoms considered in the various type specifications. Detailed discussions are available in Section 1.3. PLANE indicates the XY plane for Y up and the XZ plane for Z up. FLOOR indicates the XZ floor for Y up and the XY floor for Z up.

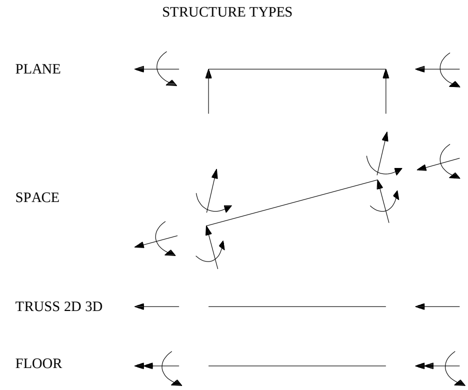


Figure 5.1

2) The optional title provided by the user is printed on top of every page of the output. The user can use this facility to customize his output.

Limits

1)	Joint number	1 to 999999
2)	Number of joints:	200000*
3)	Member/Element numbers:	1 to 999999
4)	Number of Members & Elements:	200000*
5)	Load Case numbers:	1 to 99999
6)	Number of primary & combination cases	4000
7)	Number of modes & frequencies	2700
8)	Number of load cases that may be combined by a Repeat Load or Load Combination command	550

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

5.3 Unit Specification

Purpose

This command allows the user to specify or change length and force units for input and output.

General format:

```
UNIT     * { length-unit
             force-unit }

length-unit = { INCHES
                FEET or FT or FO
                CM
                METER
                MMS
                DME
                KM }

force-unit = { KIP
               POUND
               KG
               MTON
               NEWTON
               KNS
               MNS
               DNS }
```

Note:

DME denotes Decimeters. MNS denotes mega Newtons (1,000,000 Newtons) and DNS denotes DecaNewtons (10 Newtons). MTON denotes Metric Ton (1000 kilograms). All other units are self explanatory.

Description

The UNIT command can be specified any number of times during an analysis. All data is assumed to be in the most recent unit specification preceding that data. Also, the input unit for angles is always degrees. However, the output unit for joint rotations (in joint displacement) is radians. For all output, the units are clearly specified by the program.

Example

```
UNIT KIP FT
UNIT INCH
UNIT METER KNS
UNIT CM MTON
```

Notes

This command may be used as frequently as needed to specify data or generate output in the desired length and/or force units. Mixing of different unit systems (Imperial, Metric, SI etc.) is allowed.

This command may be anywhere a primary level command (e.g., JOINT COORD, MEMBER INCIDENCE, etc.) can be. In addition, it may also be wherever a first level load command may be.

Exceptions: The second level load commands (MEMBER LOAD, JOINT LOAD and FLOOR LOAD), allow the UNIT command to be entered on a separate line, not within a continuation or between semi-colons.

5.4 Input/Output Width Specification

Purpose

These commands may be used to specify the width(s) of the lines of output file(s).

General format:

$$\left\{ \begin{array}{l} \underline{\textbf{INP}}\textbf{UT} \\ \underline{\textbf{OUT}}\textbf{PUT} \end{array} \right\} \quad \underline{\textbf{WID}}\textbf{TH} \quad i_1$$

For OUTPUT WIDTH,
i_1 = 72 or 118 depending on narrow or wide output.

Description

The user may specify the required input/output width, as required, using this command. For INPUT width, 79 is always used. The program can create output using two different output widths - 72 (default) and 118. The 72-character width may be used for display on most CRTs and for printing on “portrait” wide paper. The 118-character width may be used for printing on “landscape” wide paper.

Notes

This is a customization facility that may be used to improve the presentation quality of the run documents.

5.5 Set Command Specification

Purpose

This command allows the user to set various general specifications for the analysis/design run.

General format:

```
        { NL                 i1  }
        { DISPLACEMENT       i2  }
        { SDAMP              i3  }
        { WARP               i4  }
        { ITERLIM            i5  }
        { PRINT              i7  }
        { SHEAR                  }
SET     {         { ON  }        }
        { ECHO    {     }        }
        {         { OFF }        }
        { GUI                i6  }
        { Z                  UP  }
```

where,

i_1 = Maximum number of primary load cases (NL)

i_2 = Maximum allowable displacement tolerance for any joint in the structure.

i_3 = Damping ratio to be used for all springs in computing the modal composite damping in dynamics.

i_4= Warping restraint ratio to be used for I section members in computing the torsional rigidity. If 0.0 then no warping restraint, normal default option; If 1.0 then full warping restraint. Cw, the warping constant, will be computed and used in the torsional rigidity calculation. Values between 0.0 and 1.0 will result in a partial warping restraint.

i_5 = Maximum number of tension/compression iterations.

i_6 = 1, Bypass forming data and files needed for post-processing.

i_7 = 1, Bypass printing Zero Stiffness messages.

Description

See Sections 5.18 and 5.38

The **SET NL** command is used in a multiple analysis run if the user wants to add more primary load cases after one analysis has been performed. Specifically, for those examples, which use the CHANGE or RESTORE command, if the user wants to add more primary load cases, the NL value should be set to the maximum number with the SET NL command. The program will then be able to set aside additional memory space for information to be added later. This command should be provided before any joint, member or load specifications. The value for i_1 should not be greater than the maximum number of primary load cases.

For PDELTA ANALYSIS with CONVERGE option the **SET DISPLACEMENT** command is used to specify the convergence tolerance. If the RMS displacement of two consecutive iterations changes less than the value entered, then that load case is converged. This command should be placed before the JOINT COORDINATE specification.

The **SET ECHO ON** command will activate and the SET ECHO OFF command will deactivate the echoing of input file commands in the output file. In the absence of the SET ECHO command, input file commands will be echoed back to the output file.

By default, the Y-axis is the vertical axis. However, the **SET Z UP** command may be used to model situations where Z-axis represents the vertical axis (direction of gravity load) of the structure. This situation may arise if the input geometry is created through some CAD software. This command will affect the default BETA angle specification. However, BETA can be set to a certain value for all members parallel to a particular global axis by using the MEMBER X (or Y or Z) type of listing. For additional information, see the CONSTANTs specification (Section 5.26).

Notes for SET Z UP

The SET Z UP Command directly influences the values of the following input:

1) JOINT COORDINATE
2) Input for the PERFORM ROTATION Command
3) BETA ANGLE

The following feature of STAAD cannot be used with the SET Z UP command:

Automatic Generation of Spring Supports for Mat Foundations

The **SET SDAMP** command will allow the damping of springs to be considered in computing the composite modal damping for each mode in a dynamic solution. This command is not used unless CDAMP ratios are also entered for the members and elements in the CONSTANTS command. Composite damping is generally only used if there are many modes in the dynamic solution and there are a wide range of damping ratios in the springs / members / elements.

The **SET WARP** command will allow the I section member end warping restraint to be considered in calculating the torsional stiffness rigidity. Full or partial or no warping restraint are allowed.

The **SET ITERLIM** command is for raising the max. iteration limit above the default of 10 in tension/compression iterations. Since this iterative procedure will not necessarily converge, this option of more iterations may not help and should be used with caution. The minimum iteration limit that may be entered is 3.

After any tension/compression analysis, the output file (.ANL) should be scanned for warnings of non-convergence. Do not use results from non-converged cases.
The **SET PRINT 1** command is for eliminating the Zero Stiffness messages.

The **SET SHEAR** command is for omitting the additional pure shear distortion stiffness terms in forming beam member stiffnesses. With this command you can exactly match simple textbook beam theory results.

After the calculations are completed, and before the Analysis window is closed, the program creates several files for the purpose of displaying results in the post processing mode. In large models, this can be a time consuming process. If the user's goal is to look at results in the output file only (.ANL file) and does not intend to go into the post-processing mode, he/she could instruct the program to skip the process of creating those files. The "SET GUI 1" may be specified immediately before the FINISH command, or somewhere near the beginning of the file after STAAD SPACE.

Other rarely used SET commands:

SET DATA CHECK	IGNORED
SET RUN	IGNORED
SET COMPRESS	TURN OFF FILE COMPRESSION
SET SOLUTION INCORE	use determinant search for frequencies for small problems
SET SOLVER	should ignore
SET CORE	should ignore (Memory)
SET EXM	should ignore (EXTENDED MEMORY)
SET NJ	should ignore
SET NM	should ignore
SET CONNECTIVITY	should ignore
SET MASS	=1 use generated moments as masses.
SET MODAL	should ignore
SET THISTORY	=2 Use exact force integration in time history.
SET INTERPOLATION	Lin or Log for spectra
SET DISPLACEMENT METHOD	should ignore
SET ???? ??? file extension for L43	
SET BUBBLE	=1 do not use bubble fns in solids
SET NOSECT	Beam section results will not be calculated for the post-processing mode
SET TMH	should ignore
SET SSVECT	to instruct the program to use a different initial set of trial vectors for eigensolution. May be used if eigen extraction fails.
SET INCLINED REACTION	to obtain reactions at inclined supports in the inclined axis system

5.6 Separator Command

Purpose

This command may be used to specify the desired separator character that can be used to separate multiple lines of data on a single line of input.

General format:

SEPARATOR a_1

Description

See Section 5.1.2

The semicolon (;) is the default character which functions as the separator for multiple line data on one line. However, this separator character can be changed by the SEPARATOR command to any character a_1, other than the comma or asterisk.

Notes

Comma (,) or asterisk (*) may not be used as a separator character.

5.7 Page New Command

Purpose

This command may be used to instruct the program to start a new page of output.

General format:

PAGE NEW

Description

With this command, a new page of output can be started. This command provides the flexibility, the user needs, to design the output format.

Notes

The presentation quality of the output document may be improved by using this command properly.

5.8 Page Length/Eject Command

Purpose

These commands may be used to specify the page length of the output and the desired page eject character.

General format:

```
PAGE    { LENGTH   i  }
        { EJECT    a1 }
```

The page length in STAAD output is based on a default value of 60 lines . However, the user may change the page length to any number i (number of lines per page) desired.

Description

Standard page eject character (CNTRL L for PCs and 1 for Mini/Mfrm) is embedded in the STAAD program. The PAGE EJECT command with the input of the character a_1 will alter the default page eject character in the program. A blank character will suppress page ejection.

5.9 Ignore Specifications

Purpose

This command allows the user to provide member lists in a convenient way without triggering error messages pertaining to non-existent member numbers.

General format:

IGNORE LIST

Description

IGNORE LIST may be used if the user wants the program to ignore any nonexistent member that may be included in a member list specification. For example, for the sake of simplicity, a list of members may be specified as MEMB 3 TO 40 where members 10 and 11 do not exist. An error message can be avoided in this situation by providing the IGNORE LIST command anywhere in the beginning of input. A warning message, however, will appear for each nonexistent member.

5.10 No Design Specification

Purpose

This command allows the user to declare that no design operations will be performed during the run. The memory reserved for design will be released to accommodate larger analysis jobs.

General format:

INPUT NODESIGN

Description

STAAD always assumes that at some point in the input, the user may wish to perform design for steel or concrete members. These design processes require more computer memory. If memory availability is a problem, the above command may be used to eliminate extra memory requirements.

5.11 Joint Coordinates Specification

Purpose

These commands allow the user to specify and generate the coordinates of the JOINTs of the structure. The JOINT COORDINATES command initiates the specification of the coordinates. The REPEAT and REPEAT ALL commands allow easy generation of coordinates using repetitive patterns.

General format:

JOINT COORDINATES (CYLINDRICAL (REVERSE)) (NOCHECK) band-spec

i_1, x_1, y_1, z_1, (i_2, x_2, y_2, z_2, i_3)
REPEAT n, xi_1, yi_1, zi_1, (xi_2, yi_2, zi_2,..., xi_n, yi_n, zi_n)
REPEAT ALL n, xi_1, yi_1, zi_1, (xi_2, yi_2, zi_2,..., xi_n, yi_n, zi_n)

n is limited to 150

JTORIG xOrigin yOrigin zOrigin
band-spec = (NOREDUCE BAND)

NOCHECK= Do not perform check for multiple structures or orphan joints.

Description

See Section 1.5.1

The command JOINT COORDINATES specifies a Cartesian Coordinate System (see Figure 1.2). Joints are defined using the global X, Y and Z coordinates. The command JOINT COORDINATES CYLINDRICAL specifies a Cylindrical Coordinate System (see Figure 1.3). Joints are defined using the r, θ and z coordinates. JOINT COORDINATES CYLINDRICAL REVERSE specifies a Reverse Cylindrical Coordinate system (see Figure 1.4). Joints are defined using the r, y and θ coordinates.

JTORIG causes the program to use a different origin than (0, 0, 0) for all of the joints entered with this JOINT COORDINATES command. It is useful in instances such as when the center of cylinder is not at (0, 0, 0) but at a different point in space. The JTORIG command should be entered on a separate command line. Basically after the joint coordinates are entered or generated, then the xOrigin, yOrigin, and zOrigin values are added to the coordinates. For example a cylinder could be generated about the Y axis then moved by this command to its proper place. To create multiple offset structural parts, enter additional JOINT COORDINATES commands, each one followed by its JTORIG command. An example showing the use of this command is provided later in this section.

The multiple JOINT COORDINATES command concept allows UNIT changes and PERFORM ROTATION commands in between, such that these commands would apply to a selected portion of the joints. However, the PERFORM ROTATION command applies to all prior defined joints, not just those in the previous JOINT COORDINATE command.

NOREDUCE BAND causes the program to execute without performing a bandwidth reduction.

Example

```
JOINT COORDINATES NOREDUCE BAND
```

The REPEAT command causes the previous line of input to be repeated ‘n’ number of times with specified coordinate increments. The REPEAT ALL command functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT ALL command, or all joint data if no previous REPEAT ALL command has been given. Note: Use “REPEAT ALL 0” to start a section of data to be repeated if necessary. (When using the REPEAT and REPEAT ALL commands, joint numbering must be consecutive and should begin with 1.)

* i_1 =	The joint number for which the coordinates are provided. Any integer number within the limit (see section 5.2 for limit) is permitted.
x_1, y_1 and z_1 =	X, Y & Z (R, θ & Z for cylindrical or R, Y & θ for cylindrical reverse) coordinates of the joint.

For PLANE analyses z_1 is an optional data item when defining input for individual joints. z_1 is always required for joint generation. The following are used only if joints are to be generated.

* i_2 =	The second joint number to which the joint coordinates are generated.
x_2, y_2, and z_2 =	X, Y & Z (R, θ & Z for cylindrical or R, Y & θ for cylindrical reverse) coordinates of the joint i_2.
i_3 =	Joint number increment by which the generated joints will be incremented. Defaults to 1 if left out.
n =	Number of times repeat is to be carried out. “n” cannot exceed 150 in any one single REPEAT command.
xi_k, yi_k & zi_k =	X, Y & Z (R, θ & Z [R, Y & θ]) coordinate increments for k th repeat.

The X, Y and Z (R, θ & Z [R, Y & θ]) coordinates will be equally spaced between i_1 and i_2.

* The REPEAT command uses the highest joint number entered so far plus one for the intermediate generated joint numbers.

Example 1

```
JOINT COORDINATES
1  10.5  2.0  8.5
2  0.0  0.0  0.0
3  5.25  0.0  8.5  6  50.25  0.0  8.5
```

In this example, X Y Z coordinates of joints 1 to 6 are provided. The joints between 3 & 6 will be generated with joints equally spaced from 3 to 6. Hence, joint 4 will have coordinates of 20.25 0.0 8.5 and joint 5 will have coordinates of 35.25 0.0 8.5.

Example 2

```
JOINT COORDINATES
1  0.0  0.0  0.0  4  45  0.0  0.0
REPEAT  4  0.0  0.0  15.0
REPEAT ALL  10  0.0  10.0  0.0
```

Here, the 220 joint coordinates of a ten story 3 X 4-bay structure are generated. The REPEAT command repeats the first input line 4 times, incrementing each Z coordinate by 15. Thus, the first 2 lines are sufficient to create a "floor" of twenty joints.

```
1  0.  0.  0.  ;  2  15.  0.  0.  ;  3  30.  0.  0.  ;  4  45.  0.  0.
5  0.  0.  15.  ;  6  15.  0.  15.  ;  7  30.  0.  15.  ;  8  45.  0.  15.
.....    .....       .....       .....
17  0.  0.  60.  ;  18  15.  0.  60.  ;  19  30.  0.  60.  ;  20  45.  0.  60.
```

The REPEAT ALL command repeats all previous data (i.e. the 20 joint "floor") ten times, incrementing the Y coordinate by 10 each time. This creates the 200 remaining joints of the structure:

Example 3

```
21  0.0  10.0  0.0    ;  22  15.0  10.0  0.0  ;  ...  ;

40  45.0  10.0  60.0  ;  41  0.0  20.0  0.0  ;  ...  ;
200  45.0  90.0  60.0  ;  201  0.0  100.0  0.0  ;  ...  ;
219  30.0  100.0  60.0  ;  220  45.0  100.0  60.0
```

The following examples illustrate various uses of the REPEAT command.

```
REPEAT 10  5. 10. 5.
```

The above REPEAT command will repeat the last input line 10 times using the same set of increments (i.e. x = 5., y = 10., z = 5.)

```
REPEAT 3  2. 10. 5. 3. 15. 3. 5. 20. 3.
```

The above REPEAT command will repeat the last input line three times. Each repeat operation will use a different increment set.

```
REPEAT 10  0. 12. 0. 15*0 0. 10. 0. 9*0
```

The above REPEAT command will repeat the last input line 10 times; six times using x, y and z increments of 0., 12. and 0., and four times using increments of 0., 10. and 0. Each x, y and z value of 0 represents no change from the previous increment. To create the 2nd through 6th repeats, five sets of 0., 0. and 0. (15*0) are supplied. The seventh repeat is done with increments of 0., 10. and 0. The 8th through 10th repeats are done with the same increments as 7, and is represented as 9*0.

Notes

The PRINT JOINT COORDINATE command may be used to verify the joint coordinates provided or generated by REPEAT and REPEAT ALL commands. Also, use the Post Processing facility to verify geometry graphically.

5.12 Member Incidences Specification

Purpose

This set of commands is used to specify MEMBERs by defining connectivity between JOINTs. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The member/element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD is capable of detecting multiple structures automatically.

General format:

MEMBER INCIDENCES
i_1, i_2, i_3, (i_4, i_5, i_6)
REPEAT n, m_i, j_i
REPEAT ALL n, m_i, j_i

Description

See Section 1.5.2

The REPEAT command causes the previous line of input to be repeated 'n' number of times with specified member and joint increments. The REPEAT ALL command functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT ALL command or to the beginning of the specification if no previous REPEAT ALL command has been issued. (When using REPEAT and REPEAT ALL commands, member numbering must be consecutive).

i_1 = Member number for which incidences are provided. Any integer number (maximum six digits) is permitted.
i_2 = Start joint number.
i_3 = End joint number.

Note: Use "REPEAT ALL 0", to start a set of members that will be repeated if you don't want to repeat back to the last REPEAT ALL.

The following data are used for member generation only:

i_4 = Second member number to which members will be generated.

i_5 = Member number increment for generation.

i_6 = Joint number increment which will be added to the incident joints. (i_5 and i_6 will default to 1 if left out.)

n = Number of times repeat is to be carried out.

m_i = Member number increment

j_i = Joint number increment

Example

```
MEMBER INCIDENCES
1  1  2
2  5  7  5
7  11  13  13  2  3
```

In this example, member 1 goes from joint 1 to 2. Member 2 is connected between joints 5 and 7. Member numbers from 3 to 5 will be generated with a member number increment of 1 and a joint number increment 1 (by default). That is, member 3 goes from 6 to 8, member 4 from 7 to 9, member 5 from 8 to 10. Similarly, in the next line, member 9 will be from 14 to 16, 11 from 17 to 19 and 13 from 20 to 22.

Additional example

```
MEMBER INCIDENCES
1  1  21  20
21  21  22  23
REPEAT  4  3  4
36  21  25  39
REPEAT  3  4  4
REPEAT  ALL  9  51  20
```

This example creates the 510 members of a ten story 3 X 4-bay structure (this is a continuation of the example started in Section 5.12). The first input line creates the twenty columns of the first floor:

```
1 1 21 ; 2 2 22 ; 3 3 23 ; ... ; 19 19 39 ; 20 20 40
```

The two commands (21 21 22 23 and REPEAT 4 3 4) create 15 members which are the second floor "floor" beams running, for example, in the east-west direction:

```
21 21 22; 22 22 23; 23 23 24
24 25 26; 25 26 27; 26 27 28
...   ...   ...
33 37 38; 34 38 39; 35 39 40
```

The next two commands (36 21 25 39 and REPEAT 3 4 4) function similar to the previous two commands, but here create the 16 second floor "floor" beams running in the north-south direction:

```
36 21 25; 37 22 26; 38 23 27; 39 24 28
40 25 29; 41 26 30; 42 27 31; 43 28 32
...   ...   ...   ...
48 33 37; 49 34 38; 50 35 39; 51 36 40
```

The preceding commands have created a single floor unit of both beams and columns, a total of 51 members. The REPEAT ALL now repeats this unit nine times, generating 459 new members and finishing the ten story structure. The member number is incremented by 51 (the number of members in a repeating unit) and the joint number is incremented by 20, (the number of joints on one floor).

Notes

The PRINT MEMBER INFO command may be used to verify the member incidences provided or generated by REPEAT and REPEAT ALL commands.

Also, use the Post Processing facility to verify geometry graphically.

5.13 Elements and Surfaces

This section describes the commands used to specify:

a. Plate and Shell elements (see section 5.13.1).
b. Solid elements (see section 5.13.2).
c. Surface entities (see section 5.13.3).

5.13.1 Plate and Shell Element Incidence Specification

Purpose

This set of commands is used to specify ELEMENTs by defining the connectivity between JOINTs. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD is capable of detecting multiple structures automatically.

General format

ELEMENT INCIDENCES (SHELL)

i_1, i_2, i_3, i_4, (i_5), (TO i_6, i_7, i_8)

REPEAT n, e_i, j_i
REPEAT ALL n, e_i, j_i

Description

See Section 1.6

ELEMENT INCIDENCES SHELL must be provided immediately after MEMBER INCIDENCES (if any) are specified. The REPEAT command causes the previous line of input to be repeated 'n' number of times with specified element and joint increments. The REPEAT ALL command functions similar to the REPEAT command, except that it repeats all previously specified input back to the most recent REPEAT ALL command; or to the beginning of the specification if no previous REPEAT ALL command had been issued. Use "REPEAT ALL 0 0 0" to start a set of elements that will be repeated if you don't want to repeat back to the last REPEAT ALL.

i_1 = Element number (any number up to six digits). If MEMBER INCIDENCE is provided, this number must not coincide with any MEMBER number.

$i_2 \ldots i_5$ = Clockwise or counterclockwise joint numbers which represent the element connectivity. i_5 is not needed for triangular (3 noded) elements.

The following data is needed if elements are to be generated:

i_6 = Last element number to which elements are generated.

i_7 = Element number increment by which elements are generated. Defaults to 1 if omitted.

i_8 = Joint number increment which will be added to incident joints. Defaults to 1 if omitted.

The following data is needed if REPEAT or REPEAT ALL commands are used to generate elements:

n = Number of times repeat is to be carried out.

e_i = Element number increment.

j_i = Joint number increment.

Example

```
ELEMENT INCIDENCE
1  1  2  7  6
2  3  4  8
3  8  9  11  10  TO  8
9  1  3  7  TO  14
```

Notes

The PRINT ELEMENT INFO command may be used to verify the element incidences provided or generated by REPEAT and REPEAT ALL commands.

Also, use the Post Processing facility to verify geometry graphically.

5.13.2 Solid Element Incidences Specification

Purpose

4 through 8 noded elements, also known as solid elements, are described using the commands described below. Technical information on these elements is available in section 1.6.2 of this manual.

General format

The element incidences for solid elements are to be identified using the expression SOLID to distinguish them from PLATE/SHELL elements.

ELEMENT INCIDENCES SOLID

i_1, i_2, i_3, i_4, i_5, i_6, i_7, i_8, i_9, (TO i_{10}, i_{11}, I_{12})

REPEAT n, e_i, j_i
REPEAT ALL n, e_i, j_i

Description

ELEMENT INCIDENCES SOLID must be provided immediately after MEMBER INCIDENCES (if any) are specified as well as after the ELEMENT INCIDENCES SHELL (if any).

i_1	=	Element number
$i_2 \ldots i_9$	=	Joint numbers of the solid element
i_{10}	=	Last element number to be generated
i_{11}	=	Element number increment
i_{12}	=	Joint number increment
n	=	Number of times REPEAT is to be carried out
e_i	=	Element number increment
j_i	=	Joint number increment

Specify the four nodes of any of the faces of the solid element in a counter-clockwise direction as viewed from the outside of the element and then go to the opposite face and specify the four nodes of that face in the same direction used in specifying the nodes of the first face. The opposite face must be behind the first face, as defined by the right hand rule, i.e. the opposite (back) face points to the first (front) face, which points to the viewer.

Use “REPEAT ALL 0” to start a set of solids that will be repeated if you don’t want to repeat back to the last REPEAT ALL.

Example

```
ELEMENT INCIDENCE SOLID
1  1  5  6  2  21  25  26  22  TO  3
4  21  25  26  22  41  45  46  42  TO  6
```

5.13.3 Surface Entities Specification

Purpose

In order to facilitate rapid modeling of complex walls and slabs, a type of entity called **Surface** is available. At the modeling level, it corresponds to the entire structural part, such as a wall, floor slab or bridge deck. At the analysis level, it is first decomposed into a number of quadrilateral plate elements. Thus the Surface is a superelement for modeling purposes (it is composed from a number of plate elements). Consequently, the user has the convenience of specifying only one large structural component per wall or slab, yet may maintain full control over the computational accuracy by setting the desired number of finite element divisions. Surfaces may include rectangular openings.

The attributes associated with the surface element, and the sections of this manual where the information may be obtained, are listed below:

Attributes	**Related Sections**
Surfaces incidences -	5.13.3
Openings in surfaces -	5.13.3
Local coordinate system for surfaces -	1.6.3
Specifying sections for stress/force output -	5.13.3
Property for surfaces -	5.21.2
Material constants -	5.26.3
Surface loading -	5.32.3.4
Stress/Force output printing -	5.42
Shear Wall Design -	3.8.2, 5.54

General Format:

SET DIVISION m

<u>SUR</u>FACE <u>INC</u>IDENCE n1, ..., ni <u>SUR</u>FACE s <u>DIV</u>ISION sd1, ..., sdj -

<u>RECO</u>PENING x1 y1 z1 x2 y2 z2 x3 y3 z3 x4 y4 z4 <u>DIV</u>ISION od1, ..., odk

where:

m - number of segments to be generated between each pair of adjacent nodes

n1, ..., ni - node numbers defining the perimeter of the surface,

s - surface ordinal number,

sd1, ..., sdj - number of divisions for each of the node-to-node distance on the surface perimeter,

x1 y1 z1 (...)- coordinates of the corners of the opening,

od1, ..., odk - divisions along edges of the opening.

The SET DIVISION command specifies a default number of generated meshing divisions for all node to node segments. In its absence, that number is taken as 10.

If the sd1, ..., sdj or the od1, ..., odk list does not include all node-to-node segments, or if any of the numbers listed equals zero, then the corresponding division number is set to the default value (=10, or as previously input by the SET DIVISION command).

Notes

1. The surface definition must comprise a minimum of four nodal points forming corners of a rectangle. However, any number of additional nodes may be incorporated into the

surface boundaries provided the nodes are collinear on edges they belong to. In addition, the user specifies the number of edge divisions that will be the basis for mesh generation. A single command per wall is used for this purpose. The program will subdivide all edges into the requested number of fragments and each of these fragments will become an edge of a plate element. However, if the original surface edges have additional nodal points between the corners, all node-to-node lengths of the surface edge will be divided into the same number of fragments.

2. Surface thickness and material constants are specified in a manner similar to that for plate elements, except that, currently, only a constant surface thickness is supported.
3. A support generation function allows quick assignment of support specifications to multiple nodal points.
4. Surface elements may be loaded by uniformly distributed loads in any global direction or by loads normal to the plane.
5. It is possible to obtain in-plane bending moments as well as stresses along any arbitrary line cutting the surface.

Example

```
SET DIVISION 12
SURFACE INCIDENCES
2 5 37 34 SUR 1
34 37 54 51 SUR 2
19 16 65 68 SUR 3 DIV 10 16 10 16 –
RECO 5.8 1.5 6.9 6.4 1.5 6.9 6.4 0.5 6.9 5.8 0.5 6.9 DIV 5 10 5 10
```

This example illustrates definition of three Surface elements. SET DIVISION 12 command establishes a default number of boundary divisions for automatic mesh generation. This command will apply to outer edges of the elements as well as to the edges of openings, if present.

SURFACE INCIDENCES command start the specifications of the elements. SUR 1 and SUR 2 commands define Surface elements No. 1 and 2 with default boundary divisions and no openings. SUR 3 command defines Surface No. 3 with non-default edge divisions and one opening. The DIV command following SUR 3 defines Surface element edge divisions. Non-default opening edge divisions are defined by the DIV command following the RECO command.

5.14 Plate Element Mesh Generation

Purpose

There are several methods available in STAAD to model panel type entities like walls or slabs as an assembly of plate elements. This process is called meshing.

Two of those methods have a set of commands which can be provided in the STAAD input file. The first method, which is described in section 5.14.1 is based entirely on commands in the input file alone, and does not have any graphical interface for creation or modification.

The second method is referred to as the Parametric mesh generator and is best used from STAAD's graphical screens. The aspect of this method, which enables commands to be written into the input file, is described in section 5.14.2.

5.14.1 Element Mesh Generation

Purpose

This set of commands is used to generate finite element meshes. The procedure involves the definition of super-elements, which are subsequently divided into smaller elements.

Description

This is the second method for the generation of element incidences. If the user needs to divide a big element into a number of small elements, he may use this facility which generates the joint numbers and joint coordinates, the element numbers and the element incidences automatically. Use of this feature consists of two parts:

1. Definition of the super-element boundary points: A super-element may be defined by either 4 boundary points or 8 boundary points (see Figure on next page). A boundary point is denoted by a unique alphabet (A-Z in upper case or a-z in lower case) and its corresponding coordinates. Hence, any 4 or 8 of the 52 characters may be used to define the super-element boundary. If 4 points are used to define the super-element, each side of the super-element will be assumed to have a straight edge connecting the 2 points defining that side. If 8 points are used, each side will be a smooth curve connecting the 3 points defining that side.
2. Generation of sub-elements: define the super-element using boundary points (4 or 8 as explained above) and specify the total number of sub-elements required.

General Format:

DEFINE MESH ({ CYL / RCYL } (x_o,y_o,z_o))
A_i x_i y_i z_i
. . .
A_j x_j y_j z_j

GENERATE ELEMENT { (QUADRILATERAL) / TRIANGULAR }

MESH A_i A_j ..... n_1 (n_2)
MESH A_m A_n ..... n_3 (n_4)
....

(up to 21 MESH input lines)

where

A_i, A_j = Alphabets A - Z or alphabets a - z. Maximum is 52.

x_i,y_i,z_i = Coordinates for boundary point A_i.
If CYL or RCYL is defined, above coordinates will be in cylindrical or reverse cylindrical coordinates system. Optional coordinates x_o, y_o and z_o will be the cartesian coordinates for the origin of the cylindrical coordinates. Defaults to 0, 0, 0 if not provided.
The 3 fields (x, y, z) may be replaced by a joint number whose coordinates have been defined in the JOINT COORDINATE command by entering

A_i JOINT jn instead.

A_i,A_j,A_k... = A rectangular super-element defined by four or eight boundary points. There should be no spaces between the letters.

n_1 = Number of elements along the side A_i A_j of the super-element. (Must not exceed 28).

n_2 = Number of elements along the side A_j A_k of the super-element. (Must not exceed 28).

If n_2 is omitted, that is, only n_1 is provided, then n_1 will indicate the total number of elements within the super-element. In this case, n_1 must be the square of an integer.

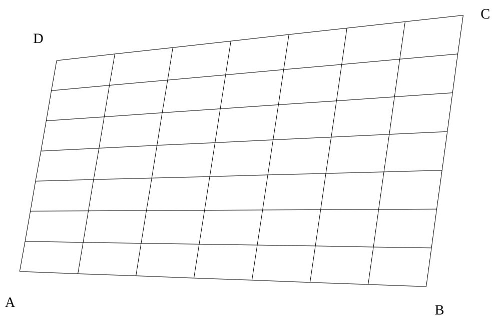
Limits

There is a limit of 21 Mesh commands. Up to 33000 joints may be generated and up to 67000 elements. Total number of joints in the model after this command is completed may not exceed 100,000.

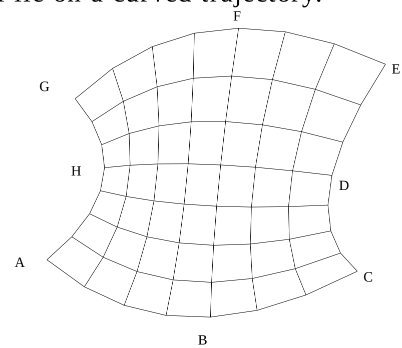
Notes

All coordinates are in current unit system. While using this facility the user has to keep the following points in mind:

1. All super-elements must be 4-noded or 8-noded. Generated elements for 4-noded super-elements will retain the straight-line edges of the super-elements, while joints of elements generated from 8-noded super-elements will lie on a curved trajectory.



Mesh generated for a 4-noded super-element



Mesh generated for a 8-noded super-element

Figure 5.2

2. 2 super-elements, which have a common boundary, must have the same number of elements along their common boundary.

3. Sequence of super-elements - MESH commands define the super-elements. The sequence of this MESH command should be such that once one is defined, the next super-elements should be the ones connected to this. Therefore, for convenience, the first super-element should be the one, which is connected by the largest number of super-elements. In the example shown here for the tank, the bottom super-element is specified first.

4. This command must be used after the MEMBER INCIDENCE & ELEMENT INCIDENCE section and before the MEMBER PROPERTIES & ELEMENT PROPERTIES section. The elements that are created internally are numbered sequentially with an increment of one starting from the last member/element number plus one. Similarly the additional joints created internally are numbered sequentially with an increment of one starting from the last joint number plus one. It is advisable that users keep the joint numbers and member/element numbers in a sequence with an increment of one starting from one.

5. If there are members embracing a super-element which is being meshed, the users will have to take care of the required additions/modifications in the MEMBER INCIDENCE section themselves since a few more new joints might appear on the existing common boundary as a result of meshing the super-element. See the following figure:

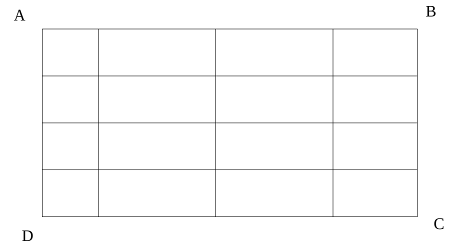


Figure 5.3

Note : If a member exists between points A and B, the user must breakup this member into 4 parts. Members will not be meshed automatically.

6. The sub-elements will have the same direction (Clockwise or Anti-clockwise) as the super-elements. For a super-element bounded by four points A, B, C and D, if ABCD, BCDA etc. are in clockwise direction, CBAD or DCBA etc. are in anti-clock wise direction. If the particular super-element is denoted as ABCD, all the sub-elements in it will have a clockwise element incidence in this example.

7. Element incidences of the generated sub-elements may be obtained by providing the command 'PRINT ELEMENT INFORMATION' after the 'MESH...' command in the input file.

8. If the STAAD input file contains commands for JOINT COORDINATES, MEMBER INCIDENCES, ELEMENT INCIDENCES and MESH GENERATION, they should be specified in the following order:

```
STAAD SPACE
UNIT . . .
JOINT COORDINATES
...
MEMBER INCIDENCES
...
ELEMENT INCIDENCES
...
DEFINE MESH
...
GENERATE ELEMENT
...
```

9. Newly created joints will be merged with existing joints if they are within 0.001 inches of each other.

Example

The following section of input illustrates the use of MESH GENERATION facility, the user may compare this with the geometry inputs for Example Prob. No. 10 in the STAAD example manual:

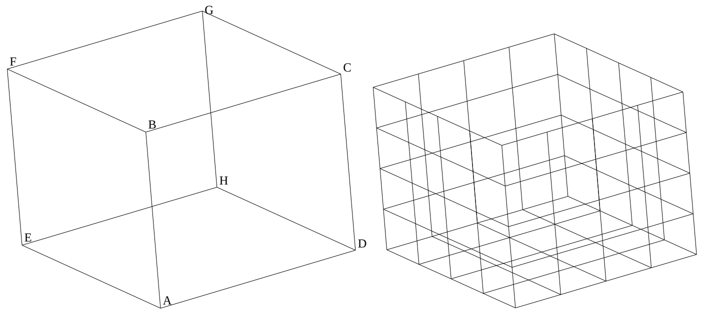
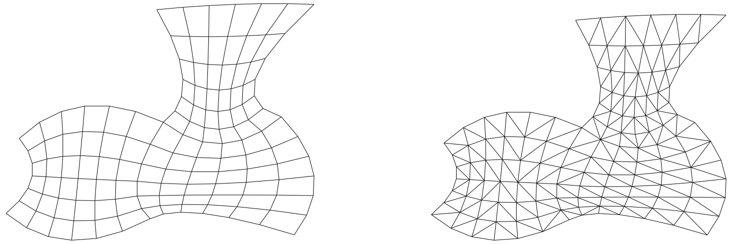


Figure 5.4

```
STAAD SPACE TANK STRUCTURE WITH
* MESH GENERATION
UNIT  . . .
DEFINE MESH
A 0 0 0 ; B 0 20 0 ; C 20 20 0
D 20 0 0 ; E 0 0 -20 ; F 0 20 -20
G 20 20 -20 ; H 20. 0. -20
GENERATE ELEMENT
MESH AEHD 16
MESH EABF 16
MESH ADCB 16
MESH HEFG 16
MESH DHGC 16
```

Typical generated Quad and Triangular elements:



Typical generated Quad elements

Typical generated Triangular elements

Figure 5.5

5.14.2 Persistency of Parametric Mesh Models in the STAAD Input File

Purpose

There is a feature in STAAD's graphical model generation facilities called the Parametric Model. It is meant for creating a plate element mesh and is described in Section 2.3.6.11 of the STAAD Graphical Environment manual.

In the past, once the parametric mesh model was merged with the base model, no information about the parametric mesh was retained by STAAD. So, if any modification was required at a later stage, the parametric mesh had to be created afresh. The Parametric model feature has now been enhanced and multiple parametric mesh models can now be saved as part of the STAAD model. This gives the users the flexibility to come back to the saved mesh models at any time and make modifications to it like adding an opening or adding a density line.

Description

Special tag based commands have been introduced to support saving of parametric mesh models as part of the STAAD input file as shown below.

```
2072 1114 1113 1160; 2073 1045 1160 1113;
ELEMENT PROPERTY
810 TO 1779 1821 TO 2073 THICKNESS 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
PARAMETRIC MODEL second_floor
MESH PARAM 0 3
MESH ORG 3 5 8
BOUNDARY 10
11 1 93 1 94 1 95 1 83 1 71 1 70 1 69 1 41 1 26 1
```

```
OPENING CIRC 72 360 96 43.2666 12
OPENING POLY 5
216 360 67.2 1 270 360 33.6 2 324 360 67.2 2 270 360 100.8 2 216 360 100.8 2
DENSITY POINTS 2
180 360 168 1 360 360 168 1
DENSITY LINE  0 360 168 100 180 360 168 200
DENSITY LINE 180 360 168 1 360 360 168 1
DENSITY LINE 360 360 168 1 540 360 168 1
DENSITY LINE 180 360 0 1 180 360 168 1
DENSITY LINE 180 360 168 1 180 360 336 1
DENSITY LINE 360 360 0 1 360 360 168 1
DENSITY LINE 360 360 168 1 360 360 336 1
DENSITY LINE 54 360 302.4 1 162 360 201.6 1
DENSITY LINE 216 360 201.6 1 324 360 235.2 1
GENERATED PLATES ALL
END
<! STAAD PRO GENERATED DATA DO NOT MODIFY!!!
PARAMETRIC MODEL roof
MESH PARAM 60 3
MESH ORG 2 3 5
BOUNDARY 6
36 1 65 1 66 1 53 1 52 1 51 1
GENERATED PLATES ALL
END
!> END GENERATED DATA BLOCK
!> END GENERATED DATA BLOCK
DEFINE MATERIAL START
ISOTROPIC STEEL
```

In the STAAD GUI, go to the **Geometry** | **Parametric Models** page and the saved parametric mesh models will appear within the **Parametric Models** dialog box as shown in the next figure.

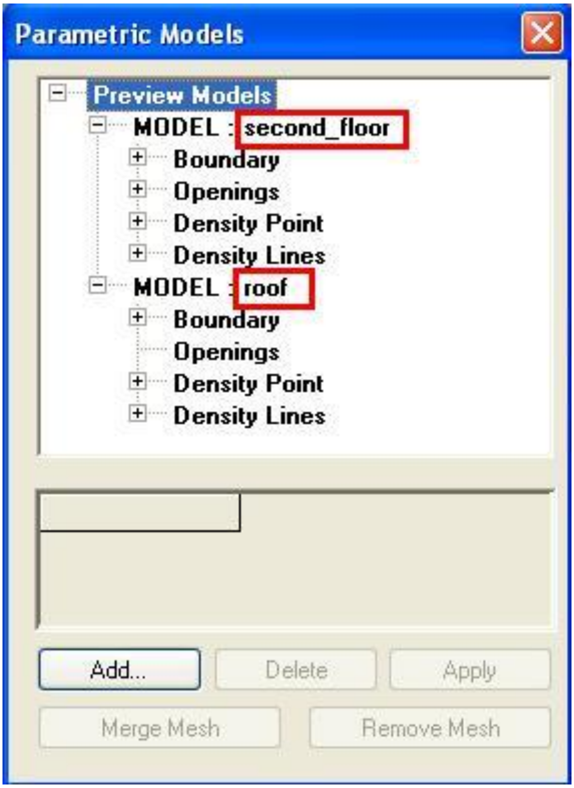


Figure 12

There are two parametric mesh models named **second_floor** and **roof** inside the **Parametric Models** dialog box as shown in the previous figure.

5.15 Redefinition of Joint and Member Numbers

Purpose

This command may be used to redefine JOINT and MEMBER numbers. Original JOINT and MEMBER numbers are substituted by new numbers.

General Format:

$$\mathbf{\underline{SUB}ST}\ \left\{\begin{array}{l}\left\{\begin{array}{l}\mathbf{\underline{JOI}NT}\\ \mathbf{\underline{MEM}BER}\end{array}\right\}\quad\left\{\begin{array}{l}\mathbf{\underline{XR}ANGE}\\ \mathbf{\underline{YR}ANGE}\\ \mathbf{\underline{ZR}ANGE}\end{array}\right\}\\ \mathbf{\underline{COL}UMN}\end{array}\right\}\quad \mathbf{f_1, f_2}\ \mathbf{\underline{STA}RT}\ \mathbf{i}$$

where, f_1 and f_2 are two range values of x, y, or z and i is the new starting number.

Description

Joint and member numbers can be redefined in STAAD through the use of the SUBSTITUTE command. After a new set of numbers is assigned, input and output values will be in accordance with the new numbering scheme. The user can design numbering schemes that will result in simple input specification as well as easy interpretation of results. For example, all joints in first floor of a building may be renumbered as 101, 102 …., all second floor joints may be renumbered as 201, 202 ….., etc.

Example

```
UNIT METER
SUBST JOINT YR  9.99  10.0 START  101
SUBST COLUMN START  901
```

Joints with Y coordinates ranging from 9.99 to 10 meters will have a new number starting from 101. Columns will be renumbered starting with the new number 901.

Note

Meaningful respecification of JOINT and MEMBER numbers may significantly improve ease of interpretation of results.

This command may be in between incidence commands:

MEMBER INCIDENCE
SUBSTITUTE
ELEMENT INCIDENCE

5.16 Entities as single objects

In the mathematical model, beams, columns, walls, slabs, block foundations, etc. are modelled using a collection of segments, which are known by the names members, plate elements, solid elements, etc. Hence, the bottom chord of a truss may be modelled using 5 members, with each member representing the segment between points where diagonals or vertical braces meet the bottom chord.

Often, it is convenient to cluster these segments under a single name so that assignment of properties, loads, design parameters. etc. is simplified. There are presently two options in STAAD for clustering entities - Group names and Physical members.

5.16.1 Listing of entities (Members / Elements / Joints, etc.) by Specification of GROUPS

This command allows the user to specify a group of entities such as joints, members, plate & solid elements and save the information using a 'group-name'. The 'group-name' may be subsequently used in the input file instead of a member/element/joint list to specify other attributes. This very useful feature allows avoiding of multiple specifications of the same member/joint list. Following is the general format required for the GROUP command.

General format:

START GROUP DEFINITION

(GEOMETRY)
_(group-name) member/element/solid-list
......   (default)

OR

JOINT
_(group-name) joint-list
......
MEMBER
_(group-name) member-list
......
ELEMENT
_(group-name) element-list
......
SOLID
_(group-name) solid element-list
......
FLOOR
_(group-name) member-list

END GROUP DEFINITION

where,

group-name = an alphanumeric name specified by the user to identify the group. The group-name must start with the '_' (underscore) character and is limited to 24 characters.

member-list = the list of members/elements/solids belonging to the group. TO, BY, ALL, BEAM, PLATE, and SOLID are permitted. ALL means all members+ plates+ solids; BEAM means all beams; PLATE all plates; and SOLID all solids.

joint-list = the list of joints belonging to the group. TO, BY, and ALL are permitted.

Notes

1) The GROUP definition must start with the START GROUP DEFINITION command and end with the END command.
2) More than one GROUP name may be specified within the same definition specification.
3) The words JOINT, MEMBER, ELEMENT, FLOOR and SOLID may be provided if the user wishes to identify the group name and lists with those specific items. However, if the group name and list is merely a means of grouping together more than one type of structural component under a single heading, the word GEOMETRY may be provided. In the absence of any of those words (GEOMETRY, JOINT, MEMBER, ELEMENT, FLOOR or SOLID), the list is assumed to be that for GEOMETRY.
4) The same joint or member/element number may be included in up to 4 groups. Multiple definitions are useful for output but can be ambiguous for input data such as constants, section property, release, etc.
5) If two or more consecutively entered groups have the same name, then they will be merged. If not consecutive, the second entry of the same name will be ignored.
6) A member group may be used in lieu of a member-list with virtually any command which requires member lists, such as MEMBER LOADS, steel and concrete parameters, etc. There

is one place however where a MEMBER GROUP will not suffice, and that is for defining panels during a FLOOR LOAD assignment.

In section 5.32.4 of this manual, as explained under the topic "Applying floor load on members grouped under a FLOOR GROUP name", a panel has to be specified using a FLOOR GROUP, not a MEMBER GROUP. A FLOOR GROUP is presently not accepted in lieu of a member-list for any other command.

Example

```
START GROUP DEFINITION
_TRUSS 1 TO 20 25 35
_BEAM 40 TO 50
END
MEMBER PROPERTIES
_TRUSS TA LD L40304
_BEAM TA ST W12X26
```

Example

```
START GROUP DEFINITION
JOINT
_TAGA 1 TO 10
MEMBER
_TAGB 40 TO 50
GEOMETRY
_TAGC 101 TO 135
END

MEMBER PROPERTIES
_TAGB TA LD L40304
_TAGC TA ST W12X26
```

5.16.2 Physical Members

Purpose

STAAD allows grouping analytical predefined members into physical members using a special member group PMEMBER. PMEMBER defines a group of analytical collinear members with same cross section and material property.

To model using PMEMBER, one needs to model regular analytical members and then, group those together.

While creating a PMEMBER, the following are the pre-requisites.

1. Existence of the analytical members in the member-list.
2. Selected members should be interconnected.
3. The selected individual members must be collinear.
4. Local axis of the individual members comprising the physical member should be identical (i.e. x, y and z are respectively parallel and in same sense).
5. A member in one Physical Member Group should not be part of any other Physical Member Group.

Description

PMEMBER can be created either in the modeling mode or in the Steel-Designer mode. Modeling mode and Steel-Designer mode PMEMBER-s will be labeled as M and D respectively. Modeling mode PMEMBER will allow variable cross-sections. Steel-Designer mode will allow importing of PMEMBER-s created in the modeling mode.

To define a Physical Member, the following command is used after the MEMBER INCIDENCE Command:

DEFINE PMEMBER
{Member list} PMEMBER (pmember-no)

Example:

```
JOINT COORDINATE
1 0 0 0 6 10.0 0 0
MEMBER INCIDENCE
1 1 2 5
DEFINE PMEMBER
1 TO 5 PMEMB 1
```

To define the member property of a Physical Member, the following command is used:

```
PMEMBER PROPERTY
{Pmember-list} PRIS ...
```

The Physical Member supports all types of member properties available in STAAD.

If multiple definitions of member properties for a particular analytical member is encountered (e.g. analytical member properties is defined twice, once via PMEMBER PROP command and again via the MEMBER PROP command, then the MEMBER PROP command will override the PMEMBER PROP definition.

To define the Material constants of a Physical Member, the following command is used:

```
PMEMBER CONSTANT
E  CONCRETE pmember-list
DEN CONCRETE pmember-list
……
……
……
```

Any member, which is a part of any PMEMBER is not allowed to be assigned constants explicitly.

At present, a Physical Member can be loaded with Uniformly Distributed Load and Moment, Concentrated Load and Moment, and Trapezoidal Load. The command syntax is as follows:

```
PMEMBER LOAD
{Pmember List} UNI / CON / UMOM / UCON / TRAP f1 f2 f3 f4
```

After the analysis, the Post Analysis results of a PMEMBER can be seen by using the following command:

```
PRINT PMEMBER FORCE
```

This command will produce member forces for all the analytical members in the group.

5.17 Rotation of Structure Geometry

Purpose

This command may be used to rotate the currently defined joint coordinates (and the attached members/elements) about the global axes.

General format:

$$\underline{\text{PER}}\text{FORM } \underline{\text{ROT}}\text{ATION } {}^{*}\left\{ \begin{array}{ll} \underline{X} & d_1 \\ \underline{Y} & d_2 \\ \underline{Z} & d_3 \end{array} \right\}$$

where, d_1, d_2, d_3 are the rotations (in degrees) about the X, Y and Z global axes respectively. This command may be entered after the Joint Coordinates or between two Joint Coordinate commands or after all Member/Element Incidences are specified. This command can be used to rotate the structure geometry (defined prior to this command) by any desired angle about any global axis. The rotated configuration is used for analysis and design. While specifying this command, the sense of the rotation should conform to the right hand rule.

Description

This command can be used to rotate the geometric shape through any desired angle about any global axis. The rotated configuration can be used for analysis and design.

Example

```
PERFORM ROTATION  X  20  Z  -15
```

5.18 Inactive/Delete Specification

Purpose

This set of commands may be used to temporarily INACTIVATE or permanently DELETE specified JOINTs or MEMBERs.

General format:

```
            { MEMBERS   member-list  }
INACTIVE    {                        }
            { ELEMENTS  element-list }

            { MEMBERS   member-list  }
DELETE      {                        }
            { JOINTS    joint-list   }
```

Description

These commands can be used to specify that certain joints or members be deactivated or completely deleted from a structure. The INACTIVE command makes the members and elements temporarily inactive; the user must re-activate them during the later part of the input for further processing. The DELETE command will completely delete the members/elements from the structure; the user cannot re-activate them. The Delete Joint command must be immediately after the Joint Coordinates. The DELETE member commands must be provided immediately after all member/element incidences are provided and before any INACTIVE commands.

Notes

a) The DELETE MEMBER command will automatically delete all joints associated with deleted members, provided the joints are not connected by any other active members or elements.
b) The DELETE MEMBER command will also delete all the joints, which were not connected to the structure in the first

place. For example, such joints may have been generated for ease of input of joint coordinates and were intended to be deleted. Hence, if a DELETE MEMBER command is used, a DELETE JOINT command should not be used.

c) The DELETE MEMBER command is applicable for deletion of members as well as elements. If the list of members to be deleted extends beyond one line, it should be continued on to the next line by providing a blank space followed by a hyphen (**-**) at the end of the current line.

d) The INACTIVE MEMBER command cannot be used in situations where inactivating a member results in joints becoming unconnected in space.

e) The INACTIVE MEMBER command should not be used if the MEMBER TENSION/COMPRESSION command is used.

f) The INACTIVated members may be restored for further processes (such as an analysis or design for a 2nd set of load cases) by using the CHANGE command. See Section 5.37 and Example 4 for more information.

g) The DELETE MEMBER command should be used to delete elements too. Specify the command as DELETE MEMBER j where j is the element number of the element you wish to delete. In the example shown below, 29 to 34 and 43 are element numbers.

h) Loads that have been defined on members declared as INACTIVE members will not be considered in the analysis. This applies to SELFWEIGHT, MEMBER LOADS, PRESTRESS and POSTSTRESS LOADS, TEMPERATURE LOADs, etc.

i) The DELETE JOINT command must be specified before all incidence commands such as MEMBER INCIDENCE, ELEMENT INCIDENCE, etc.

Example

```
INACTIVE   MEMBERS    5 7 TO 10
DELETE     MEMBERS    29 TO 34 43
```

5.19 User Steel Table Specification

Purpose

STAAD allows the user to create and use customized Steel Section Table (s) for Property specification, Code checking and Member Selection. This set of commands may be used to create the table(s) and provide necessary data.

General format:

<u>STA</u>RT <u>USE</u>R <u>TAB</u>LE
<u>TAB</u>LE i_1 (f_n)
section-type
section-name
property-spec
<u>END</u>

where,

i_1	=	table number (1 to 99). During the analysis process, the data in each user provided table is stored in a corresponding file with an extension .U0?. For example, the data of the 5th table is stored in .U05. The first part of the input file name is the same as that of the STAAD input file. These files are located in the same working directory as the input file. Hence, they may later be used as external user provided tables for other input files.
f_n	=	external file name containing the section type, name and corresponding properties (up to 72 characters)
section-type	=	a steel section name including: WIDE FLANGE, CHANNEL, ANGLE, DOUBLE ANGLE, TEE, PIPE, TUBE, GENERAL, ISECTION & PRISMATIC.

section-name = Any user designated section name, use 1 to 36 characters. First three characters of Pipes and Tubes must be PIP and TUB respectively. Only alphanumeric characters and digits are allowed for defining section names. (Blank spaces, asterisks, question marks, colon, semi-colon etc. are not permitted.)

Example

```
START USER TABLE
TABLE 1
UNIT INCHES KIP
WIDE FLANGE
P24X55-abcdefghijklmnopqrstuvwxyz111
16.2 23.57 0.375 7.005 0.505 1350 29.1 1.00688 8.83875 7.07505
P24X56
18.3 20.99 .4 8.24 .615 1330 57.5 1.83 0.84 7.0
MEMBER PROPERTY
27 UPTABLE 1 P24X55-abcdefghijklmnopqrstuvwxyz111
39 UPTABLE 1 P24X56
```

property-spec = Properties for the section. The requirements are different for each section type as follows. Shear areas AY and AZ must be provided to ensure proper shear stress or shear strength calculations during design. Also, SY, SZ, IZ and IY must be provided for design. The default length units for properties are the current units. If UNIT command is entered within the User Table in the input file then those units become the current units. However, a UNIT command on an external file only affects that file and has no effect on the units in subsequent input file commands. The user may specify the desired length unit by using the UNIT command as the first command in the table (see example following this description). If data is from input file, then use up to 3 lines of input per property-spec (end all but last with a hyphen, -). If data is from external file, then use only one line, but it can be up to 250 characters long.

Description

Following section types are available under this option.

Wide Flange

1) AX = Cross section area
2) D = Depth of the section
3) TW = Thickness of web
4) WF = Width of the flange
5) TF = Thickness of flange
6) IZ = Moment of inertia about local z-axis (usually strong axis)
7) IY = Moment of inertia about local y-axis
8) IX = Torsional constant
9) AY = Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
10) AZ = Same as above except in local z-axis.

Channel

1) AX, 2) D, 3) TW, 4) WF, 5) TF, 6) IZ, 7) IY, 8) IX, 9) CZ, 10) AY, 11) AZ

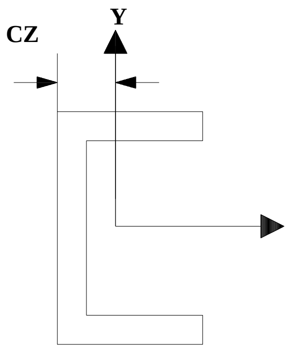


Figure 5.6

Angle

1) D, 2) WF, 3) TF, 4) R, 5) AY, 6) AZ

R = radius of gyration about principal axis, shown as r(Z-Z) in the AISC manual. Must not be zero.

Double Angle

1) D, 2) WF, 3) TF, 4) SP, 5) IZ, 6) IY, 7) IX, 8) CY, 9) AY, 10) AZ 11) RVV

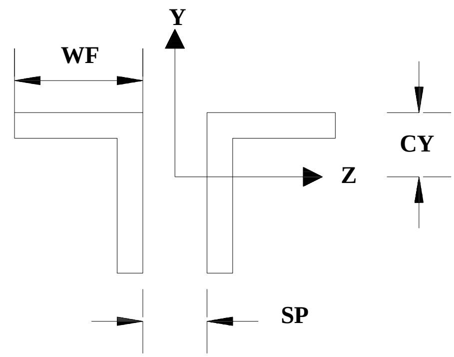


Figure 5.7

Note: The parameter RVV is defined as the radius of gyration about the minor principal axis for single angles - z-z axis for “TA ST” angles, and y-y axis for “TA RA” angles.

Tee

1) AX, 2) D, 3) WF, 4) TF, 5) TW, 6) IZ, 7) IY, 8) IX, 9) CY, 10) AY, 11) AZ

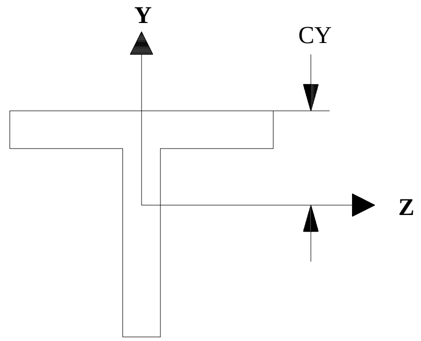


Figure 5.8

Pipe

1) OD = Outer diameter
2) ID = Inner diameter
3) AY, 4) AZ

Tube

1) AX, 2) D, 3) WF, 4) TF, 5) IZ, 6) IY, 7) IX, 8) AY, 9) AZ

General

The following cross-sectional properties should be used for this section-type. This facility allows the user to specify a built-up or unconventional steel section. Provide both the Y and Z parameters for design or code checking.

1)	AX	= Cross section area.
2)	D	= Depth of the section.
3)	TD	= Thickness associated with section element parallel to depth (usually web). To be used to check depth/thickness ratio.
4)	B	= Width of the section.
5)	TB	= Thickness associated with section element parallel to flange. To be used to check width/thickness ratio.

6) IZ = Moment of inertia about local z-axis.
7) IY = Moment of inertia about local y-axis.
8) IX = Torsional Constant.
9) SZ = Section modulus about local z-axis.
10) SY = Section modulus about local y-axis.
11) AY = Shear area for shear parallel to local y-axis.
12) AZ = Shear area for shear parallel to local z-axis.
13) PZ = Plastic modulus about local z-axis.
14) PY = Plastic modulus about local y-axis.
15) HSS = Warping constant for lateral torsional buckling calculations.
16) DEE = Depth of web. For rolled sections, distance between fillets should be provided.

Note:
Properties PZ, PY, HSS and DEE must be provided for code checking/member selection per plastic and limit state based codes (AISC LRFD, British, French, German and Scandinavian codes). For codes based on allowable stress design (AISC-ASD, AASHTO, Indian codes), zero values may be provided for these properties.

Isection

This section type may be used to specify a generalized I-shaped section. The cross-sectional properties required are listed below. This facility can be utilized to specify tapered I-shapes.

1) DWW = Depth of section at start node.
2) TWW = Thickness of web.
3) DWW1 = Depth of section at end node.
4) BFF = Width of top flange.
5) TFF = Thickness of top flange.
6) BFF1 = Width of bottom flange.
7) TFF1 = Thickness of bottom flange.
8) AYF = Shear area for shear parallel to Y-axis.
9) AZF = Shear area for shear parallel to Z-axis.
10) XIF = Torsional constant (IX or J).

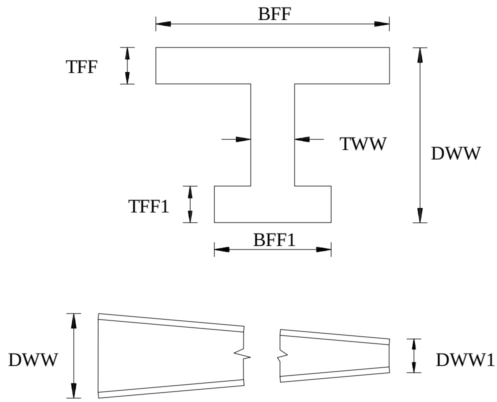


Figure 5.9

NOTES:

1) DWW should never be less than DWW1. The user should provide the member incidences accordingly.
2) The user is allowed the following options for the values AYF, AZF and XIF.

   a) If positive values are provided, they are used directly by the program.
   b) If zero is provided, the program calculates the properties using the following formula.

   AYF = D x TWW (where D =Depth at section under consideration)

   AZF = 0.66 ((BFF x TFF) + (BFF1 x TFF1))

   XIF = 1/3 ((BFF x TFF^3) + (DEE x TWW^3) + (BFF1 x TFF1^3))
   (where DEE = Depth of web of section)

   c) If negative values are provided, they are applied as factors on the corresponding value(s) calculated by the program using the above formula. The factor applied is always the absolute of the value provided, i.e. if the user provides the value of XIF as -1.3, then the program will multiply the value of XIF, calculated by the above formula, by a factor of 1.3.

Prismatic

The property-spec for the PRISMATIC section-type is as follows -

1) AX = Cross-section area
2) IZ = Moment of inertia about the local z-axis
3) IY = Moment of inertia about the local y-axis
4) IX = Torsional constant
5) AY = Shear area for shear parallel to local y-axis.
6) AZ = Shear area for shear parallel to local z-axis.
7) YD = Depth of the section in the direction of the local y-axis.
8) ZD = Depth of the section in the direction of the local z-axis.

Example

```
START USER TABLE
TABLE  1
UNIT . . .

WIDE FLANGE
W14X30
8.85  13.84  .27  6.73  .385  291.  19.6  .38  0  0
W21X50
14.7  20.83  .38  6.53  .535  984  24.9  1.14  7.92  0
W14X109
32.  14.32  .525  14.605  .86  1240  447  7.12  7.52  0
TABLE  2
UNIT . . .
ANGLES
L25255
2.5  2.5  0.3125 .489  0  0
L40404
4.  4.  .25  .795  0  0
END
```

* These section-names must be provided in ascending order by weight, since the member-selection process uses these tables and the iteration starts from the top. The above example can also be input as follows:

```
START USER TABLE
TABLE  1 TFILE1
TABLE  2 TFILE2
END
```

Where TFILE1 and TFILE2 are names of files which must be created prior to running STAAD, and where the file TFILE1 will contain the following:

```
UNIT  . . .
WIDE FLANGE
W14X30
8.85  13.84  .27  6.73  .385  291.  19.6  .38  0  0
W21X50
14.7  20.83  .38  6.53  .535  984  24.9  1.14  7.92  0
W14X109
32.  14.32  0.525  14.605  .86  1240  447  7.12  7.52  0
```

and the file TFILE2 will contain:

```
UNIT  . . .
ANGLES
L25255
2.5  2.5  .3125  .489  0  0
L40404
4.  4.  .25  .795  0  0
```

Notes

The User-Provided Steel Table(s) may be created and maintained as separate file(s). The same files may be used for all models using sections from these tables. These files should reside in the same directory where the input file is located. On each file the first table should contain a UNITS command.

5.20 Member Property Specification

Purpose

This set of commands may be used for specification of section properties for frame members.

The options for assigning properties come under 2 broad categories:

- Those which are specified from built-in property tables supplied with the program, such as for steel, aluminum and timber.
- Those which are **NOT** assigned from built-in tables, but instead are specified on a project-specific basis, such as for concrete beams and columns, or custom-made sections for industrial structures.

Properties which are NOT specified from built-in property tables

MEMBER PROPERTIES

```
member-list   { PRISMATIC property-spec   }
              { TAPERED argument-list     }
              { UPTABLE i1 section-name   }
```

For specification of PRISMATIC properties, see Section 5.20.2
For specification of TAPERED members, see Section 5.20.3
For specification from USER PROVIDED TABLES, see Section 5.20.4

Examples are available in Section 5.20.6.

The MEMBER PROPERTY command may be extended to multiple lines by ending all lines but the last with a space and hyphen (-).

Properties which are specified from built-in property tables

1. General format for standard steel (hot rolled):

```
                                    { AMERICAN      }
                                    { AUSTRALIAN    }
                                    { BRITISH       }
                                    { CANADIAN      }
                                    { CHINESE       }
                                    { DUTCH         }
                                    { EUROPEAN      }
MEMBER PROPERTIES                   { FRENCH        }
                                    { GERMAN        }
                                    { INDIAN        }
                                    { JAPANESE      }
                                    { KOREAN        }
                                    { MEXICAN       }
                                    { RUSSIAN       }
                                    { SAFRICAN      }
                                    { SPANISH       }
                                    { VENEZUELAN    }

member-list { TABLE type-spec section-name-in-table (additional-spec) }
            { ASSIGN profile-spec                                      }
```

AMERICAN, BRITISH, EUROPEAN (etc.) option will instruct the program to pick up properties from the appropriate steel table. The default depends on the country of distribution.

a. For **type-specs** and **additional-specs**, see Section 5.20.1

b. For **ASSIGN profile-spec**, see Section 5.20.5

Section 2 of this manual and the sections on steel design for various countries in the International Codes manual contain information on the section types which can be assigned for the various countries named in the above list.

Examples are available in Section 5.20.6.

The MEMBER PROPERTY command may be extended to multiple lines by ending all lines but the last with a space and hyphen (-).

2. General format for cold formed steel:

```
                              | BUTLER          |
                              | COLD AMERICAN   |
                              | COLD BRITISH    |
MEMBER PROPERTIES            {  COLD INDIAN     }
                              | KINGSPAN        |
                              | LYSAGHT         |
                              | RCECO           |

member-list   TABLE ST section-name-in-table
```

Section 2 of this manual and the sections on steel design for various countries in the International Codes manual contain information on the section types which can be assigned for the various countries/organizations named in the above list.

3. General format for steel joists:

```
MEMBER PROPERTIES  SJIJOIST

member-list   TABLE ST section-name-in-table
```

Section 1 of this manual contains information on the joist types which can be assigned from the Steel Joist Institute's tables.

4. General format for Aluminum:

MEMBER PROPERTIES ALUMINUM

member-list TABLE ST section-name-in-table

The section on aluminum design in the International Codes manual contain information on the section types which can be assigned for the aluminum table in the above list.

5. General format for Timber:

MEMBER PROPERTIES $\left\{ \begin{array}{l} \textbf{AITC} \\ \textbf{TIMBER CANADIAN} \end{array} \right\}$

member-list TABLE ST section-name-in-table

Section 4 of this manual and the sections on timber design in the International Codes manual contain information on the section types which can be assigned for the above list.

5.20.1 Type Specs and Additional Specs for assigning properties from Steel Tables

Purpose

The following commands are used for specifying section properties from built-in steel table(s).

General format:

type-spec . table-name additional-spec.

type-spec = { **ST** | **RA** | **D** | **LD** | **SD** | **T** | **CM** | **TC** | **BC** | **TB** | **FR** }

ST specifies single section from the standard built-in tables.
RA specifies single angle with reverse Y-Z axes (see Section 1.5.2).
D specifies double channel.
LD specifies long leg, back to back, double angle.
SD specifies short leg, back to back, double angle.
T specifies tee section cut from I shaped beams.
CM specifies composite section, available with I shaped beams.
TC specifies beams with top cover plate.
BC specifies beams with bottom cover plate.
TB specifies beams with top and bottom cover plates.
FR specifies Front to Front (toe to toe) channels. Spacing between the channels must be provided using the SP option

mentioned in the additional spec specification described below.

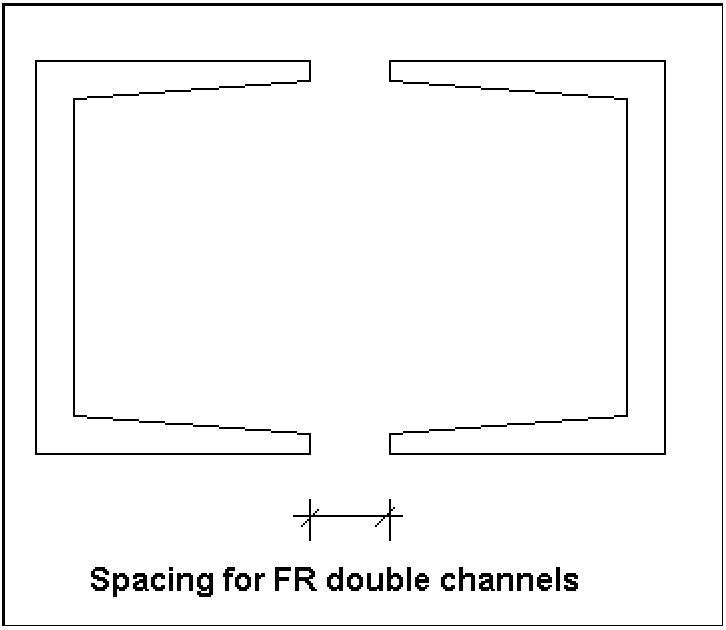


Figure 5.10a

table-name = Table section name like W8X18, C15X33 etc. The documentation on steel design per individual country codes contains information regarding their steel section specification also. For details on specifying sections from the American steel tables, see Section 2.2.1 of this manual.

$$\textbf{additional-spec} = * \left\{ \begin{array}{ll} \underline{\textbf{SP}} & f_1 \\ \underline{\textbf{WP}} & f_2 \\ \underline{\textbf{TH}} & f_3 \\ \underline{\textbf{WT}} & f_4 \\ \underline{\textbf{DT}} & f_5 \\ \underline{\textbf{OD}} & f_6 \\ \underline{\textbf{ID}} & f_7 \\ \underline{\textbf{CT}} & f_8 \\ \underline{\textbf{FC}} & f_9 \\ \underline{\textbf{CW}} & f_{10} \\ \underline{\textbf{CD}} & f_{11} \end{array} \right\}$$

See Section 1.7.2

SP f_1 = a. This set describes the spacing (f_1) between angles or channels if double angles or double channels are used. f1 defaults to 0.0 if not specified.
b. For composite sections, SP = rib height.

WP f_2 = a. Width (f_2) of the cover plate if a cover plate is used with I shaped sections.
b. For composite sections, WP = bottom coverplate width.

TH f_3 = a. Thickness (f_3) of plates or tubes.
b. For composite sections, TH = bottom coverplate thickness.

WT f_4 = Width (f_4) of tubes, where TUBE is the table-name.

DT f_5 = Depth (f_5) of tubes.

OD f_6 = Outside diameter (f_6) of pipes, where PIPE is the table-name.

ID f_7 = Inside diameter (f_7) of pipes.

CT f_8 = Concrete thickness (f_8) for composite sections.

FC f_9 = Compressive strength (f_9) of the concrete for composite sections.

CW f_{10} = Concrete width (f_{10}) for composite sections.

CD f_{11} = Concrete density (f_{11}) for composite sections. Default value is 150 pounds/cu.ft.

Example

See Section 5.20.6

Notes

All values $f_{1\text{-}9}$ must be supplied in current units.

Some important points to note in the case of the composite section are:

1. The 'CM' parameter can be assigned to I-shaped sections only. A 'CM' (composite) section is one obtained by considering a portion of a concrete slab to act in unison with the I shaped steel section. FC is the strength or grade of concrete used in the slab. In the USA, FC is called the specified compressive strength of concrete. Typical values of FC range between 2.0 and 5.0 ksi, and 20 to 50 Mpa.

2. The width of the concrete slab (CW) (if not entered) is assumed to be the width of the top flange of the steel section + 16 times the thickness of the slab.

3. In order to calculate the section properties of the cross-section, the modular ratio is calculated assuming that:

   Es= Modulus of elasticity of steel = 29000 Ksi.

   Ec= Modulus of elasticity of concrete = $1802.5\sqrt{FC}$ Ksi

   where FC (in Ksi) is defined earlier.

   Some other general notes on this subject of member property designations are :

4. The 'T' parameter stands for a T-shaped section obtained by cutting an I-shaped section at exactly its mid height level along the web. Hence, the area of a T shape is exactly half the area of the corresponding I shape. The depth of a T shape is half the depth of the I shape it was cut from.

   What we refer to as I shaped sections are sections which look like the English alphabet I. The American Wide Flange, the British UB and UC sections, Japanese H sections, etc., all fall under this category. Consequently, the 'T' shape cut from a Japanese H shape is one obtained by cutting the H shape at exactly its mid-height level of the web.

   Not all I shaped sections have a corresponding T. This may be inferred by going through the section libraries of individual countries and organizations. In such cases, if a user were to specify such a T section, the program will terminate with the message that the section does not exist.

5. Steel Cover plates also can be added only to I shaped sections. Thus, the 'TC', 'BC' and 'TB" are not applicable to any shape other than an I shape.

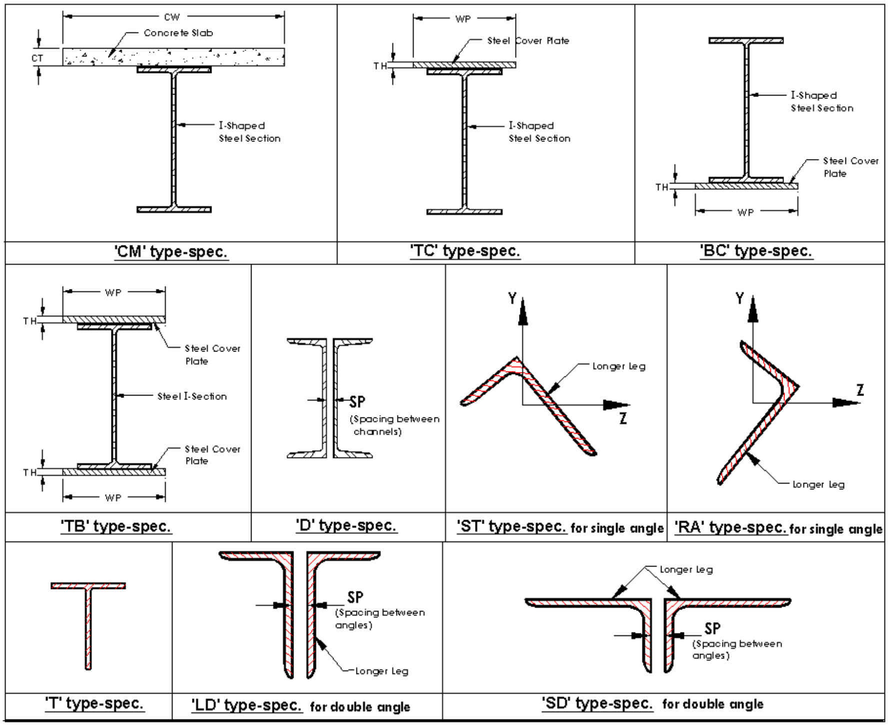

CW
Concrete Slab
CT
I-Shaped
Steel Section
'CM' type-spec.
WP
Steel Cover Plate
TH
I-Shaped
Steel Section
'TC' type-spec.
I-Shaped
Steel Section
Steel Cover
Plate
TH
WP
'BC' type-spec.
WP
TH
Steel Cover
Plate
Steel I-Section
Steel Cover
Plate
TH
WP
'TB' type-spec.
SP
(Spacing between
channels)
'D' type-spec.
Y
Longer Leg
Z
'ST' type-spec. for single angle
Y
Z
Longer Leg
'RA' type-spec. for single angle
'T' type-spec.
SP
(Spacing between
angles)
Longer Leg
'LD' type-spec. for double angle
Longer Leg
SP
(Spacing between
angles)
'SD' type-spec. for double angle

Figure 5.10b

5.20.2 Prismatic Property Specification

Purpose

The following commands are used to specify section properties for prismatic cross-sections.

General format:

For the PRISMATIC specification, properties are provided directly (End each line but last with a hyphen "-") as follows:

property-spec = * { <u>**AX**</u> f_1 | <u>**IX**</u> f_2 | <u>**IY**</u> f_3 | <u>**IZ**</u> f_4 | <u>**AY**</u> f_5 | <u>**AZ**</u> f_6 | <u>**YD**</u> f_7 | <u>**ZD**</u> f_8 | <u>**YB**</u> f_9 | <u>**ZB**</u> f_{10} }

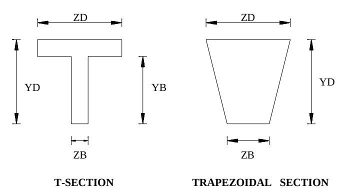


Figure 5.11

See Section 1.7.1

AX f_1 = Cross sectional area of the member. Set to zero for TEE, Rectangular, Trapezoid, or circular.
IX f_2 = Torsional constant.
IY f_3 = Moment of inertia about local y-axis.
IZ f_4 = Moment of inertia about local z-axis (usually major).
AY f_5 = Effective shear area in local y-axis.
AZ f_6 = Effective shear area in local z-axis.

If any of the above 6 parameters are omitted, it will be calculated from the YD, ZD, YB, and/or ZB dimensions.

YD f_7 = Depth of the member in local y direction.
(Diameter of section for circular members)

ZD f_8 = Depth of the member in local z direction. If ZD is not provided and YD is provided, the section will be assumed to be circular.

YB f_9 = Depth of stem for T-section.

ZB f_{10} = Width of stem for T-section or bottom width for TRAPEZOIDAL section.

The values that STAAD calculates for the omitted terms can be obtained by specifying the command

PRINT MEMBER PROPERTIES

The values of many of the derived properties like shear areas (AY, AZ), section modulii (SY, SZ), etc. will be shown in the output file.

This command can be used regardless of the manner in which the properties are specified - PRISMATIC, user table, built-in table.

5.20.2.1 Prismatic Tapered Tube Property Specification

Purpose

The following commands are used to specify section properties for prismatic tapered tube cross-sections. For the property types shown below, additional information can be obtained from Table 2.1 of the ASCE 72 document, 2nd edition.

General format:

$$\text{property-spec} = {}^{*}\left\{\begin{array}{l}\underline{\text{ROU}}\text{ND}\\ \underline{\text{H}}\text{EX}\underline{\text{D}}\text{E}\underline{\text{C}}\text{AGONAL}\\ \underline{\text{DOD}}\text{ECAGONAL}\\ \underline{\text{OCT}}\text{AGONAL}\\ \underline{\text{HEX}}\text{AGONAL}\\ \underline{\text{SQU}}\text{ARE}\end{array}\right\} \text{START } d_1 \text{ END } d_2 \text{ THICK } t$$

START d_1 = Depth of section at start of member.

END d_2 = Depth of section at end of member.

THICK t = Thickness of section (constant throughout the member length).

Example

```
UNIT . . .
MEMBER PROPERTY
1 PRIS ROUND STA 10 END  8  THI 0.375
2 PRIS HDC      STA 15 END 10 THI 0.375
3 PRIS DOD      STA 12 END 12 THI 0.375
```

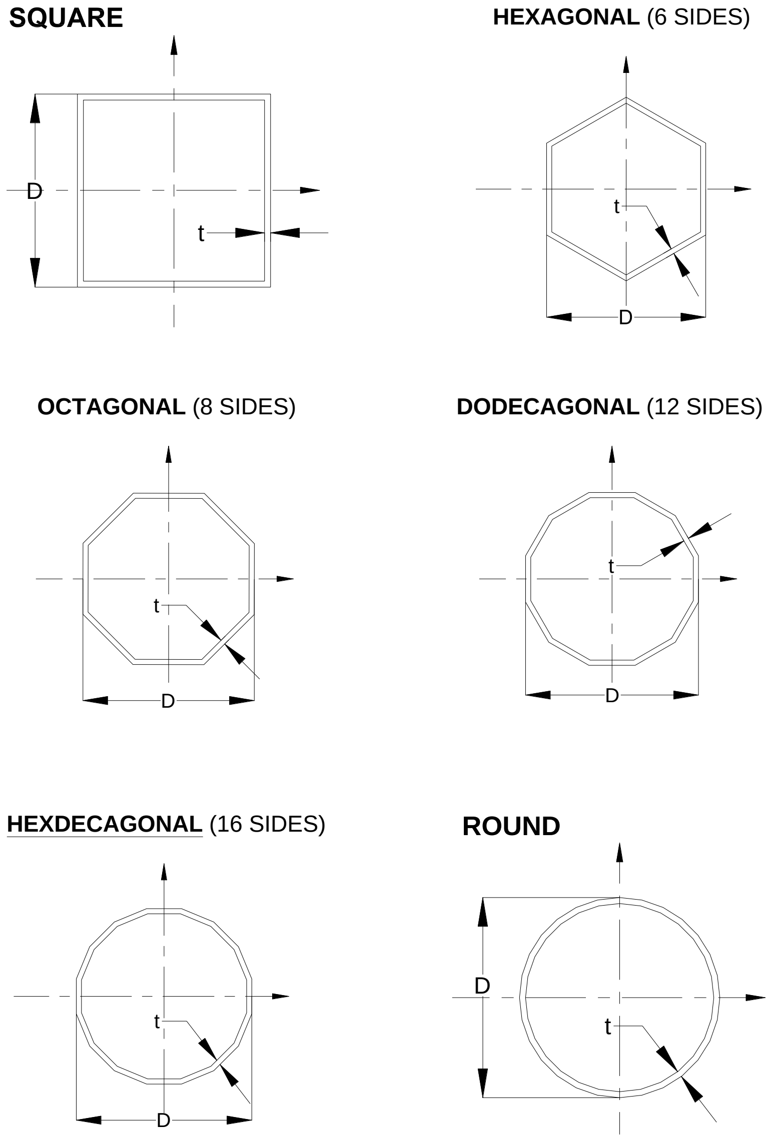


Figure 5.12 - Prismatic Tapered Tube Shapes

5.20.3 Tapered Member Specification

Purpose

The following commands are used to specify section properties for tapered I-shapes.

General format:

argument-list = f_1 f_2 f_3 f_4 f_5 (f_6 f_7)

where,
f_1 = Depth of section at start node.
f_2 = Thickness of web.
f_3 = Depth of section at end node.
f_4 = Width of top flange.
f_5 = Thickness of top flange.
f_6 = Width of bottom flange. Defaults to f_4 if left out.
f_7 = Thickness of bottom flange. Defaults to f_5 left out

See Section 1.7.4

Example

```
MEMBER PROPERTY
1 TO 5 TAPERED 13.98  0.285  13.98  6.745  .455  6.745  .455
```

Notes

1. All dimensions (f_1, f_2, ...f_7) should be in current units.
2. f_1 (Depth of section at start node) should always be greater than f_3 (Depth of section at end node). The user should provide the member incidences accordingly.

5.20.4 Property Specification from User Provided Table

Purpose

The following commands are used to specify section properties from a previously created USER-PROVIDED STEEL TABLE.

General format:

member-list UPTABLE I_1 section-name

UPTABLE stands for user-provided table

i_1 = table number as specified previously (1 to 99)

section-name = section name as specified in the table. (Refer to Section 5.19)

See Section 1.7.3

Example

See Section 5.20.6

5.20.5 Assign Profile Specification

Purpose

The ASSIGN command may be used to instruct the program to assign a suitable steel section to a frame member based on the profile-spec shown below.

General format:

See Section 1.7.5

$$\textbf{profile-spec} = \left\{ \begin{array}{l} \underline{\textbf{BEA}}\textbf{M} \\ \underline{\textbf{COL}}\textbf{UMN} \\ \\ \underline{\textbf{CHA}}\textbf{NNEL} \\ \underline{\textbf{ANG}}\textbf{LE (DOUBLE)} \end{array} \right\}$$

Example

See Section 5.20.6

Notes

Sections are always chosen from the relevant built-in steel table. To find out the details of the sections that are chosen, the command PRINT MEMBER PROPERTIES should be provided after specification of all member properties.

5.20.6 Examples of Member Property Specification

This section illustrates the various options available for MEMBER PROPERTY specification

Example

```
UNIT . . .
MEMBER PROPERTIES
1 TO 5 TABLE ST W8X31
9 10 TABLE LD L40304 SP 0.25
12 TO 15 PRISMATIC AX 10.0 IZ 1520.0
17 18 TA ST PIPE OD 2.5 ID 1.75
20 TO 25 TA ST TUBE DT 12. WT 8. TH 0.5
27 29 32 TO 40 42 PR AX 5. IZ 400. IY 33. -
IX 0.2 YD 9. ZD 3.
43 TO 47 UPT 1 W10X49
50 51 UPT 2 L40404
52 TO 55 ASSIGN COLUMN
56 TA TC W12X26 WP 4.0 TH 0.3
57 TA CM W14X34 CT 5.0 FC 3.0
```

This example shows each type of member property input. Members 1 to 5 are wide flanges selected from the AISC tables; 9 and 10 are double angles selected from the AISC tables; 12 to 15 are prismatic members with no shear deformation; 17 and 18 are pipe sections; 20 to 25 are tube sections; 27, 29, 32 to 40, and 42 are prismatic members with shear deformation; 43 to 47 are wide flanges selected from the user input table number 1; 50 and 51 are single angles from the user input table number 2; 52 through 55 are designated as COLUMN members using the ASSIGN specification. The program will assign a suitable I-section from the steel table for each member.

Member 56 is a wideflange W12X26 with a 4.0 unit wide cover plate of 0.3 units of thickness at the top. Member 57 is a composite section with a concrete slab of 5.0 units of thickness at the top of a wide flange W14X34. The compressive strength of the concrete in the slab is 3.0 force/length^2.

5.20.7 Composite Decks

As explained in section 1.7.7 of this manual, a composite deck generation facility is now built into the program. The command syntax for defining the deck within the STAAD input file is as shown below.

START DECK DEFINITION

_DECK deck-name
PERIPHERY member-list
DIRECTION d_1 d_2 d_3
COMPOSITE member-list
OUTER member-list

VENDOR name
FC f_1
CT f_2
CD f_3
RBH f_4
RBW f_5
PLT f_6
PLW f_7

DIA f_8
HGT f_9
DR1 f_{10}
SHR f_{11}
CMP f_{12}

CW f_{13} MEMB cw-member-list
END DECK DEFINITION

where,

deck-name = an alphanumeric name specified by the user to identify the deck. The deck-name line must start with '_DEC'. The deck-name is the second word and is limited to 23 characters. This name must not be the same as any group name.

member-list = the list of members belonging to the deck. TO, BY, ALL, and BEAM are permitted. ALL means all members in structure; BEAM means all beams.

d_1 = x component of the direction of the deck.

d_2 = y component of the direction of the deck.

d_3 = z component of the direction of the deck.

The following parameters may be in any order or omitted. They only apply to the composite members listed above. Do not enter a member list for these parameters.

f_1 = compressive strength of the concrete for all composite members listed above for this composite deck.

f_2 = concrete thickness.

f_3 = concrete density.

f_4 = height of rib of form steel deck. This is the distance from the top of the I beam to the bottom of the concrete deck.

f_5 = width of rib of form steel deck.

f_6 = thickness of cover plate welded to bottom flange of composite beam.

f_7 = width of cover plate welded to bottom flange of composite beam.

f_8 = diameter of shear connectors.

f_9 = height of shear connectors after welding.

f_{10} = ratio of moment due to dead load applied before concrete hardens to the total moment.

f_{11} = temporary shoring during construction.
0 = no shoring
1 = with shoring

f_{12} = composite action with connectors.
0 = no composite action in design
1 = composite action
2 = ignore positive moments during design

The following parameter may be specified by member list. They only apply to the composite members listed above.

f_{13} = concrete width for each composite member listed. cw-member-list = the list of composite members in this deck that have this width. Enter as many CW lines as necessary to define the width of all composite members of this deck.

This Deck definition data should be entered after the member properties have been entered.

Notes

1) The DECK definition must start with the START DECK DEFINITION command and end with the END command.
2) More than one DECK may be specified between the START and END.
3) The same member number may be included in up to 4 deck/groups. Multiple definitions are useful for output but can be ambiguous for input data such as constants, section property, release, etc.
4) If two or more consecutively entered decks have the same name, then they will be merged. If not consecutive, the second entry of the same name will be ignored.
5) The _deck-name must be unique within the Deck definitions and the Group definitions.
6) PER, DIR, OUT are data created by the GUI. Do not edit this data.
7) This Deck definition data should be entered after the member properties have been entered.

Example

```
START DECK DEFINITION
_DECK  DEC-1
PERIPHERY  4  16  40  18  38  56  50  49
DIRECTION  0.000000  0.000000  -1.000000
COMPOSITE  41  7  4  38
OUTER  7  8  31  30
VENDOR  USSTEEL
DIA  0.700
HGT  2.75
CT  11.0
FC  3.1
RBW  2.6
RBH  0.1
CMP  1.0
SHR  1
CD  0.0000870
CW  123.000000  MEMB  41
CW  123.000000  MEMB  7
CW  61.500000  MEMB  4
CW  61.500000  MEMB  38
END  DECK  DEFINITION
```

5.20.8 Curved Member Specification

Purpose

The following commands are used to specify that a member is curved. The curve must be a segment of a circle and the internal angle subtended by the arc must be less than 180 degrees. Any non-tapered cross-section is permitted.

General Format:

MEMBER CURVED

member-list RADIUS r GAMMA g PRESS p

where

r = radius in length units
g = The angle in degrees used to define the plane of the circle. The angle is defined using the same set of rules used to define the orientation (beta angle) of a straight member connected between the 2 nodes.
p = Pressure/Flexibility parameter for pipe bends. See notes.

The plane of the circle defines the plane formed by the straight line joining the 2 ends of the arc, and the local Y axis of an imaginary straight member between those 2 points. The positive value of the GAMMA angle is obtained using the same sense as the positive value of the beta angle of that imaginary straight line member whose local Y axis points towards the vertex of the arc.

Several diagrams intended to show the GAMMA angle for various segments lying in the 3 global planes are shown.

Pressure/Flexibility Parameter. This applies only to pipe bend (elbow) members (OD and ID entered). These members will flex more due to ovalization depending on internal pressure. The

ASME Boiler and Pressure Vessel Code, Section III, NB-3687.2, 1971, for Class I components is used to calculate the flexibility reduction factor.

Set p = 0 or omit for this flexibility increase calculation to occur with internal pressure equal to zero.

Set p > 0 to specify internal pressure to use in this flexibility calculation. Pressure reduces the flexibility increase.

Set p = -9999 to ignore this additional flexibility calculation and use only beam theory.

Set p = flexibility reduction factor (-FLEXF below); which must be a negative number less than –1.0 .

---

0) ASME PIPE ELBOW FLEXIBILITY FACTORS THEORY [ASME SECTION NB-3687.2]

This Section only applies if
(Bend Radius/Mean Radius) ≥ 1.70
or
if (Arclength) > (2 * Mean Radius)

FLEXF = (Flexibility Factor) =

$$\frac{1.65*(\text{Mean Radius})^2}{t*(\text{Bend Radius})}*\frac{1}{1+(\text{Press})(\text{Mean Radius})(\text{FACT.})}$$

where

$$\text{FACT.}=\frac{6*(MR\,/\,t)^{4/3}*(BR\,/\,MR)^{1/3}}{Et}$$

MR	=	Mean Radius of elbow wall
BR	=	Bend Radius
Press	=	Internal Pressure
t	=	elbow wall thickness
E	=	Modulus of Elasticity

If the Flexibility Factor computed is less than 1.0, then STAAD will use 1.0 . The Flexibility Factor directly multiplies or contributes to most non-shear terms in the elbow flexibility matrix.

Notes:

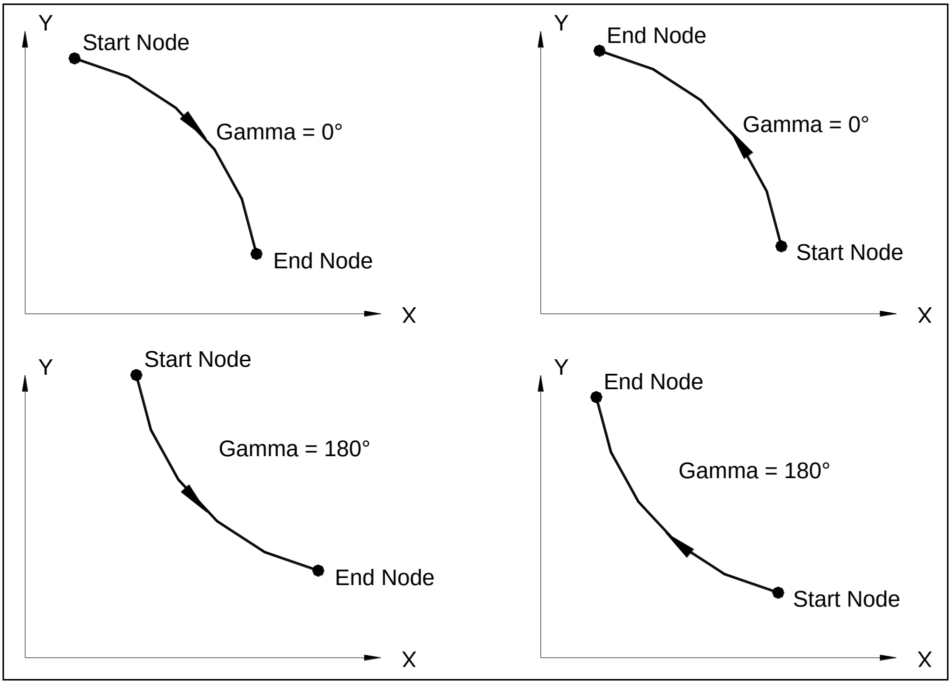
1) The input for defining the curved member involves 2 steps. The first is the member incidence, which is the same as that for a straight line member. The second is the command described above, which indicates that the segment between the 2 nodes of the member is curved, and not a straight line.

2) Any non-tapered cross section property currently available in STAAD can be assigned to these members.

3) Currently, two load types are permitted on curved members. One is the SELFWEIGHT load type, described in Section 5.32.9 of the STAAD.Pro Technical Reference manual. The other is the UNIformly distributed load type of the MEMBER LOAD options explained in Section 5.32.2 of the same manual. The uniformly distributed load has to be applied over the full span of the member. Other member loads such as LINEAR, TRAP, CONCENTRATED force or moment, UNIFORM moment, etc. are not currently supported. These options are expected to become available in future versions of the program.

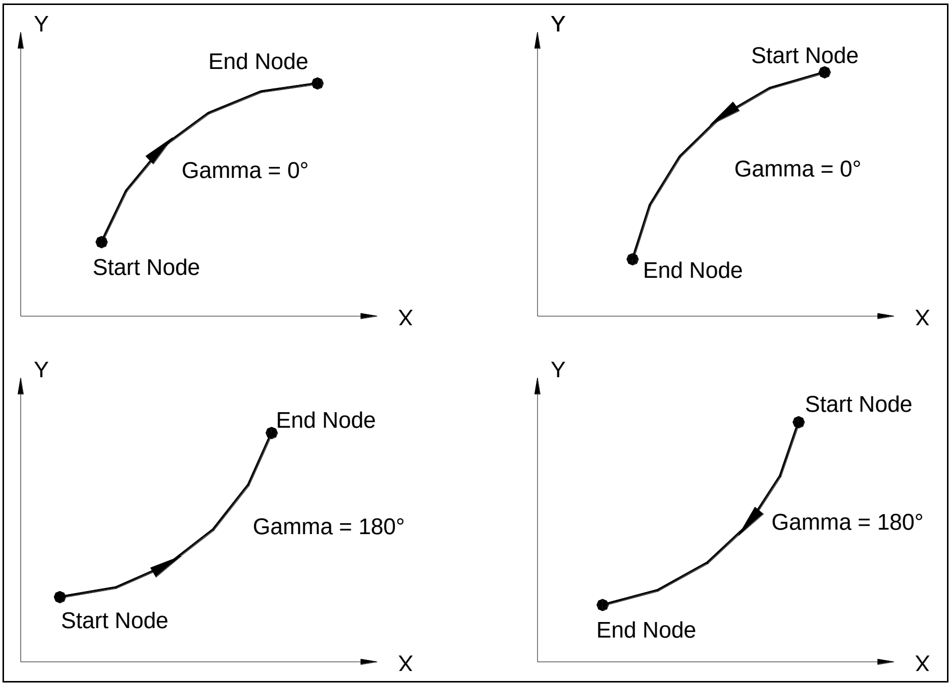
4) Some of the other member load types such as PRESTRESS, TEMPERATURE, STRAIN loads, etc. are also not currently supported. These options too are expected to become available in future versions of the program.

5) The results of the analysis currently consist of the nodal displacements of the ends of the curved member, and the member end forces. The nodal displacements are in the global coordinate system. The member end forces are in the local coordinate system, with each end of the member having its own unique local axis system. Results at intermediate sections, such as sectional displacements, and sectional forces will be available in future versions of the program.



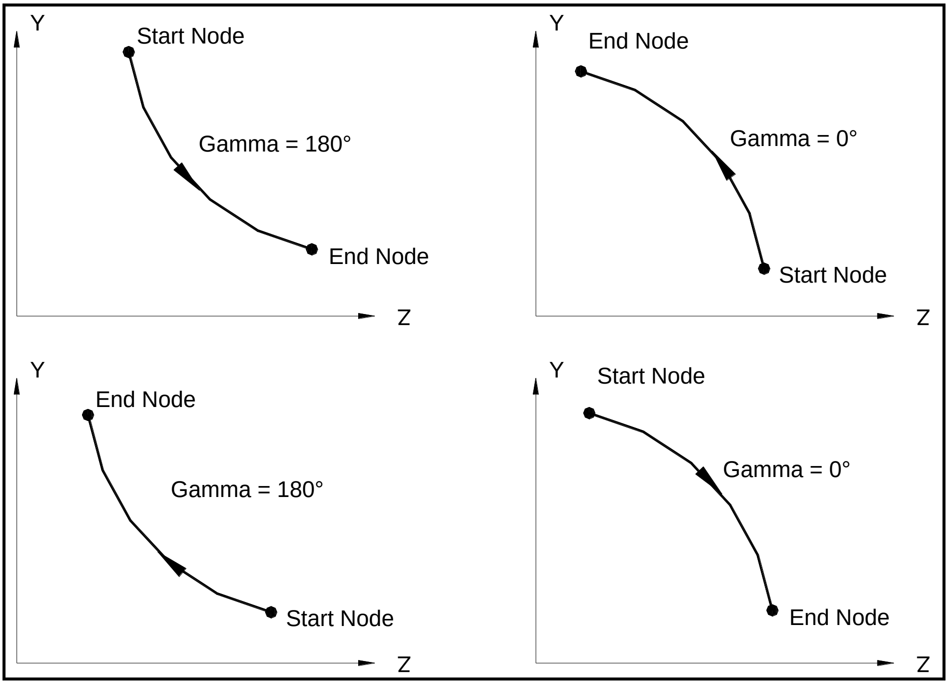
Gamma angle for various configurations of the circular arc lying in the global XY plane

Figure 5.13a



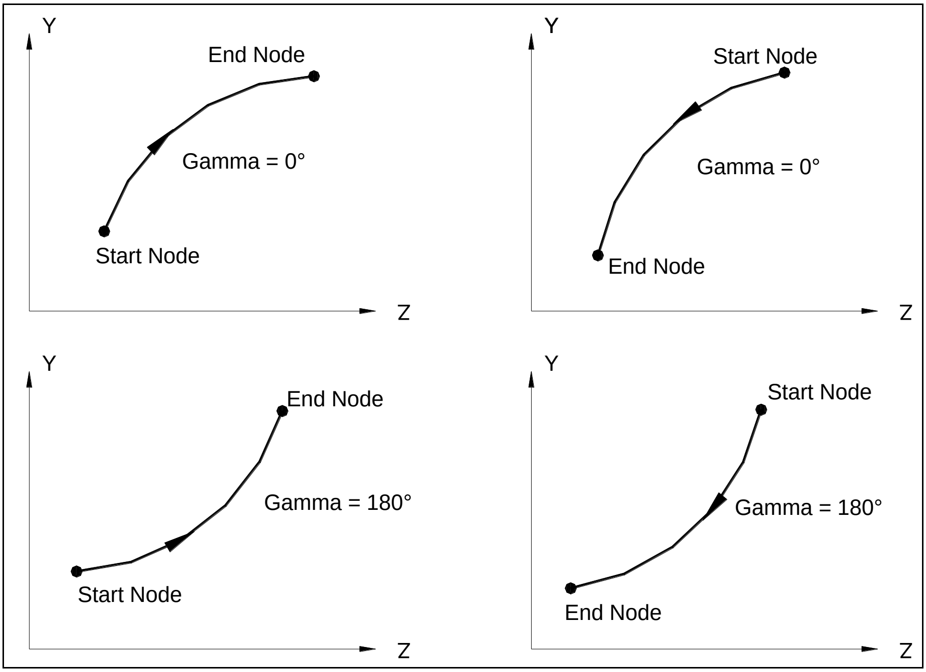
Gamma angle for various configurations of the circular arc lying in the global XY plane

Figure 5.13b



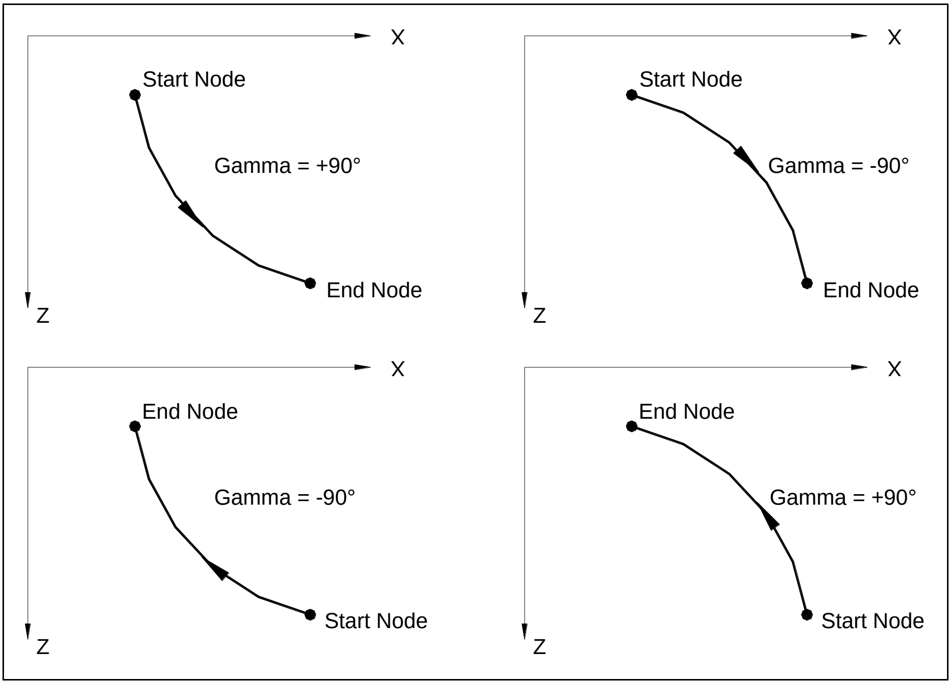
Gamma angle for various configurations of the circular arc lying in the global YZ plane

Figure 5.13c



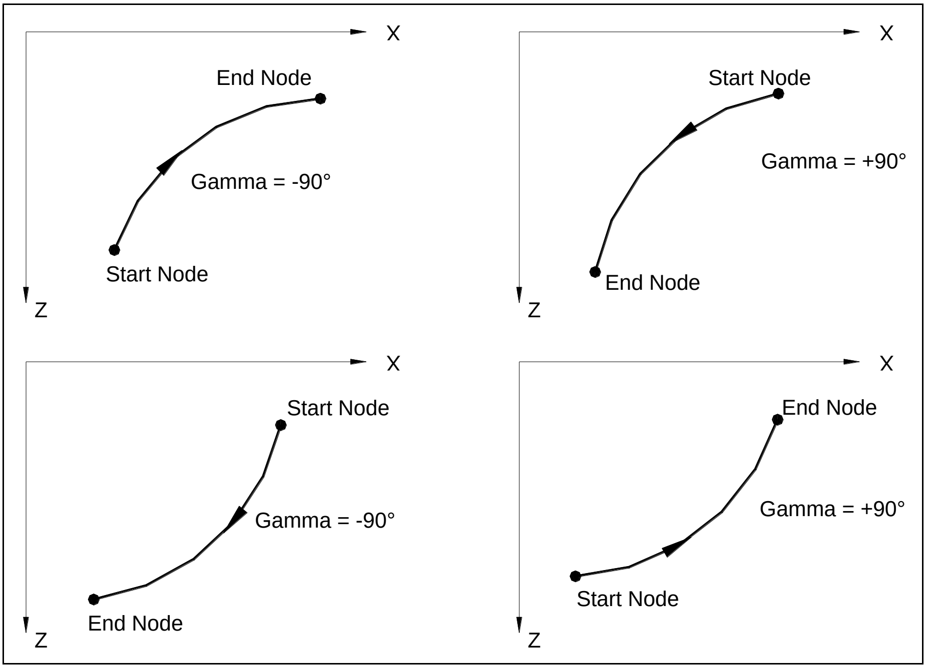
Gamma angle for various configurations of the circular arc lying in the global YZ plane

Figure 5.13d



Gamma angle for various configurations of the circular arc lying in the global XZ plane

Figure 5.13e



Gamma angle for various configurations of the circular arc lying in the global XZ plane

Figure 5.13f

Member local axis system

The local axis directions for curved members are dependent on the point of interest along the curve. The general rules for local axis, as laid out in Section 1.5.2 of this manual are applicable. The figure shown later for member end forces indicates the directions of axes at the start and end nodes.

Rotation of local axis

There is currently no facility available to change the orientation of a curved member. Hence, BETA ANGLE and REFERENCE POINT options, explained in section 1.5.3 and 5.26.2 of this manual, are not available for curved members.

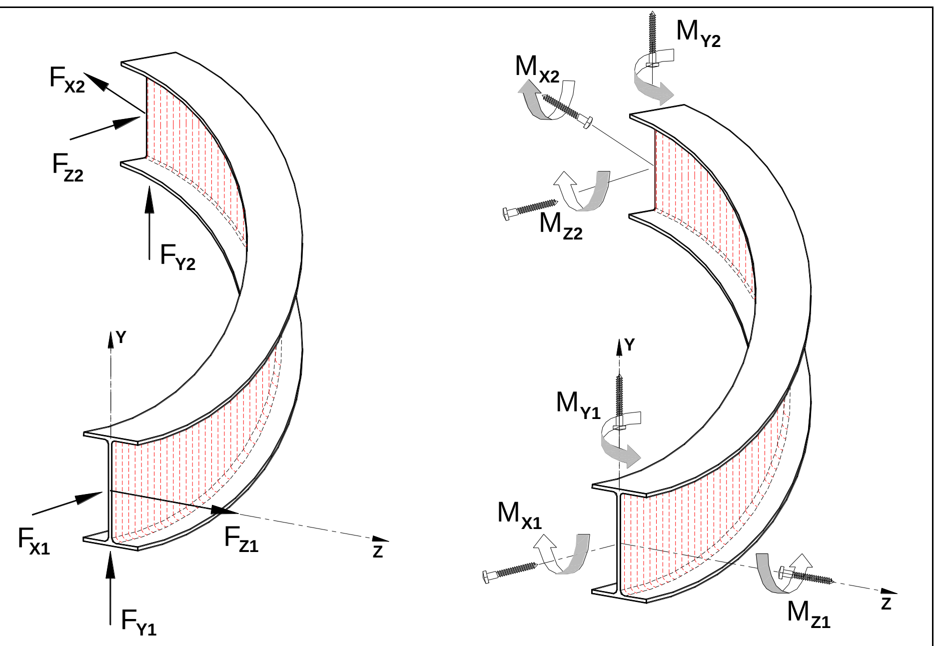
Sign convention for joint displacements

The displacements of the nodes of the curved member are along the global axis system just as in the case of straight members.

Sign convention for member end forces

The member end forces for curved members are quite similar to that for straight members. The only distinguishing item is that they are normal and tangential to the local axis at the corresponding ends. For example, FX at the start is tangential to the curve at the start node, and FX at the end is tangential to the curve at the end node. Similarly, FZ is along the radial direction at the two ends.

Member releases, offsets, tension/compression, truss and cable may not be specified for curved beams.



Global Y is Vertical

Figure 5.13g

Example

```
staad space
unit kip feet
joint coord cyl reverse
1 150 0 0 13 150 0 90
repeat 1 30 0 0
repeat all 1 0 15 0

memb inci
1 1 27 26
101 27 28 112
113 40 41 124
201 27 40 213

start group definition
member
_column 1 to 26
_circumferential 101 to 124
_radial 201 to 213
end group definition

member properties
_column pris yd 3.0
_circumferential pris yd 3.0
_radial pris yd 3.0

constant
E concrete all
density concrete all
poisson concrete all

MEMBER CURVED
101 TO 112 RADIUS 150 GAMMA 90.0
113 TO 124 RADIUS 180 GAMMA 90.0

supports
1 to 26 pinned

load 1
self y -1.0

perform analysis print stat check

PRINT MEMBER FORCE LIST 101 113
finish
```

Notes

1. The radius should be in current units.
2. Certain attributes like releases, TENSION/COMPRESSION flags, and several member load types are currently not available. Section forces too are currently not available.

5.20.9 Applying Fireproofing on members

STAAD.Pro now includes a method to automatically consider the weight of fireproofing material applied to structural steel.

Two types of fireproofing configurations are currently supported. They are:

Block Fireproofing (BFP):

The next figure shows this configuration. The fire-protection material forms a rectangular block around the steel section.

The area of fireproofing material (A_{fp}) at any section along the member length is calculated in the following manner.

For Wide Flanges (I-shaped sections), Channels and Tees,

$A_{fp} = [(B_f + 2T)*(D+2T)] - A_{steel}$

For single angles, $A_{fp} = [(B_f + 2T)*(D+2T)] - A_{steel}$

B_f is the flange width
D the overall depth of the steel section
T is the thickness of the fireproofing material beyond the outer edges of the cross section as shown in the next figure.
A_{steel} = Area of the steel section

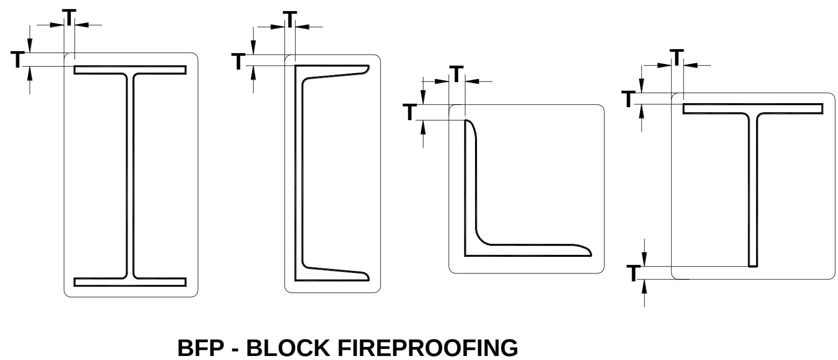


Figure 5.14

Contour Fireproofing (CFP):

In this configuration, the fire-protection material forms a coating around the steel section as shown in the next figure. The area of fireproofing material (A_{fp}) for this case is calculated in the following manner.

For Wide Flanges (I-shaped sections)

$$A_{fp} = [(B_f + 2T)*(T_f+2T)]*2 + [(D-2T-2T_f)*(T_w+2T)] - A_{steel}$$

For single angles,

$$A_{fp} = [(L_1+2T)*(2T+T_a) + (L_2-T_a)*(2T+T_a)] - A_{steel}$$

For Tees,

$$A_{fp} = [(B_f + 2T)*(T_f+2T)] + [(D-T_f)*(T_w+2T)] - A_{steel}$$

B_f is the flange width
D the overall depth of the steel section
T is the thickness of the fireproofing material beyond the outer edges of the cross section as shown in the next figure.
T_f is the thickness of the flange for the I shape and Tee
T_a is the thickness of the leg of the angle
T_w is the thickness of the web for the I shape and Tee
A_{steel} = Area of the steel section

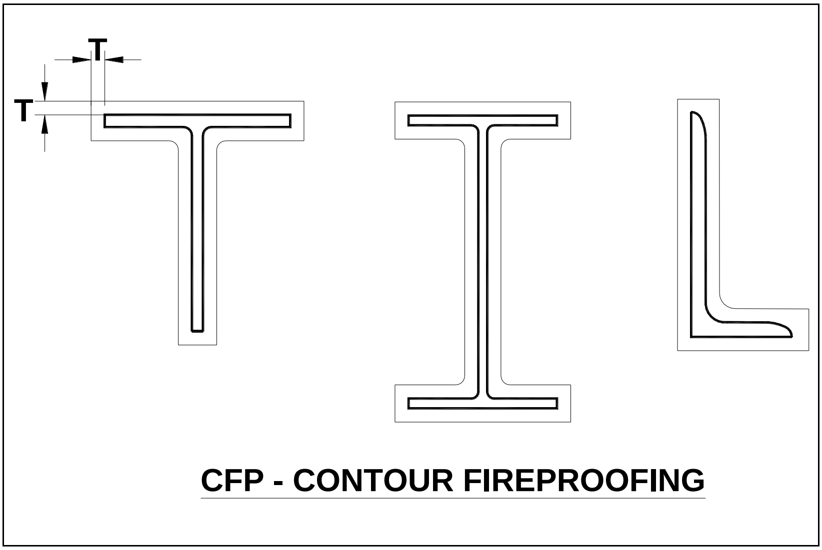


Figure 5.15

The number of input items required to apply this attribute is four – a) the type of fireproofing b) the thickness T shown in the above figures c) the density of the fireproofing material d) the members on which it is to be applied.

For each such member, A_{fp} is calculated and multiplied by the density of the fireproofing material to obtain the weight per unit length of the member. This is added to the weight per unit length of the steel section itself and the total is used in calculating selfweight. Hence, SELFWEIGHT must be one of the load components of load cases if the weight of the fireproofing material should be considered as part of those load cases.

Command Syntax

MEMBER FIREPROOFING

Member-list FIRE $\left\{ \begin{array}{c} \underline{\textbf{BFP}} \\ \underline{\textbf{CFP}} \end{array} \right\}$ THICKNESS f1 DENSITY f2

where,

f1 = thickness (T in the figures above) in length units
f2 = density of fireproofing material in (force / length ^ 3) units

In the actual load case itself, nothing besides the SELFWEIGHT command is necessary to instruct the program to include the weight of the fireproofing material in the selfweight calculation.

Notes

1. STAAD calculates the fire proofing weight only for the following sections:

   For block fireproofing - I-shaped sections like those from the built-in tables (American W,S,M,HP, British UC and UB, etc.), tapered I shaped sections, single channels, angles and Tees.

   For CFP-contour fireproofing - the sections are I-beam straight or tapered, angle, Tee.

   I-shaped sections like those from the built-in tables (American W,S,M,HP, British UC and UB, etc.), tapered I shaped sections, angles and Tees..

2. Fire proofing weight is not calculated for the following section types: Pipe, tube, composite I beams with slab on top, double channel, double angle, HSS, I-beam with cover plates, prismatic, solid circle or rectangle, castellated, cold formed sections, wood, aluminum, tapered poles, etc.

Example Problem

```
STAAD SPACE
UNIT KIP FEET
JOINT COORDINATES
1  0.  0. ; 2  0.  15. ; 3  20.  15. ; 4  20.  0.

MEMBER INCIDENCE
1  1  2 ; 2  2  3 ; 3  3  4

MEMBER PROPERTY AMERICAN
1  3  TABLE  ST  W12X26
2  TABLE  ST  W14X34

CONSTANTS
E  STEEL  ALL
POISSON STEEL ALL
DENSITY STEEL ALL
SUPPORT
1  FIXED ; 4  PINNED

UNIT POUND INCH
MEMBER FIREPROOFING
1 3 FIRE BFP THICK 2.0 DENSITY 40
2 FIRE CFP THICK 1.5 DENSITY 40
UNIT KIP FT
LOADING 1 DEADWEIGHT OF STEEL + FIREPROOFING
SELF Y -1.0

LOAD 2 LIVE
MEMBER LOAD
2  UNI GY -0.8

LOAD COMBINATION 3
1  0.75  2  0.75

PERFORM ANALYSIS
PRINT MEMBER FORCES
PRINT SUPPORT REACTIONS
FINISH
```

5.20.10 Member Property Reduction Factors

Purpose

Concrete design specifications recommend the use of cracked section properties for the analysis and design of concrete sections. Though the methodology to handle cracked section properties is non-linear in nature, i.e. the section capacities should be checked and modified depending upon the section forces the section is handling. The model should then be re-analyzed with modified reduced section properties and redesigned. This iteration should be continued until the forces in all sections designed are below the allowable limit of ultimate strength.

Similarly, the specifications in the AISC 13th edition manual suggest reducing the stiffness of the member during the analysis.

In STAAD.Pro, a user can now specify a set of reduction factors to be applied on the calculated section properties such as Area, Moments of Inertia and Torsional Constant. If a user wishes to adopt this approach to account for cracking of concrete sections, he/she may refer to ACI 318 (section 10.11.1) for a set of values to use for these reduction factors depending upon the nature of forces and moments the member is subjected to.

General format

The format of the command for the STAAD.Pro STD file is:-

MEMBER CRACKED

<Member List> REDUCTION {RAX/RIX/RIY/RIZ}factor

The reduction factor should be a fraction of unity.

Also, this is a multiplication factor on the property value. It does not signify the amount by which the property is reduced, but, it is simply a value by which the unreduced property is multiplied by. Thus, the calculated (or the user specified value) of the property will be multiplied by the reduction factor to arrive at the value used in the analysis.

For example, a factor of 0.45 defined for RAX will mean that if the cross sectional area of the gross section is 0.8 ft2, the value used in the analysis will be 0.8 X 0.45 = 0.36 ft2.

Multiple factors can be assigned on the same line.

Note - The reduction factor is considered only for analysis but not for design.

Example

```
MEMBER CRACKED
1 REDUCTION RAX 0.35 RIX 0.40 RIY 0.45 RIZ 0.45
```

5.21 Element / Surface Property Specification

General

Individual plate elements, and the Surface element need to have their thickness specified before the analysis can be performed. The commands for specifying this information are explained in this section. No similar properties are required for solid elements. However, constants such as modulus of elasticity, Poisson's Ratio, etc. are required.

5.21.1 Element Property Specification

Purpose

This set of commands may be used to specify properties of plate finite elements.

General Format:

ELEMENT PROPERTY

element-list THICKNESS f_1 (f_2, f_3, f_4)

f_1 = Thickness of the element.
$f_2 \ldots f_4$ = Thicknesses at other nodes of the element, if different from f_1.

Description

See Section 1.6

Elements of uniform or linearly varying thickness may be modeled using this command. The value of the thickness must be provided in current units.

Example

```
UNIT . . .
ELEMENT PROPERTY
1 TO 8 14 16 TH 0.25
```

5.21.2 Surface Property Specification

Purpose

This set of commands may be used to specify properties of surface entities.

General Format:

SURFACE PROPERTY

surface-list THICKNESS t

t = Thickness of the surface element, in current units.

Example

```
SURFACE PROPERTY
1 TO 3 THI 18
```

The attributes associated with surfaces, and the sections of this manual where the information may be obtained, are listed below:

Attributes	**Related Sections**
Surfaces incidences -	5.13.3
Openings in surfaces -	5.13.3
Local coordinate system for surfaces -	1.6.3
Specifying sections for stress/force output -	5.13.3
Property for surfaces -	5.21.2
Material constants -	5.26.3
Surface loading -	5.32.3.4
Stress/Force output printing -	5.42
Shear Wall Design -	3.8.2, 5.54

5.22 Member/Element Releases

STAAD allows specification of releases of degrees of freedom for frame members and plate elements. Section 5.22.1 describes MEMBER release options and Section 5.22.2 describes ELEMENT release options.

5.22.1 Member Release Specification

Purpose

This set of commands may be used to fully release specific degrees of freedom at the ends of frame members. They may also be used to describe a mode of attachment where the member end is connected to the joint for specific degrees of freedom through the means of springs.

General format:

MEMBER RELEASES

$$\textbf{member-list}\left\{\begin{array}{l}\textbf{START}\\ \textbf{END}\\ \textbf{BOTH}\end{array}\right\} \; {}^{*}\left\{\begin{array}{l}\textbf{FX}\\ \textbf{FY}\\ \textbf{FZ}\\ \textbf{MX}\\ \textbf{MY}\\ \textbf{MZ}\end{array}\right\} \; {}^{*}\left\{\begin{array}{ll}\textbf{KFX} & f1\\ \textbf{KFY} & f2\\ \textbf{KFZ} & f3\\ \textbf{KMX} & f4\\ \textbf{KMY} & f5\\ \textbf{KMZ} & f6\end{array}\right\}$$

where FX through MZ and KFX through KMZ represent force-x through moment-z degrees of freedom in the member local axes and f1 through f6 are spring constants for these degrees of freedom. If FX through MZ is used, it signifies a full release for that d.o.f, and if KFX through KMZ is used, it signifies a spring attachment

Example

```
MEMBER RELEASE
1 3 TO 9 11 12 START KFX 1000.0 MY MZ
1 10 11 13 TO 18 END MZ KMX 200.0
```

In the above example, for members 1, 3 to 9, 11 and 12, the moments about the local Y and Z axes are released at their start joints (as specified in MEMBER INCIDENCES). Further, these members are attached to their START joint along their local x axis through a spring whose stiffness is 1000.0 units of force/length. For members 1, 10, 11 and 13 to 18, the moment about the local Z axis is released at their end joint. Also, the members are attached to their END joint about their local x axis through a moment-spring whose stiffness is 200.0 units of force-length/Degree. Members 1 and 11 are released at both start and end joints, though not necessarily in the same degrees of freedom.

Partial Moment Release

Moments at the end of a member may be released partially. This facility may be used to model partial fixity of connections. The following format may be used to provide a partial moment release. This facility is provided under the MEMBER RELEASE option and is in addition to the existing RELEASE capabilities.

General Format:

MEMBER RELEASE

$$\textbf{member-list} \begin{Bmatrix} \textbf{START} \\ \textbf{END} \\ \textbf{BOTH} \end{Bmatrix} \quad * \begin{Bmatrix} \textbf{MP} \ \ f1 \\ \\ \textbf{MPX} \ \ f4 \\ \textbf{MPY} \ \ f5 \\ \textbf{MPZ} \ \ f6 \end{Bmatrix}$$

See Section 1.8

where f_1 = release factor for all 3 moments; or enter f_4 , f_5 , and/or f_6 as release factors for each moment separately. The moment related stiffness co-efficient will be multiplied by a factor of $(1-f_1)$ at the specified end. Release factors must be in the range of 0.001 through 0.999.

Example

```
MEMBER RELEASE
15  TO  25  START  MP  0.25
```

The above RELEASE command will apply a factor of 0.75 on the moment related stiffness coefficients at START of members 15 to 25.

Notes

Member releases are a means for describing a type of end condition for members when the default condition, namely, fully moment and force resistant, is not applicable. Examples are bolted or riveted connections. Partial moment releases are a way of specifying bending and torsional moment capacity of connections as being a fraction of the full bending and torsional strength.

It is important to note that the factor f_1 indicates a reduction in the stiffness corresponding to the rotational degrees of freedom MX, MY and MZ. In other words, the user should not expect the moment on the member to reduce by a factor of f_1 . It may be necessary for the user to perform a few trials in order to arrive at the right value of f_1, which results in the desired reduction in moment.

Also, START and END are based on the MEMBER INCIDENCE specification. The BOTH specification will apply the releases at both ends.

At any end of the member, for any particular DOF, full, partial and spring release cannot be applied simultaneously. Only one out of the three is permitted.

If MY (or MZ) is fully released at both ends, then VZ (or VY) cannot be transmitted through the member. The final shears in the member will be entirely due to loads applied directly to the member.

5.22.2 Element Release Specification

Purpose

This set of commands may be used to release specified degrees of freedoms at the corners of plate finite elements.

General Format:

ELEMENT RELEASE

element-list { **J1** / **J2** / **J3** / **J4** } * { **FX** / **FY** / **FZ** / **MX** / **MY** / **MZ** }

See Section 1.8

where the keywords J1, J2, J3 and J4 signify joints in the order of the specification of the element incidence. For example, if the incidences of the element were defined as 35 42 76 63, J1 represents 35, J2 represents 42, J3 represents 76, and J4 represents 63. Element releases at multiple joints cannot be specified in a single line. Those must be specified separately as shown below.

FX through MZ represents forces/moments to be released per local axis system.

Example

Correct Usage	Incorrect Usage
ELEMENT RELEASE	**ELEMENT RELEASE**
10 TO 50 J1 MX MY	**10 TO 50 J1 J2 MX MY**
10 TO 50 J2 MX MY	**10 TO 50 J3 J4 MY**
10 TO 50 J3 MY	
10 TO 50 J4 MY	

Notes

1. All releases are in the local axis system. See Figure 1.13 for the various degrees of freedom. Fx and Fy have the same sense as Sx and Sy in Figure 1.13. Fz has the same sense as SQx or SQy. Generally, do not over release. The element must still behave as a plate after the releases.
2. Selfweight is applied at each of the nodes as if there were no releases.
3. Thermal stresses will include the fixed-end thermal pre-stress as if there were no release.
4. May not be used with the Element Plane Stress or Element Ignore Inplane Rotation commands on the same element.
5. **Note that the usual definitions of local Mx and My are reversed here. See Figure 1.13 for the definitions of Mx and My. Releasing Fz, Mx, My will release all bending capability. Releasing Fx, Fy, Mz will release all in-plane stiffness.**

5.22.3 Element Ignore Stiffness

Purpose

Structural units like glass panels or corrugated sheet roofs are subjected to loads like wind pressures or snow loads. While these units are designed to carry those loads and transmit those loads to the rest of the structure, they are not designed to provide any additional stiffness to the structure. One way to handle the situation is to not input the unit as part of the structural model, and apply the load using load generation techniques like AREA LOAD or FLOOR LOAD.

STAAD provides another way of handling such units. This is through the help of the ELEMENT IGNORE STIFFNESS command. To use this feature, the glass panel or roof unit must be defined using plate elements. The IGNORE STIFFNESS command enables one to consider the unit just for the purpose of application of load, while its stiffness will not be considered during the assembly of the stiffness matrix. In other words, it is like an INACTIVE member which is active for load application but INACTIVE for stiffness.

General Format:

IGNORE STIFFNESS {ELEMENT} element-list

Example

```
IGNORE STIFFNESS ELEMENT 78 TO 80
```

5.23 Member Truss/Cable/Tension/Compression Specification

A member can have only one of the following specifications:

MEMBER TRUSS
MEMBER TENSION
MEMBER COMPRESSION
MEMBER RELEASES

If multiple specifications are applied to the same member, only the last entered will be used (Warnings will be printed).

MEMBER TRUSS, MEMBER TENSION, MEMBER COMPRESSION, and MEMBER CABLE are axial-only for stiffness.

MEMBER CABLE are special truss members that may also be specified as tension-only.

Sections 5.23.1 through 5.23.3 describe these specifications.

5.23.1 Member Truss Specification

Purpose

This command may be used to model a specified set of members as TRUSS members.

Description

This specification may be used to specify TRUSS type members in a PLANE, SPACE or FLOOR structure. The TRUSS members are capable of carrying only axial forces. Typically, bracing members in a PLANE or SPACE frame will be of this nature.

General format:

MEMBER TRUSS

member-list TENSION f_1

where f_1 = optional Initial Tension in truss member (in current units)

For NON-LINEAR CABLE ANALYSIS Only: The Tension parameter is ignored except in Non Linear Cable Analysis. For that analysis type, a truss with pretension is considered to be nonlinear (large displacement). In this analysis type, trusses without preload are assumed to be linear members that carry both tension and compression regardless of this command

See Sections 1.9 and 1.10

This command is superfluous when a TRUSS type structure has already been specified using the command STAAD TRUSS.

Example

```
MEMB TRUSS
1 TO 8 10 12 14 15
```

Notes

The TRUSS member has only one degree of freedom - the axial deformation. Note also that Member Releases are not allowed. Selfweight and transverse loads may induce shear/moment distributions in the member.

Member loads are lumped at each end, whereas a frame member with moment releases only retains the axial component of the applied member load.

5.23.2 Member Cable Specification

Purpose

This command may be used to model a specified set of members as CABLE members.

Description for use in all analyses except Non Linear Cable Analysis:

The CABLE members, in addition to elastic axial deformation, are also capable of accommodating the stiffness effect of initial tension and tension due to static loads. Theoretical discussions of CABLE members are presented in Section 1.11 of this manual.

General format:

See Sections 1.9, 1.11.

MEMBER CABLE

member-list TENSION f_1

where f_1 = Initial Tension in cable member (in current units)

Example

```
MEMB CABLE
20 TO 25 TENSION 15.5
```

Notes

The TENSION specified in the CABLE member is applied on the structure as an external load as well as is used to modify the stiffness of the member. See Section 1.11 for details. If TENSION or the value is omitted a minimum tension will be used.

This is a truss member but not a tension-only member unless you also include this member in a MEMBER TENSION input. See section 5.23.3. Note also that Member Releases are not allowed.

The tension is a preload and will not be the final tension in the cable after the deformation due to this preload.

Description for use in Non Linear Cable Analysis :

The CABLE members, in addition to elastic axial deformation, are also capable of accommodating large displacements.

General format:

MEMBER CABLE

member-list TENSION f_1
---OR---
member-list LENGTH f_2

See Sections 1.9, 1.11, 1.18.2.5, 5.37

where f_1= Initial Tension in cable member (in current units)
where f_2= Unstressed cable length (in current units)

Example

```
MEMB CABLE
20 TO 25 TENSION 15.5
```

Notes

The TENSION specified in the CABLE member is applied on the structure as an external load as well as used to modify the stiffness of the member. See Section 1.11 for details.

The tension is used to determine the unstressed length. That length will be shorter than the distance between the joints by the distance that the tension will stretch the cable.

5.23.3 Member Tension/Compression Specification

Purpose

This command may be used to designate certain members as Tension-only or Compression-only members.

General Format:

MEMBER TENSION
member - list

MEMBER COMPRESSION
member - list

MEMBER TENSION 0
(No list required)

Description

Tension-only members are truss/cable members that are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them.

LINEAR TENSION/COMPRESSION ANALYSIS
Compression-only members are truss members that are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them. Member Releases are not allowed on members with this attribute.

See Section 1.9

The procedure for analysis of Tension-only or Compression-only members requires iterations for every load case and therefore may be quite involved. The user may also consider using the INACTIVE specification (instead of Tension/Compression) if the solution time becomes unacceptably high.

If a CHANGE command is used (because of a change in the list of tension members, cable tension, supports, etc.), then the SET NL command must be used to convey to STAAD that multiple analyses and multiple structural conditions are involved.

For NON-LINEAR CABLE ANALYSIS : This command is unnecessary and ignored. Cables are automatically assumed to be partially to fully tension only (except that there should always be selfweight) without this command. In this analysis type, trusses without preload are assumed to be linear members that carry both tension and compression regardless of this command.

Example

```
MEMBER TENSION
25 TO 30 35 36
```

Example

```
MEMBER COMPRESSION
43 57 98 102 145
```

Example

```
MEMBER TENSION
12 17 19 TO 37 65
MEMBER COMPRESSION
5 13 46 TO 53 87
```

Notes for LINEAR TENSION/COMPRESSION ANALYSIS

1) Loads that have been defined on members declared as MEMBER TENSION or MEMBER COMPRESSION will be active even when the member becomes INACTIVE during the process of analysis. This applies to SELFWEIGHT, MEMBER LOADS, PRESTRESS & POSTSTRESS LOADS, TEMPERATURE LOAD, etc.

2) A member declared as a TENSION only member or a COMPRESSION only member will carry axial forces only. It will not carry moments or shear forces. In other words, it is a truss member.
3) The MEMBER TENSION and MEMBER COMPRESSION commands should not be specified if the INACTIVE MEMBER command is specified.
4) Do not use Load Combination to combine these cases. Tension/Compression cases are non-linear and should not be linearly combined as in Load Combination. Use a primary load case with the Repeat Load command.
5) The following is the general sequence of commands in the input file if the MEMBER TENSION or MEMBER COMPRESSION command is used. This example is for the MEMBER TENSION command. Similar rules are applicable for the MEMBER COMPRESSION command. The dots indicate other input data items.

```
STAAD ...
SET NL ...
UNITS ...
JOINT COORDINATES
...
MEMBER INCIDENCES
...
ELEMENT INCIDENCES
...
CONSTANTS
...
MEMBER PROPERTY
...
ELEMENT PROPERTY
...
SUPPORTS
...
MEMBER TENSION
...
LOAD 1
```

```
...
LOAD 2
...
LOAD 3
...
LOAD 4
...
LOAD 5
REPEAT LOAD
...
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ...
PRINT ...
PARAMETER
...
CHECK CODE ...
FINISH
```

MEMBER TENSION 0

This command switches off ALL tension/compression only specifications for load cases which are specified subsequent to this command, usually entered after a CHANGE command. There is no list associated with this command. Hence, for any further primary load cases, the tension/compression only attributed is disabled for ALL members.

Notes

a) See Section 5.5 for explanation of the SET NL command. The number that follows this command is an upper bound on the total number of primary load cases in the file.

b) STAAD performs up to 10 iterations automatically, stopping if converged. If not converged, a warning message will be in the output. Enter a SET ITERLIM i command (i>10) before the first load case to increase the default number of iterations. Since convergence may not be possible using this procedure, do not set the limit too high.

c) The principle used in the analysis is the following.

- The program reads the list of members declared as MEMBER TENSION and/or COMPRESSION.
- The analysis is performed for the entire structure and the member forces are computed.
- For the members declared as MEMBER TENSION / COMPRESSION, the program checks the axial force to determine whether it is tensile or compressive. If the member cannot take that load, the member is "switched off" from the structure.
- The analysis is performed again without the switched off members.
- Up to ITERLIM iterations of the above steps are made for each load case.
- This method does not always converge and may become unstable. Check the output for instability messages. Do not use results if the last iteration was unstable.

d) A revised MEMBER TENSION / COMPRESSION command and its accompanying list of members may be provided after a CHANGE command. If entered, the new MEMBER TENSION/COMPRESSION commands replace all prior such commands. If these commands are not entered after a CHANGE, then the previous commands will still be applicable.

e) The MEMBER TENSION command should not be used if the following load cases are present : Response Spectrum load case, Time History Load case, Moving Load case. If used, the MEMBER TENSION /COMPRESSION will be ignored in all load cases.

f) If UBC Load cases are included, then follow each UBC load case with an Analysis command, then a Change command.

5.24 Element Plane Stress and Inplane Rotation Specifications

Purpose

These commands allow the user to model the following conditions on plate elements

a) PLANE STRESS condition
b) In-plane rotation stiffness reformulated to be rigid or to be zero.

General Format:

ELEMENT { **PLANE STRESS** / **RIGID (INPLANE ROTATION)** / **IGNORE (INPLANE ROTATION)** } **element-list**

Description

See Section 1.6

The PLANE STRESS specification allows the user to model selected elements for PLANE STRESS action only [No bending or transverse shear stiffness].

The RIGID INPLANE ROTATION command causes the program to connect the corner Mz "in-plane rotation" action to the other corner Mz rotations rigidly. The STAAD plate element formulation normally produces a very soft Mz stiffness that improves the inplane shear deformation. However when the plate Mz is connected to a beam bending moment as the only load path for that moment, then the RIGID INPLANE option may be used to have that element carry the Mz moment to the other joints rigidly to avoid the instability at the beam end. Usually only the elements connected to beams in this manner would have this specification.

The IGNORE INPLANE ROTATION command causes the program to ignore "in-plane rotation" actions. The STAAD plate element formulation normally includes this important action automatically. However, it may be noted that some element formulations ignore this action by default. The user may utilize this option to compare STAAD results with solutions from these programs.

These options are exclusive of each other and also exclusive of element releases. No single element may have more than one of these options.

Example

```
ELEMENT PLANE STRESS
1 TO 10 15  20 25 35
ELEMENT IGNORE
30 50 TO 55
```

5.25 Member Offset Specification

Purpose

This command may be used to rigidly offset a frame member end from a joint to model the offset conditions existing at the ends of frame members.

General format:

MEMBER OFFSETS

member-list { **START** / **END** } **(LOCAL)** f_1, f_2, f_3

Description

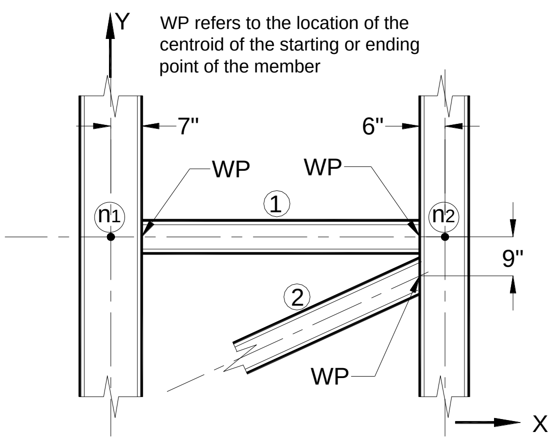


Figure 5.16

f_1, f_2, and f_3 correspond to the distance, measured in LOCALized or Global coordinate system, from the joint (START or END as specified) to the centroid of the starting or ending point of the members listed. MEMBER OFFSET command can be used for any member whose starting or ending point is not concurrent with the given incident joint. This command enables the user to account for the secondary forces, which are induced due to the eccentricity of the member. Member offsets can be specified in any direction, including the direction that may coincide with the member x-axis.

See Section 1.11

wp in the diagram refers to the location of the centroid of the starting or ending point of the member.

LOCAL is an optional parameter, if not entered then **f_1, f_2, f_3** are assumed to be in global. LOCAL means that the distances **f_1, f_2, f_3** are in the same member coordinate system that would result if the member were not offset and BETA = 0.0.

Example

```
MEMBER OFFSET
1  START  7.0
1  END  -6.0  0.0
2  END  -6.0  -9.0
```

Notes

1) If a MEMBER LOAD (see MEMBER LOAD specification) is applied on a member for which MEMBER OFFSETS have been specified, the location of the load is measured not from the coordinates of the starting joint. Instead, it is measured from the offset location of the starting joint.
2) START and END is based on the user's specification of MEMBER INCIDENCE for the particular member.

5.26 Specifying and Assigning Material Constants

Purpose

Material constants are attributes like Modulus of Elasticity and Density which are required for operations like generating the stiffness matrix, computing selfweight, and for steel and concrete design.

In STAAD, there are 2 ways in which this data may be specified :

a. a 2-step process that involves

step 1 - Creating the material data by defining MATERIAL tags specified under the heading DEFINE MATERIAL (see Section 5.26.1)

step 2 - Assigning them to individual members, plates and solids under the heading CONSTANTS (see Section 5.26.2)

This will create commands as shown below :

```
DEFINE MATERIAL
...  name
...                          <-------------- Part 1
...
END MATERIAL
CONSTANTS
MATERIAL name ...   <-------------- Part 2
```

b. Assign material attributes explicitly by specifying the individual constants as explained in section 5.26.2.

CONSTANTS
E ...
POISSON ..

Section 5.26.3 explains the commands required to assign material data to Surface elements.

5.26.1 The Define Material Command

Purpose

This command may be used to specify the material properties by material name. Then assign the members and elements to this material name in the CONSTANTS command.

General format:

DEFINE MATERIAL

ISOTROPIC ***name***
or
2DORTHOTROPIC ***name***

$$\left\{\begin{array}{l}\textbf{E}\\\textbf{G}\\\textbf{POISSON}\\\textbf{DENSITY}\\\textbf{ALPHA}\\\textbf{DAMPING}\\\textbf{CDAMP}\end{array}\right\}\begin{array}{ll}f_1 & f_2\\ f_1 & \\ f_1 & \\ f_1 & \\ f_1 & f_2\\ f_1 & \\ f_1 & \end{array}$$

Repeat ISOTROPIC or 2DORTHOTROPIC *name* and values for as many materials as desired then:

END MATERIAL (DEFINITION)

Name	material name (name of up to 36 characters).
E	specifies Young's Modulus.
G	specifies Shear Modulus. Enter only for beams & plates and when Poisson would not be 0.01 to 0.499.
POISSON	specifies Poisson's Ratio. If G is not entered, then this value is used for calculating the Shear Modulus (G=0.5xE/(1+POISSON)). Must be 0.01 to 0.499. Poisson must be entered for orthotropic

See next command for example

	plates or when Poisson cannot be computed from G.
DENSITY	specifies weight density.
ALPHA	Co-efficient of thermal expansion.
DAMPING or CDAMP	
	Damping ratio to be used in computing the modal damping by the composite damping method.
f_1 f_2	Value of the corresponding constants. For E, G, POISSON, DENSITY, ALPHA and damping. For plates only, the first value is for local x direction and the second for local y.

5.26.2 Specifying CONSTANTS for members, plate elements and solid elements

Purpose

This command may be used to specify the material properties (Moduli of Elasticity and Shear, Poisson's ratio, Density, Coefficient of linear expansion, and material damping) of the members and elements. In addition, this command may also be used to specify the member orientation (BETA angle or REFERENCE point).

General format:

CONSTANTS

MATERIAL ***name*** { **MEMBER memb/elem-list** | **(ALL)** }

where ***name*** is the material name as specified in the DEFINE MATERIAL command (see section 5.26.1).

-OR-

{ **E** | **G** | **POISSON** | **DENSITY** | **BETA** | **ALPHA** | **CDAMP** } f_1 { **MEMBER memb/elem-list** | **BEAM** | **PLATE** | **SOLID** | **(ALL)** }

{ **REF** f_2, f_3, f_4 | **REFJT** f_5 | **REFVECTOR f6 f7 f8** } **MEMBER memb/elem-list**

See Section 1.5.3

List specifier: MEM, BEA, PLA, SOL, ALL. Only MEM may be followed by a list. Blank means ALL. ALL means all members and elements; BEA means all members; PLA, all plates; SOL, all solids.

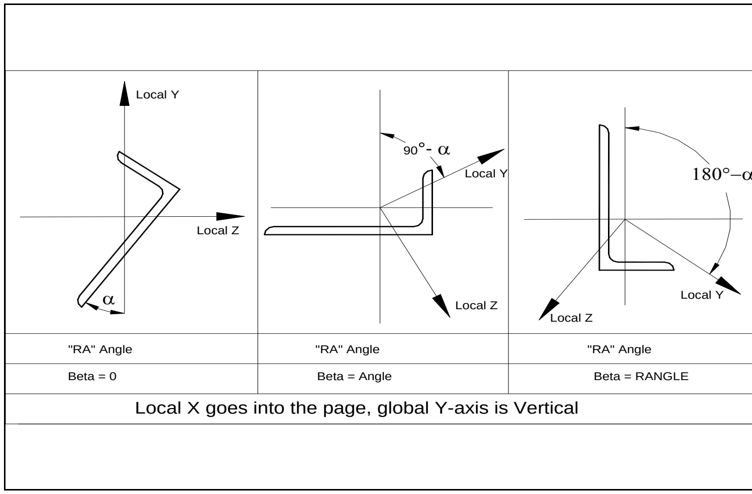
E	specifies Young's Modulus. This value must be provided before the POISSON for each member/element in the Constants list.
G	specifies Shear Modulus. Enter only for beams & plates and when Poisson would not be 0.01 to 0.499.
POISSON	specifies Poisson's Ratio. If G is not entered, then this value is used for calculating the Shear Modulus (G=0.5xE/(1+POISSON)). Must be 0.01 to 0.499. Poisson must be entered when Poisson cannot be computed from G.
DENSITY	specifies weight density.
ALPHA	Co-efficient of thermal expansion.
CDAMP	Damping ratio to be used in computing the modal damping by the composite damping method.
BETA	specifies member rotation angle in degrees (see Section 1.5.3).

Note : Single angle sections are oriented according to their principal axes by default. If it is necessary to orient them such that their legs are parallel to the global axes, the BETA specification must be used. STAAD offers the following additional specifications for this purpose:

BETA ANGLE

BETA RANGLE

Both of the above options will result in an orientation with the legs parallel to the global axis. The 'ANGLE' option rotates the section through the angle (90 - "alpha") (where "alpha" = angle between the principal axis system and the global axis system). [Please review the figures on the next page.] The 'RANGLE' option rotates the section through an angle equal to (180 - "alpha"). Both options will work the same way for equal angles. For unequal angles, the right option must be used based on the required orientation.


Local Y
Local Z
α
90°- α
Local Y
Local Z
180°–α
Local Y
Local Z
"RA" Angle
"RA" Angle
"RA" Angle
Beta = 0
Beta = Angle
Beta = RANGLE
Local X goes into the page, global Y-axis is Vertical

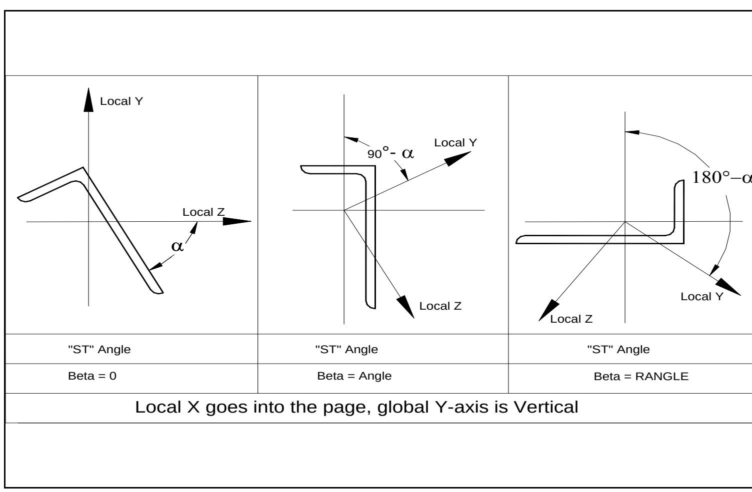
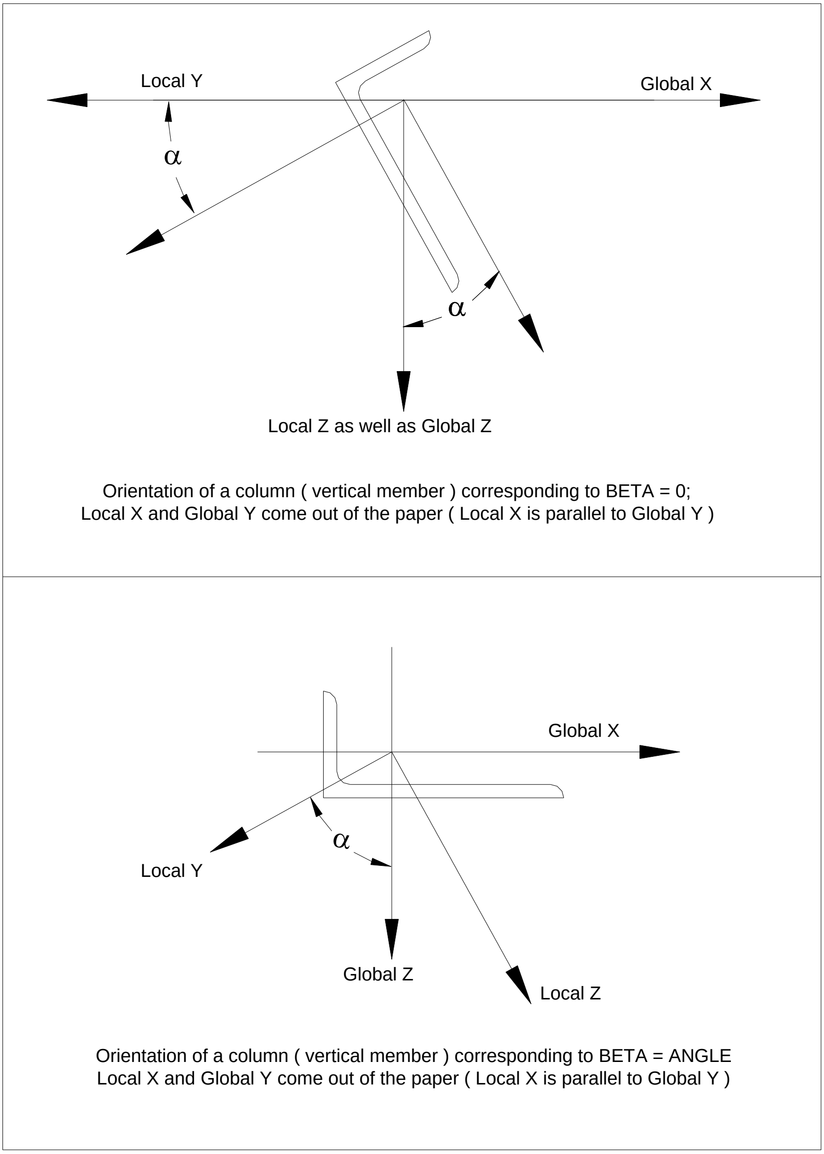

Local Y
Local Z
α
90°- α
Local Y
Local Z
180°–α
Local Y
Local Z
"ST" Angle
"ST" Angle
"ST" Angle
Beta = 0
Beta = Angle
Beta = RANGLE
Local X goes into the page, global Y-axis is Vertical

Figure 5.17a



Orientation of a column (vertical member) corresponding to BETA = 0;
Local X and Global Y come out of the paper (Local X is parallel to Global Y)

Orientation of a column (vertical member) corresponding to BETA = ANGLE
Local X and Global Y come out of the paper (Local X is parallel to Global Y)

Figure 5.17b

f_1 Value of the corresponding constants. For E, G, POISSON, DENSITY, ALPHA and CDAMP, built-in material names can be entered instead of f_1. The built-in names are STEEL, CONCRETE & ALUMINUM. Appropriate values will be automatically assigned for the built-in names.

CONSTANTS in Kip, inch, Fahrenheit units				
Constant	**Steel**	**Concrete**	**Aluminum**	**Units**
E (US)	29000	3150	10000	Kip/in^2
Poisson's	0.30	.17	.33	
Density	.000283	.0000868	.000098	Kip/in^3
Alpha	6.5E-6	5.5E-6	12.8E-6	L/L/deg-F
CDAMP	.03	.05	.03	Ratio
E (nonUS)	29732.736			Kip/in^2
CONSTANTS in MKS units				
Constant	**Steel**	**Concrete**	**Aluminum**	**Units**
E (US)	199 947 960	21 718 455	68 947 573	kN/m^2
Poisson's	0.30	.17	.33	
Density	76.819 541	23.561 612	26.601 820	kN/m^3
Alpha	12.0E-6	10.0E-6	23.0E-6	L/L/deg-C
CDAMP	.03	.05	.03	Ratio
E (nonUS)	205 000 000			kN/m^2

E (US) is used if US codes were installed or if Member Properties American is specified for an analysis; otherwise E (nonUS) is used.

f_2, f_3, f_4 Global X, Y, and Z coordinates for the reference point or

f_5 use location of joint f_5 for the reference point, from which the BETA angle will be calculated by STAAD (see section 1.5.3).

f_6, f_7, f_8 X, Y, Z distances by which one should move along the member's local X, Y, Z axis according to the BETA=0 condition, from the member's start node.

Example

```
DEFINE MATERIAL
ISOTROPIC CFSTEEL
E 28000.
POISSON 0.25
DENSITY 0.3E-3
ALPHA 11.7E-6
DAMP 0.075
END MATERIAL

CONSTANTS
MATERIAL CFSTEEL MEMB 1 TO 5

CONSTANTS
E  2.1E5  ALL
BETA  45.0  MEMB  5  7  TO  18
DENSITY STEEL MEMB  14  TO  29
BETA  90  MEMB X
REFVECTOR 0 2 1 MEMBER 27 TO 32
```

The second BETA command in the above example will set BETA as 90° for all members parallel to the X-axis.

The command "REFVECTOR 0 2 1 MEMBER 27 TO 32" in the above example instructs the program to do the following :

i. Establish the beam's local X,Y and Z axis corresponding to Beta=0

ii. Set the start node of the reference vector to be the same as the start node of the member.

iii. From the start node of the reference vector, move by a distance of 0 along the beam's local X axis, 2 along the local Y axis, and 1 along the local Z axis. This establishes the end node of the reference vector.

iv. At the end of step 3, the start node as well as the end node of the reference vector are known. That is the now the final direction of the member's local Y axis.

Since the local Y axis corresponding to Beta 0 is known, and the local-Y axis corresponding to the beam's final position has been established in step 4, Beta angle is calculated as the angle between these two vectors.

In this example, the angle is $Tan^{-1}(1/2) = 26.5651$ degrees

Notes

i. The value for E must always be given first in the Constants list for each member/element.
ii. All numerical values must be provided in the current units.
iii. It is not necessary or possible to specify the units of temperature or ALPHA. The user must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for temperature (see Section 5.32.6).
iv. If G is not specified, but Poisson is specified, G is calculated from $E / [2(1+\text{Poisson})]$.
v. If neither G nor Poisson is specified, Poisson is assumed based on E, and G is then calculated.
vi. If G and Poisson are both specified, the input value of G is used, G is not calculated in this situation.
vii. If G and Poisson are both required in the analysis, such as for the stiffness matrix of plate elements, and G is specified, but Poisson is not, then, Poisson is calculated from [(E/2G) –1].
viii. To obtain a report of the values of these terms used in the analysis, specify the command PRINT MATERIAL PROPERTIES.

5.26.3 Surface Constants Specification

Explained below is the command syntax for specifying constants for surface entities.

The attributes associated with surfaces, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces incidences -	5.13.3
Openings in surfaces -	5.13.3
Local coordinate system for surfaces -	1.6.3
Specifying sections for stress/force output -	5.13.3
Property for surfaces -	5.21.2
Material constants -	5.26.3
Surface loading -	5.32.3.4
Stress/Force output printing -	5.42
Shear Wall Design -	3.8.2, 5.54

General format:

SURFACE CONSTANTS

$$\left\{ \begin{array}{l} \textbf{E} \\ \textbf{POISSON} \\ \textbf{G} \\ \textbf{DENSITY} \\ \textbf{ALPHA} \end{array} \right\} \quad \textbf{f} \quad \left[\begin{array}{l} \textbf{LIST surface} - \textbf{list} \\ \textbf{ALL} \end{array} \right]$$

where f is one of the following, as appropriate:

Young's Modulus (E), Poisson's Ratio, Modulus of Rigidity (G), Weight density, Coefficient of thermal expansion, all in current units. In lieu of numerical values, built-in material names may be used for the above specification of constants. The built-in names are STEEL, CONCRETE and ALUMINUM.

Example

```
SURFACE CONSTANTS
E 3150 LIST 1 TO 4
POISSON 0.17 ALL
DENSITY 8.68e-005 LIST 1 TO 4
ALPHA 5.5e-006 LIST 1 TO 4
```

Notes:

1. If G is not specified, but Poisson is specified, G is calculated from $E / [2(1+\text{Poisson})]$.
2. If neither G nor Poisson is specified, Poisson is assumed based on E, and G is then calculated.
3. If G and Poisson are both specified, the input value of G is used, G is not calculated in this situation.
4. If G and Poisson are both required in the analysis, such as for the stiffness matrix of plate elements, and G is specified, but Poisson is not, then, Poisson is calculated from [(E/2G) –1].
5. To obtain a report of the values of these terms used in the analysis, specify the command PRINT MATERIAL PROPERTIES.

5.26.4 Modal Damping Information

Purpose

To define unique modal damping ratios for every mode. If all modes have the same damping, then enter damping with the Define Time History Load or with the Dynamic Loading commands.

Damping may be entered here

1. EXPLICITly for some or all modes;
2. by specifying that STAAD EVALUATE each mode's damping based on the frequency of the mode and the minimum and maximum damping entered here. The formula used to evaluate the damping is given below.
3. by specifying that STAAD CALCULATE each mode's damping based on the frequency of the mode and the mass factor, ALPHA, and the STIFFNESS factor, BETA. The formula used to calculate the damping is given below.

The damping entered will be used in Dynamic Analyses such as Time History load cases; Response Spectrum load cases that use the CQC/ASCE4 methods and/or Spectra vs. Period curves versus damping; and Steady State load cases.

General Syntax

DEFINE DAMPING INFORMATION

```
{ EVALUATE  dmin  (dmax)                                   }
{ CALC      ALPHA  c1   BETA c2   MAX  c3   MIN  c4        }
{ EXPLICIT  d1  (d2  d3  d4  d5 ….)                        }
END
```

1. Enter dmin and dmax as the minimum and maximum damping ratios respectively to be used in the formula below, or

2. Alternatively enter c1 and/or c2 (optionally include the limits of c3 and c4) to be used in the formula below, or
3. Alternatively enter d1, d2, d3, etc. as the damping ratios for each mode. With the Explicit option each value can be preceded by a repetition factor (rf*damp) without spaces. For example:

   EXPLICIT 0.03 7*0.05 0.04 <= mode 1 damping is .03, modes 2 to 8 are .05, mode 9 is .04.

If there are fewer entries than modes, then the last damping entered will apply to the remaining modes. This input may be continued to 10 more input lines with word EXPLICIT only on line 1; end all but last line with a space then a hyphen. There may be additional sets of EXPLICIT lines before the END.

Damping ratios must be in the range 0.0 through 1.0.

The formula used for CALCULATE (to calculate the damping per modal frequency based on mass and/or stiffness proportional damping) is:

D(i) $= (\alpha / 2\omega_i) + (\omega_i \ \beta / 2)$ = damping ratio for modes i = 1 to N. If the resulting damping is greater than MAX, then MAX will be used (MAX=1 by default). If the resulting damping is less than MIN, then MIN will be used (MIN=1.E-9 by default). This is the same damping as $D = (\alpha M + \beta K)$.

Example: To get 4% damping ratio at 4 Hz and 6% damping ratio at 12 Hz

Mode	Hz	Rad/sec	Damp Ratio
1	4.0	25.133	0.04
3	12.0	75.398	0.06

$D(i) = (\alpha / 2\omega_i) + (\omega_i \ \beta / 2)$

$.04 = \alpha / 50.266 + 12.567 \beta$

$.06 = \alpha / 150.796 + 37.699 \beta$

$\alpha = 1.13097$

$\beta = .0013926$

However they are determined, the α and β terms are entered in the CALC data above. For this example calculate the damping ratio at other frequencies to see the variation in damping versus frequency.

Mode	Hz	Rad/sec	Damp Ratio
1	4.0	25.133	.04000
3	12.0	75.398	.06000
	2	12.0664	.05375
	8	50.2655	.04650
	20	120.664	.09200
	4.5	28.274	.03969

The damping, due to β times stiffness, increases linearly with frequency; and the damping, due to alpha times mass, decreases parabolicly. The combination of the two is hyperbolic.

The formula used for EVALUATE (to evaluate the damping per modal frequency) is:

Damping for the first 2 modes is set to dmin from input.
Damping for modes i = 3 to N given dmin and the first two frequencies ω_1 and ω_2 and the ith modal frequency ω_i .
$A_1 = dmin / (\omega_1 + \omega_2)$
$A_0 = A_1 * \omega_1 * \omega_2$

$D(i) = (A_0 / \omega_i) + (A_1 * \omega_i)$ damping for modes i = 3 to N.
If the resulting damping is greater than the dmax value of maximum damping, then dmax will be used.

Example: (for dmin = .02 , dmax = .12 and the ω_i given below)

Mode	ω_i	Damping ratio	
1	3	.0200	
2	4	.0200	
3	6	.0228568	
N	100	.1200	(calculated as .28605 then reset to maximum entered)

5.26.5 Composite Damping for Springs

Purpose

This command may be used to designate certain support springs as contributing to the computation of modal damping by the composite damping method. The Response Spectrum or Time History dynamic response analyses must select composite damping for this data to have any effect on results.

General Format:

SPRING DAMPING

$$\textbf{joint - list} \quad * \left\{ \begin{array}{ll} \underline{\textbf{KFX}} & \mathbf{f_1} \\ \underline{\textbf{KFY}} & \mathbf{f_2} \\ \underline{\textbf{KFZ}} & \mathbf{f_3} \end{array} \right\}$$

Description

At least one of KFX, KFY, or KFZ must be entered. Each one entered must have a spring damp value following it. $\mathbf{f_1}$ $\mathbf{f_2}$ $\mathbf{f_3}$ are damping ratios (0.001 to 0.990).

If this Spring Damping command is entered, then all springs in the structure are included in the composite damping calculation, otherwise no spring is considered in that calculation.

This input command does not create a spring, only that if a support spring exists at the joint in the specified direction then it will be assigned the damping ratio. See Section 5.27.1 to 5.27.3 to define springs. This is not a discrete damper definition.

5.26.6 Member Imperfection Information

Purpose

To define camber and drift specifications for selected members. Drift is usually for columns and camber for beams.

General Format:

DEFINE IMPERFECTION

CAMBER { **Y** | **Z** } **(f_1)** **RESPECT (f_2)** * { **XR** f_4 f_5 | **YR** f_4 f_5 | **ZR** f_4 f_5 | **MEM** memb-list | **LIST** memb-list | **ALL** }

DRIFT { **Y** | **Z** } **(f_3)** * { **XR** f_4 f_5 | **YR** f_4 f_5 | **ZR** f_4 f_5 | **MEM** memb-list | **LIST** memb-list | **ALL** }

Where

f1	Camber value. Default = 300.
f2	Respect value. Default = 1.6 .
f3	Drift value. Default = 200.
f4 and f5	global coordinate values to specify X or Y or Z range for member selection.

Imperfections will be simulated by loads. These loads will be generated for the specified members if there is an Imperfection Analysis specified and if the specified members are active, in compression, and are not truss or tension/compression only members. Please refer to sections 1.18.2.4 and 5.37 for more details.

Notes:

Camber is the maximum offset of the neutral axes in the defined direction from a vector that passes through the ends of the beam (i.e. the local X axis) defined as the ratio of offset/member length.

Drift is the offset of the end a member from its specified location defined as a ratio of offset/member length.

RESPECT is a non dimensional constant used to skip the camber imperfection calculation if the compressive load is small or EI is great or length is short. A combination of these terms is calculated and called EPSILON. If EPSILON is less than RESPECT, then the imperfection calculation is skipped for that local direction, for that case, for that member. The imperfection calculation is also skipped for any member that is in tension.

EPSILONy = Length * SQRT[(abs(axial load)) / EIz]
EPSILONz = Length * SQRT[(abs(axial load)) / EIy]

5.27 Support Specifications

STAAD support specifications may be either parallel or inclined to the global axes. Specification of supports parallel to the global axes is described in Section 5.27.1. Specification of inclined supports is described in Section 5.27.2.

5.27.1 Global Support Specification

Purpose

This set of commands may be used to specify the SUPPORT conditions for supports parallel to the global axes.

For SURFACE elements, if nodes located along a straight line are all supported identically, as in the case of the base of a wall, support generation can be performed for assigning restraints to those nodes. See the “GENERATE” option in the command syntax below. The user needs to provide only the starting and ending nodes of the range, and the type of restraint.

General format:

<u>SUP</u>PORTS

$$\left\{\begin{array}{l}\textbf{joint-list}\\ \textbf{ni TO nj }\underline{\textbf{GEN}}\textbf{ERATE}\end{array}\right\}\left\{\begin{array}{l}\underline{\textbf{PIN}}\textbf{NED}\\ \\ \underline{\textbf{FIX}}\textbf{ED}\quad\textbf{(}\underline{\textbf{BUT}}\textbf{ release-spec [spring-spec.]}\\ \underline{\textbf{ENF}}\textbf{ORCED}\quad\textbf{(}\underline{\textbf{BUT}}\textbf{ release-spec)}\end{array}\right\}$$

$$\textbf{release-spec} = \left\{\begin{array}{l}\underline{\textbf{FX}}\\ \underline{\textbf{FY}}\\ \underline{\textbf{FZ}}\\ \underline{\textbf{MX}}\\ \underline{\textbf{MY}}\\ \underline{\textbf{MZ}}\end{array}\right\}$$

$$\textbf{spring-spec} = * \left\{\begin{array}{ll}\underline{\textbf{KFX}} & \mathbf{f_1}\\ \underline{\textbf{KFY}} & \mathbf{f_2}\\ \underline{\textbf{KFZ}} & \mathbf{f_3}\\ \underline{\textbf{KMX}} & \mathbf{f_4}\\ \underline{\textbf{KMY}} & \mathbf{f_5}\\ \underline{\textbf{KMZ}} & \mathbf{f_6}\end{array}\right\}$$

Description of Pinned and Fixed

See Section 1.14

PINNED support is a support that has translational, but no rotational restraints. In other words, the support has no moment carrying capacity. A FIXED support has both translational and rotational restraints. A FIXED BUT support can be released in the global directions as described in release-spec (FX for force-X through MZ for moment-Z). Also, a FIXED BUT support can have spring constants as described in spring-spec (translational spring in global X-axis as KFX through rotational spring in global Z-axis as KMZ). Corresponding spring constants are f_1 through f_6. The rotational spring constants are always per degree of rotation. All six releases may be provided as may be required when using the CHANGE command. If both release specifications and spring specifications are to be supplied for the same support joint, release specifications must come first.

Example

```
SUPPORTS
1 TO 4  7 PINNED
5  6 FIXED BUT FX MZ
8  9 FIXED BUT MZ KFX 50.0 KFY 75.
18  21 FIXED
27 FIXED BUT KFY 125.0
```

In this example, joints 1 to 4 and joint 7 are pinned. No moments are carried by those supports. Joints 5 and 6 are fixed for all DOF except in force-X and moment-Z. Joints 8 and 9 are fixed for all DOF except moment-Z and have springs in the global X and Y directions with corresponding spring constants of 50 and 75 units respectively. Joints 18 and 21 are fixed for all translational and rotational degrees of freedom. At joint 27, all the DOF are fixed except the FY DOF where it has a spring with 125 units spring constant.

Notes

1) Users are urged to refer to Section 5.38 for information on specification of SUPPORTS along with the CHANGE command specifications.
2) Spring constants must be provided in the current units.
3) All spring DOF must be entered after the last non-spring DOF is specified, if both are on the same line.
4) If there are two entries for the same joint, then:
   a) any direction that is pinned/fixed on either will be fixed in that direction.
   b) any direction released on one and is a spring on the other will use the spring.
   c) Any direction that is pinned/fixed on one and a spring on the other will use the spring.
   d) Any direction that is a spring on 2 or more entries will have the spring constants added.

Description of Enforced

Enforced Support defines which translational and rotational directions, at a joint, may have a support displacement imposed. The support displacements are defined for each load case in section 5.32.8. If no support displacement is entered, then zero displacement will be imposed, as if that direction was FIXED. The enforced displacement directions will be fixed for dynamic load cases.

If there are two entries for the same joint, then any direction that is enforced on either will be enforced in that direction, overriding any other support specification for that joint-direction.

Example for support generation

```
SUPPORTS
3 TO 7 GENERATE PIN
```

The above command will generate pinned supports for all joints located between nodes No. 3 and 7 along a straight line. This may include joints explicitly defined by the user or joints generated by the program for internal use only (e.g., as a result of SET DIVISION and SURFACE INCIDENCES commands).

Currently the support generation command can only be used in conjunction with the **Surface** element support specifications.

5.27.2 Inclined Support Specification

Purpose

These commands may be used to specify supports that are inclined with respect to the global axes.

General Format:

SUPPORT

joint-list INClined $\left\{ \begin{array}{l} f_1\ f_2\ f_3 \\ \textbf{REF}\ f_4\ f_5\ f_6 \\ \textbf{REFJT}\ f_7 \end{array} \right\}$ $\left\{ \begin{array}{l} \textbf{PINNED} \\ \textbf{FIXED}\ \textbf{(BUT release-spec[spring-spec.])} \\ \textbf{ENFORCED}\ \textbf{(BUT release-spec)} \end{array} \right\}$

where f_1, f_2, f_3 are x, y, z global distances from the joint to the “reference point"; or
where f_4, f_5, f_6 are x, y, z global coordinates of the “reference point"; or
where f_7 is a joint number whose x, y, z global coordinates is the “reference point".

A vector from the joint location to the reference point location serves to define a local coordinate system (same as member with BETA=0). The inclined support directions are in this local "Inclined Support Axis System" (see more below).

See Section 1.14

Note the release-spec and spring-spec are the same as in the previous section (5.27.1). However, FX through MZ and KFX through KMZ refer to forces/moments and spring constants in the "Inclined Support Axis System" (see below).

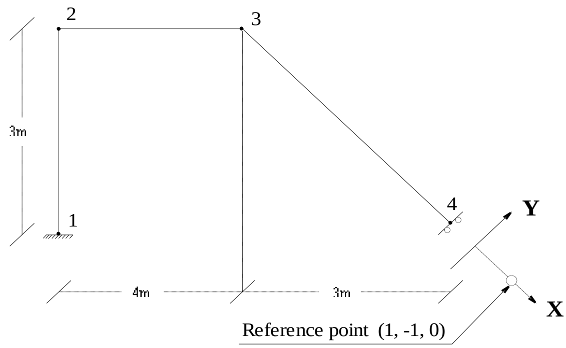


Figure 5.18

Inclined Support Axis System

The INCLINED SUPPORT specification is based on the "Inclined Support axis system". The local x-axis of this system is defined by assuming the inclined support joint as the origin and joining it with a "reference point" with co-ordinates of f_1, f_2 and f_3 (see figure) measured from the inclined support joint in the global coordinate system.

The Y and Z axes of the inclined support axis system have the same orientation as the local Y and Z axes of an imaginary member whose BETA ANGLE is zero and whose incidences are defined from the inclined support joint to the reference point. Users may refer to section 1.5.3 of this manual for more information on these concepts.

Support displacements in inclined directions are not permitted on inclined supports.

Example

```
SUPPORT
4 INCLINED 1.0 -1.0 0.0 FIXED BUT FY MX MY MZ
```

Notes

Inclined support directions are assumed to be same as global when computing some dynamic and UBC intermediate results (e.g. global participation factors). If masses and/or forces in the free directions at inclined supports are a relatively small portion of the overall forces, then the effect should be very small.

5.27.3 Automatic Spring Support Generator for Foundations

STAAD has a facility for automatic generation of spring supports to model footings and foundation mats. This command is specified under the SUPPORT command.

General Format:

```
SUPPORT

                                                    { X     }
{ joint-list  ELASTIC  FOOting  f1 (f2) }           { XOnly }
{ joint-list  ELASTIC  MAT              }  DIR      { Y     }  SUBgrade  f3
{ plate-list  PLATE    MAT              }           { YOnly }
                                                    { Z     }
                                                    { ZOnly }

        (PRINT)  ( { COMP  } )
                   { MULTI }

plate-list  PLATE  MAT    DIR   ALL   SUBgrade  f3  (f4  f5)

        (PRINT)  ( { COMP  } )
                   { MULTI }
```

where

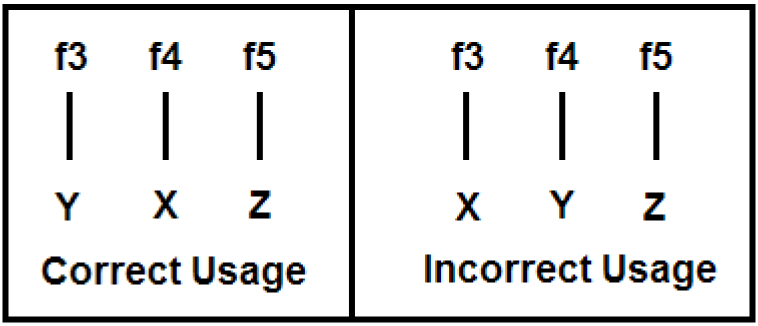
f1, f2 = Length and width of the footing. If f2 is not given, the footing is assumed to be a square with sides f1

X,Y,Z = Global direction in which soil springs are to be generated

f_3 = Soil sub-grade modulus in force/area/length units.

ALL option

f3, f4, f5 = Soil sub-grade modulus in force/area/length units in Y, X, Z directions respectively. f4, f5 default to f3 if omitted.



Do not use this command with SET Z UP.

The ELASTIC FOOTING option : If you want to specify the influence area of a joint yourself and have STAAD simply multiply the area you specified by the sub-grade modulus, use the FOOTING option. Situations where this may be appropriate are such as when a spread footing is located beneath a joint where you want to specify a spring support. Please note that it is absolutely imperative that you provide f1 (and f2 if its a non-square footing) if you choose the FOOTING option.

The ELASTIC MAT option : If you want to have STAAD calculate the influence area for the joint (instead of you specifying an area yourself) and use that area along with the sub-grade modulus to determine the spring stiffness value, use the MAT option. Situations where this may be appropriate are such as when a slab is on soil and carries the weight of the structure above. You may have modeled the entire slab as finite elements and wish to generate spring supports at the nodes of the elements.

The PLATE MAT option : Similar to the Elastic Mat except for the method used to compute the influence area for the joints. If your mat consists of plate elements and all of the influence areas are incorporated in the plate areas, then this option is preferable. Enter a list of plates or YRANGE f1 f2 at the beginning of the command, the joint influence areas are then calculated using the same principles as joint forces would be from uniform pressure on these plates. This method overcomes a major limitation of the Delaunay triangle method used in the ELASTIC MAT option, which is that the contour formed by the nodes of the mat must form a convex hull.

The PLATE MAT DIR ALL option : Similar to the Plate Mat except that the spring supports are generated in all 3 directions. If the compression only option is also specified, then the compression direction will be assumed to be in the Y direction. If the Y spring at a joint goes slack (lift off), then the X and Z spring stiffnesses for that joint will also be set to zero. Otherwise the X and Z springs act in both directions. The influence area for the X and Z springs is the same as used for the Y spring. Three values of subgrade reaction may be entered, the first is for the Y direction, the second for X and the third for Z.

The DIRection option : The keyword DIR is followed by one of the alphabets X, Y or Z (or XONLY, YONLY, or ZONLY) which indicate the direction of resistance of the spring supports. If X or Y or Z is selected then a spring support is generated in that direction plus 3 other directions receive a fixed support, e.g. if Y is selected, then FY is supported by a spring; FX and FZ and MY are fixed supports; and MX and MZ are free. If XONLY, YONLY, or ZONLY are selected then only a spring support in that direction is generated.

The SUBGRADE option : The keyword SUBGRADE is followed by the value of the subgrade reaction. The value should be provided in the current unit system signified by the most recent UNIT statement prior to the SUPPORT command.

The PRINT option : Prints the influence area of each joint.

The COMP option : The springs generated will be compression only.

The MULTI option : The springs generated will be multilinear. Add the associated multilinear curve input after each MAT command (with the multi option) to describe the displacement-spring constant curve. See section 5.27.4 for additional information on this input format. The actual spring constant used will be the subgrade modulus (f3 entered above) times the influence area (computed by STAAD) times the **s_i** values entered in the curve (so the curve stiffness values will likely be between 0.0 and 1.0).

SPRINGS d_1 s_1 d_2 s_2 ...... d_n s_n

Example

```
SUPPORTS
1 TO 126   ELASTIC  MAT  DIREC  Y  SUBG 200.
1 TO 100   PLATE  MAT  DIREC  Y  SUBG 200.
YR -.01 .01  PLA  MAT  DIR  Y  SUBG 200  MUL
SPRINGS  -0.51 40.0  -0.50 50.0  0.5 65.0
```

The first command above instructs STAAD to internally generate supports for all nodes 1 through 126 with elastic springs. STAAD first calculates the influence area perpendicular to the global Y axis of each node and then multiplies the corresponding influence area by the soil subgrade modulus of 200.0 to calculate the spring constant to be applied to the node. In the 2nd example, the nodes of plates 1 to 100 are assigned spring supports, generated using a subgrade modulus of 200 units.

Notes:

A closed surface is generated by the program based on the joint-list that accompanies the ELASTIC MAT command. The area within this closed surface is determined and the share of this area for each node in the list is then calculated.

Hence, while specifying the joint-list, one should make sure that these joints make up a closed surface. Without a proper closed surface, the area calculated for the region may be indeterminate and the spring constant values may be erroneous. Consequently, the list should have at a minimum, 3 nodes.

The internal angle formed by 2 adjacent segments connecting 3 consecutive nodes in the list should be less than 180 degrees. In other words, the region should have the shape of a convex polygon. The example below explains the method that may be used to get around a situation where a convex polygon is not available.
For the model comprised of plate elements 100 to 102 in the figure below, one wishes to generate the spring supports at nodes 1 to 8. However, a single ELASTIC MAT command will not suffice because the internal angle between the edges 1-8 and 8-7 at node 8 is 270 degrees, which violates the requirements of a convex polygon.

So, one should break it up into 2 commands:

```
1 2 3 8 ELASTIC MAT DIREC Y SUBG 200.
3 4 5 6 7 8 ELASTIC MAT DIREC Y SUBG 200.
```

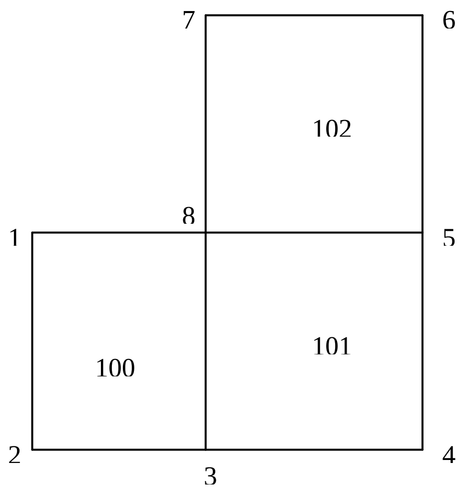


Figure 5.19

Joints 3 and 8 will hence get the contribution from both of the above commands.

The command works only when the plane of the closed region is parallel to one of the global planes X-Y, Y-Z or X-Z. For regions that are inclined to one of the global planes, the spring constant will have to be evaluated manually and specified using the FIXED BUT type of spring support.

5.27.4 Multi-linear Spring Support Specification

When soil is modeled as spring supports, the varying resistance it offers to external loads can be modeled using this facility, such as when it becomes stiffer as it is compressed. Another application of this facility is when the behavior of soil in tension differs from its behavior in compression.

General format:

MULTILINEAR SPRINGS

joint-list SPRINGS d_1 s_1 d_2 s_2 ...... d_n s_n

See Section 5.27.5 also

where (di si) pairs represent displacement and spring constant pairs (si is zero or positive), starting from the maximum negative displacement to the maximum positive displacement.

The first pair defines the spring constant from negative infinity displacement up to the displacement defined in the second pair. The second pair define the spring constant when the support displaces in the range from the displacement defined in the second pair, up to the displacement defined in the third pair. This continues for each displacement and spring constant pair until the last pair which defines the spring constant for displacements greater than the displacement in the last pair to positive infinity.

Example

```
UNIT ...
SUPPORT
1 PINNED; 2 4 FIXED BUT KFY 40.0
MULTILINEAR SPRINGS
2 4 SPRINGS -1 40.0 -0.50 50.0 0.5 65.0
```

Load-Displacement characteristics of soil can be represented by a multi-linear curve. Amplitude of this curve will represent the spring characteristic of the soil at different displacement values. A typical spring characteristic of soil may be represented as the step curve as shown in the figure below. In the above example, the multi-linear spring command specifies soil spring at joints 2 and 4. (Note that the amplitude of the step curve does not change after the first point.)

Each load case in a multi-linear analysis must be separated by a CHANGE command and have its own PERFORM ANALYSIS command. There may not be any PDELTA, dynamics, CABLE or TENSION/COMPRESSION analysis included. The multi-linear spring command will initiate an iterative analysis and convergence check cycle. The cycles will continue until the RMS of the effective spring rates used remain virtually the same for 2 consecutive cycles.

Notes:

1. SUPPORT springs must have previously been entered for each spring entered here. For the first cycle, the spring value used will be the support spring value (not the zero displacement value here). Use a realistic and stable value.
2. All directions that have been defined with an initial spring stiffness in the SUPPORT command will become multi-linear with this one curve.
3. This command can be continued to up to 11 lines by ending all but last with a hyphen. The semi-colons and the XRANGE, YRANGE, ZRANGE list items may not be used.
4. This command needs a minimum of two displacement and spring constant pairs.

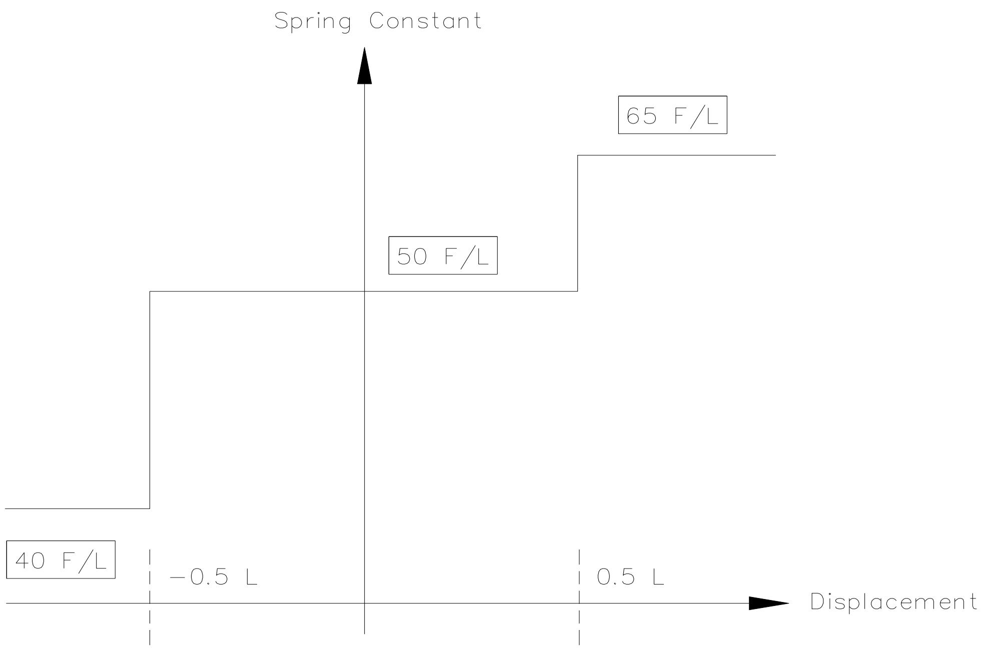


Figure 5.20

F = Force Units
L = Length units
Spring constant is always positive or zero.

5.27.5 Spring Tension/Compression Specification

Purpose

This command may be used to designate certain support springs as Tension-only or Compression-only springs.

General Format:

<u>SPR</u>ING <u>TEN</u>SION
joint – list spring-spec

<u>SPR</u>ING <u>COM</u>PRESSION
joint – list spring-spec

spring-spec = * { <u>KFX</u> / <u>KFY</u> / <u>KFZ</u> / <u>ALL</u> }

Description

Tension-only springs are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that create compression in them. Compression-only springs are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that create tension in them.

See Section 1.9

If no spring spec is entered or ALL is entered then all translational springs at that joint will be tension (or compression) only. This input command does not create a spring, only that if a support spring exists at the joint in the specified direction then it will also be tension (or compression) only. See Section 5.27.1 to 5.27.3 to define springs.

For compression only springs the ALL parameter has special meaning. The compression only spring is in the Y direction; the X and Z direction springs are bi-directional. However when the Y direction spring goes slack, the X and Z springs at the same joint are made inactive as well.

The procedure for analysis of Tension-only or Compression-only springs requires iterations for every load case and therefore may be quite involved.

Since this command does not specify whether the spring is in the positive or negative direction from the joint, it is assumed in STAAD to be in the negative direction. For negative displacement the spring is in compression and for positive the spring is in tension.

If a CHANGE command is used (because of a change in list of tension springs, supports, etc.), then the SET NL command must be used to convey to STAAD that multiple analyses and multiple structural conditions are involved.

Example

```
SPRING TENSION
12 17 19 TO 37 65
SPRING COMPRESSION
5 13 46 TO 53 87   KFY
```

Notes

1) A spring declared as TENSION only or COMPRESSION only will carry axial forces only. It will not carry moments.
2) The SPRING TENSION / COMPRESSION commands should not be specified if the INACTIVE MEMBER command is specified.
3) Do not use Load Combination to combine these cases. Tension/Compression cases are non-linear and should not be linearly combined as in Load Combination. Use a primary load case with the Repeat Load command.
4) The following is the general sequence of commands in the input file if the SPRING TENSION or COMPRESSION

command is used. This example is for the SPRING TENSION command. Similar rules are applicable for the SPRING COMPRESSION command. The dots indicate other input data items.

```
STAAD ...
SET NL ...
UNITS ...
JOINT COORDINATES
...
MEMBER INCIDENCES
...
ELEMENT INCIDENCES
...
CONSTANTS
...
MEMBER PROPERTY
...
ELEMENT PROPERTY
...
SUPPORTS
...
SPRING TENSION
...
LOAD 1
...
LOAD 2
...
LOAD 3
REPEAT LOAD
...
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ...
PRINT ...
```

```
PARAMETER
...
CHECK CODE ...
FINISH
```

a) See Section 5.5 for explanation of the SET NL command. The number that follows this command is an upper bound on the total number of primary load cases in the file.

b) STAAD performs up to 10 iterations automatically, stopping if converged. If not converged, a warning message will be in the output. Enter a SET ITERLIM i command (i>10) before the first load case to increase the default number of iterations. Since convergence may not be possible using this procedure, do not set the limit too high. If not converged, a message will be in the output.

c) The principle used in the analysis is the following.

- The program reads the list of springs declared as SPRING TENSION and/or COMPRESSION.
- The analysis is performed for the entire structure and the spring forces are computed.
- For the springs declared as SPRING TENSION / COMPRESSION, the program checks the axial force to determine whether it is tensile or compressive. Positive displacement is TENSION. If the spring cannot take that load, the spring is "switched off" from the structure.
- The analysis is performed again without the switched off springs.
- Up to ITERLIM iterations of the above steps are made for each load case.
- This method does not always converge and may become unstable. Check the output for instability messages. Do not use results if the last iteration was unstable. You may need to include some support in

each global direction that is not tension (or compression) only to be stable on every iteration.

d) A revised SPRING TENSION / COMPRESSION command and its accompanying list of joints may be provided after a CHANGE command. If entered, the new SPRING commands replace all prior SPRING commands. If not entered after a CHANGE, then the previous spring definitions are used.

e) The SPRING TENSION command should not be used if the following load cases are present : Response Spectrum load case, Time History Load case, Moving Load case. If used, the SPRING TENSION /COMPRESSION will be ignored in all load cases.

f) If the SPRING TENSION/COMPRESSION command is used in a model with UBC, IBC or other such seismic load cases, each such load case must be followed by an ANALYSIS and CHANGE command.

5.28 Master/Slave Specification

Purpose

This set of commands may be used to model specialized linkages (displacement tying, rigid links) through the specification of MASTER and SLAVE joints. Please read the notes for restrictions.

General format:

$$\mathbf{\underline{SLA}VE} \quad * \left\{ \begin{array}{l} \mathbf{\underline{XY}} \\ \mathbf{\underline{YZ}} \\ \mathbf{\underline{ZX}} \\ \mathbf{\underline{RIG}ID} \\ \mathbf{\underline{FX}} \\ \mathbf{\underline{FY}} \\ \mathbf{\underline{FZ}} \\ \mathbf{\underline{MX}} \\ \mathbf{\underline{MY}} \\ \mathbf{\underline{MZ}} \end{array} \right\} \quad \mathbf{\underline{MAS}TER} \quad \mathbf{j} \quad \mathbf{\underline{JOI}NT} \quad \textbf{joint-spec}$$

$$\textbf{joint-spec} = \left\{ \begin{array}{l} \textbf{joint-list} \\ \\ * \left\{ \begin{array}{l} \mathbf{\underline{XRA}NGE} \\ \mathbf{\underline{YRA}NGE} \\ \mathbf{\underline{ZRA}NGE} \end{array} \right\} \quad \mathbf{f_1, f_2} \end{array} \right\}$$

Description

The master/slave option provided in STAAD allows the user to model specialized linkages (displacement tying, rigid links) in the system. For example, SLAVE FY … connects the two joints such that the Y displacement at the slave will be the sum of Y displacement at the master plus the rigid rotation, R SIN θ.

Notice that instead of providing a joint list for the slaved joints, a range of coordinate values (in global system) may be used. All joints whose coordinates are within the range are assumed to be slaved joints. For convenience, the coordinate range specified for slaved joints in this entry may include the master joint for this entry. However, master and slave joints of other entries must not be included in the coordinate range. All 2 or 3 ranges can be entered to form a “tube” or “box” for selecting joints in the tube or box region.

Fx, Fy etc. are the directions in which they are slaved (any combination may be entered).

If two or more consecutive entries have the same master, the slave lists will be merged. It is also OK to have different direction specs.

The direction specifiers (XY, YZ, ZX) are combinations of the basic directions, XY is the same as entering FX, FY, MZ; etc. Any combination of direction specifiers may be entered. An example of the use of this format is: a rigid diaphragm floor that still retains bending flexibility entered as SLA ZX … .

If RIGID or if all directions are provided, the joints are assumed to be rigidly connected.

Restrictions

- Solid elements may not be connected to slave joints.
- Master joints may not be slaves in another entry.
- Slave joints may not be master or slaves in another entry.
- Slave directions at joints may not be supported directions or have displacements imposed.
- Master and/or slave joints may not be inclined supports.
- The master / slave specification is only intended for linear static and dynamic analysis.
- Multilinear springs are not permitted.

Example - Fully Rigid and Rigid Floor Diaphragm

```
SLAVE RIGID MASTER 22 JOINT 10 TO 45
SLAVE RIGID MASTER 70 JOINT YR 25.5 27.5
SLA  ZX  MAS  80  JOINT  YR  34.5  35.5
```

Enhancement effective from STAAD.Pro 2007 Build 1001

The internal processing of any Master Slave command has been enhanced to allow an automatic bandwidth reduction to take place.

The analysis engine now performs a bandwidth reduction on files that include Master/Slave commands which must occur in the input file after the definition of supports. In previous versions of STAAD.Pro, for the bandwidth reduction to take place, the data of Master/Slave would need to be repeated before the support definitions. This requirement is now no longer required.

5.29 Draw Specifications

Purpose

This command has been discontinued in STAAD.Pro. Please use the Graphical User Interface for screen and hard copy graphics.

5.30 Miscellaneous Settings for Dynamic Analysis

When dynamic analysis such as frequency and mode shape calculation, response spectrum analysis and time history analysis is performed, it involves eigenvalue extraction and usage of a certain number of modes during the analysis process. These operations are built around certain default values. This section explains the commands required to override those defaults.

Related topics can be found in the following sections:

Frequency and Mode Extraction -	5.34
Response Spectrum Analysis -	5.32.10.1
Time History Analysis -	5.31.4, 5.32.10.2
Steady State/Harmonic Analysis	5.37.6

5.30.1 Cut-Off Frequency, Mode Shapes or Time

Purpose

These commands are used in conjunction with dynamic analysis. They may be used to specify the highest frequency or the number of mode shapes that need to be considered.

General Format:

See Section 1.18.3

$$\underline{\text{CUT}}\ (\text{OFF}) \begin{Bmatrix} \underline{\text{FRE}}\text{QUENCY} & f_1 \\ \underline{\text{MOD}}\text{E SHAPE} & i_1 \\ \underline{\text{TIM}}\text{E} & t_1 \end{Bmatrix}$$

Where,

f_1 = Highest frequency (cycle/sec) to be considered for dynamic analysis.

i_1 = Number of mode shapes to be considered for dynamic analysis. If the cut off frequency command is not provided, the cut off frequency will default to 108 cps. If the cut off mode shape command is not provided, the first six modes will be calculated. These commands should be provided prior to the loading specifications.

t_1 = Ending time for a time history analysis. If zero (default), the time history will end when the last forcing function ends.

A maximum of i_1 mode shapes will be computed regardless of f_1. If during convergence testing, the 0 through f_1 frequencies are converged, then the modal calculation will be completed before i_1 mode shapes are calculated.

If the CUT OFF FREQ f1 and CUT OFF MODE i1 commands are both entered, then after completing each iteration step in the Subspace iteration, convergence testing is performed. If every frequency from 0.0 to f1 meets the convergence tolerance, then the Subspace iteration is done. Similarly, if every mode from 0 to i1 meets the convergence tolerance, then the Subspace iteration is done. If the cut off frequency f1 results in fewer modes than i1, then only those frequencies up to the cut off are used. If the cut off frequency would result in more modes than i1, then only the first i1 modes are used. That is, the modes cut off takes precedence over the frequency cut off.

5.30.2 Mode Selection

Purpose

This command allows specification of a reduced set of active dynamic modes. All modes selected by this command remain selected until a new MODE SELECT is specified.

General format:

<u>MODE</u> <u>SEL</u>ECT mode_list

Description

This command is used to limit the modes used in dynamic analysis to the modes listed in this command and deactivate all other modes that were calculated but not listed in this command. If this command is not entered, then all modes calculated are used in the dynamic analysis.

Example

```
CUT OFF MODES 10
MODE SELECT 1 TO 3
```

In this example, 10 modes will be calculated but only modes 1 and 3 will be used in dynamic analysis.

Notes

Do not enter this command within the loads data (from the first Load command in an analysis set down to the associated Analysis command).

The advantage of this command is that one may find the amount of structural response generated from a specific mode or a set of modes. For example, if 50 modes are extracted, but the effect of just the 40^{th} to the 50^{th} mode in a response spectrum analysis is to be determined, one may set the active modes to be 40 through 50. The results will then be devoid of any contribution from modes 1 through 39.

5.31 Definition of Load Systems

Purpose

This section describes the specifications necessary for defining various load systems, for automatic generation of Moving loads, UBC Seismic loads and Wind loads. In addition, this section also describes the specification of Time History load for Time History analysis.

Description

STAAD has built-in algorithms to generate moving loads, lateral seismic loads (per the Uniform Building Code), and wind loads on a structure. Use of the load generation facility consists of two parts:

1) Definition of the load system(s).
2) Generation of primary load cases using previously defined load system(s).

See Section 1.17

Definition of the load system(s) must be provided before any primary load case is specified. This section describes the specification of load system(s). Information on how to generate primary load cases using the defined load system(s) is available in Section 5.32.12.

UBC loads do not fully consider the effects of forces at inclined support directions or at slave joint directions. Applying forces at these locations may introduce errors that are generally small.

5.31.1 Definition of Moving Load System

Purpose

This set of commands may be used to define the moving load system. Enter this command only once with up to 200 TYPE commands.

General format:

DEFINE MOVING LOAD (FILE file-name)

TYPE j $\left\{ \begin{array}{l} \textbf{LOAD } f_1,f_2,.f_n \textbf{ (DISTANCE } d_1,d_2,..d_{n-1} \textbf{ (WIDTH w))} \\ \textbf{load-name (f)} \end{array} \right\}$

(DISTANCE $d_1,d_2,..d_{n-1}$ **(WIDTH w))** < optionally as 2^{nd} set >

The MOVING LOAD system may be defined in two possible ways - directly within the input file or using an external file.

See Section 5.32.12

The FILE option should be used only in the second case when the data is to be read from an external file. The filename should be limited to 24 characters.

Moving Loads can be generated for frame members only. They will not be generated for finite elements. Note: All loads and distances are in current unit system.

Define Moving Load within input file

Use the first TYPE specification. Input Data must be all in one line (as shown above) or in two sets of lines. If two sets, then the second set must begin with DIS as shown below. If two sets, then Load and Dist lines may end each line but last of each set with a hyphen (See example).

TYPE j LOAD $f_1,f_2,...f_n$ (set 1)
DISTANCE $d_1,d_2,...d_{(n-1)}$ (WIDTH w) (set 2)

Where,

j = moving load system type number (integer limit of 200 types)
n = number of loads (e.g. axles), 2 to 200.
f_i = value of conc. i^{th} load
d_i = distance between the $(i+1)^{th}$ load and the i^{th} load in the direction of movement
w = spacing between loads perpendicular to the direction of movement. If left out, one dimensional loading is assumed. (e.g. the width of vehicle). This parameter will double the total load since the f_i is applied to each wheel.

NOTE: For a single moving load use: TYPE j LOAD f_1 DIST 0

Define Moving Load using an external file

Use the second TYPE specification.

TYPE j load-name (f)

Where,

load-name	Is the name of the moving load system (maximum of 24 characters).
and f =	Optional multiplying factor to scale up or down the value of the loads. (default = 1.0)

Following is a typical file containing the data.

CS200	-----	name of load system (load-name, must start in column 1)
50. 80. 90. 100.	-----	loads (all on one 79 char input line)
7. 7. 9.	-----	distance between loads (one line)
6.5	-----	width

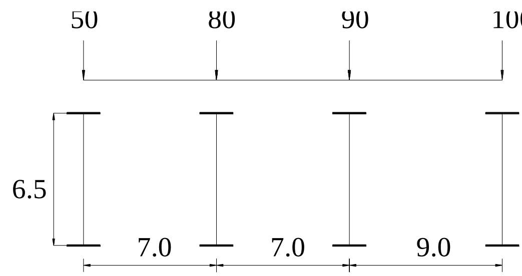


Figure 5.21

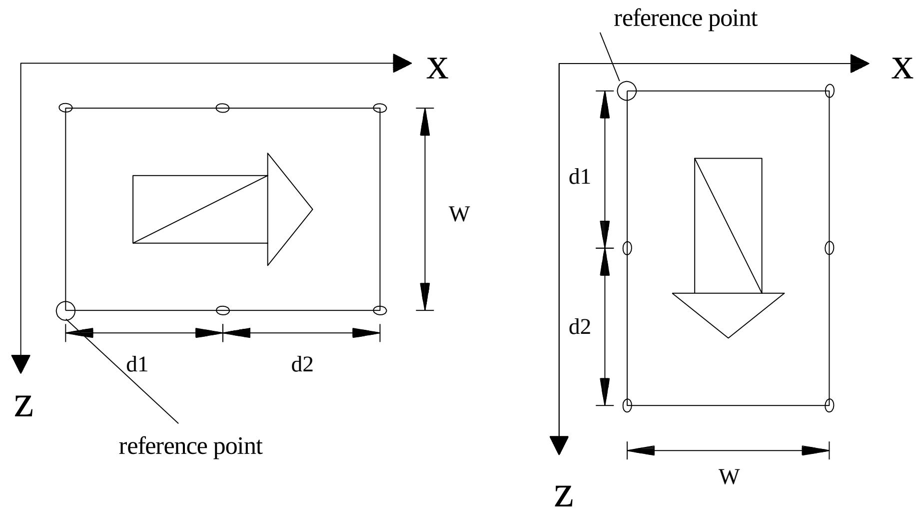
Several load systems may be repeated within the same file.

The STAAD moving load generator assumes:

1) All positive loads are acting in the negative global vertical (Y or Z) direction. The user is advised to set up the structure model accordingly.
2) Resultant horizontal direction of movement is determined from the X and Z (or Y if Z is up) increments of movements as provided by the user.

Reference wheel

The first specified concentrated load in the moving load system is designated as the reference wheel. While generating subsequent primary load cases, the initial position of the load system and the direction of movement are defined with respect to the reference wheel location. Also, when selecting the reference wheel location with a positive value of Width specified, the following two views define the reference wheel location.



Movement parallel to global X axis *Movement parallel to global Z axis*

Figure 5.22

Notice that in the left view, the reference point is on the positive Z wheel track side; whereas in the right view, the reference point is on the least positive X wheel track side.

Specifying standard AASHTO loadings

General format:

See Section 1.17.1

TYPE i { **HS20** / **HS15** / **H20** / **H15** } **(f)** **(vs)**

where,

i = moving load system type no. (integer).

f = optional multiplying factor (default = 1.0)

vs = variable spacing as defined by AASHTO, for HS series trucks (default = 14 ft.)

Example

```
DEFINE MOVING LOAD
TYPE 1 LOAD 10.0 20.0 –
15.0 10.0
DISTANCE 5.0 7.5 –
6.5 WIDTH 6.0
TYPE 2 HS20 0.80 22.0
```

Example

When data is provided through an external file called MOVLOAD

<u>**Data in input file**</u>

```
UNIT . . .
DEFINE MOVING LOAD FILE MOVLOAD
TYPE 1 AXLTYP1
TYPE 2 AXLTYP2 1.25
```

<u>**Data in external file MOVLOAD**</u>

```
AXLTYP1
10 20 15
5.0 7.5
6.0
AXLTYP2
20 20
10
7.5
```

5.31.2 Definitions for Static Force Procedures for Seismic Analysis

See Sections 1.17.2 and 5.32.12

STAAD offers facilities for determining the lateral loads acting on structures due to seismic forces, using the rules available in several national codes and widely accepted publications. The codes and publications allow for so called equivalent static force methods to be used in place of more complex methods like response spectrum and time history analysis. Among the several such codes supported by STAAD are UBC, IBC, IS 1893, AIJ, etc.

Once the lateral loads are generated, the program can then analyze the structure for those loads using the applicable rules explained in the code documents.

5.31.2.1 UBC 1997 Load Definition

Purpose

This feature enables one to generate horizontal seismic loads per the UBC 97 specifications using a static equivalent approach. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

See Sections 1.17.2, 5.32.12 and Examples manual problem no. 14

Methodology

The design base shear is computed in accordance with Section 1630.2.1 of the UBC 1997 code. The primary equation, namely, 30-4, as shown below, is checked.

$$V = \frac{C_v I}{RT} W$$ .... Eqn 30-4 of UBC 1997

In addition, the following equations are checked :

Equation 30-5 – The total design base shear shall not exceed

$$V = \frac{2.5 C_a I}{R} W$$

Equation 30-6 - The total design base shear shall not be less than

$$V = 0.11 C_a I W$$

Equation 30-7 – In addition, for Seismic Zone 4, the total base shear shall also not be less than

$$V = \frac{0.8ZN_vI}{R}W$$

For an explanation of the terms used in the above equations, please refer to the UBC 1997 code.

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE UBC LOAD command.

General Format

DEFINE UBC (ACCIDENTAL) LOAD
ZONE f1 *ubc-spec*
SELFWEIGHT
JOINT WEIGHT
Joint-list **WEIGHT w**

[See Section 5.31.2.2 for complete weight input definition]

ubc-spec **= { I f2, RWX f3, RWZ f4, STYP f5, NA f6, NV f7, (CT f8), (PX f9), (PZ f10)}**

where,

f1 = Seismic zone coefficient. Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.075, 0.15, 0.2, 0.3, 0.4, etc.
f2 = Importance factor
f3 = Numerical coefficient R for lateral load in X direction
f4 = Numerical coefficient R for lateral load in Z direction
f5 = Soil Profile type
f6 = Near source factor Na
f7 = Near source factor Nv
f8 = Optional CT value to calculate time period based on Method A (see Note 7)
f9 = Optional Period of structure (in sec) in X-direction to be used in Method B

f10 = Optional Period of structure (in sec) in Z (or Y if Z up)-direction to be used in Method B

The Soil Profile Type parameter STYP can take on values from 1 to 5. These are related to the values shown in Table 16-J of the UBC 1997 code in the following manner :

STAAD Value	UBC 1997 code value
1	S_A
2	S_B
3	S_C
4	S_D
5	S_E

The soil profile type S_F is not supported.

The seismic zone factor (ZONE) in conjunction with the soil profile type (STYP), Near source factor (Na), and the Near source factor (Nv), is used to determine the values of seismic coefficients Ca and Cv from Tables 16-Q and 16-R of the UBC 1997 code.

Examples

```
DEFINE UBC LOAD
ZONE 0.38  I  1.0  STYP  2 RWX 5.6 RWZ 5.6 NA 1.3 NV 1.6 CT 0.037
SELFWEIGHT
JOINT  WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
MEMBER WEIGHT
12 17 24 UNI 25.7
FLOOR WEIGHT
YRA 9 11 FLOAD 200 XRA –1 21 ZR –1 41
ELEMENT WEIGHT
234 TO 432 PR 150
```

Steps to calculate base shear are as follows:

1. Time Period of the structure is calculated based on clause 1630.2.2.1 (Method A) and 1630.2.2.2 (Method B).
2. The user may override the period that the program calculates using Method B by specifying a value for PX or PZ (Items f9 and f10) depending on the direction of the UBC load. The specified value will be used in place of the one calculated using Method B.
3. The governing Time Period of the structure is then chosen between the above-mentioned two periods on the basis of the guidance provided in clause 1630.2.2.2.
4. From Table 16-Q and 16-R, Ca and Cv coefficients are calculated.
5. The Design Base Shear is calculated based on clause 1630.2.1 and distributed at each floor using the rules of clause 1630.5.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the UBC load (clause 1630.6). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.
7. If the value of C_t is not specified, the program scans the Modulus of Elasticity (E) values of all members and plates to determine if the structure is made of steel, concrete or any other material. If the average E is smaller than 2000 ksi, Ct is set to 0.02. If the average E is between 2000 & 10000 ksi, Ct is set to 0.03. If the average E is greater than 10000 ksi, Ct is set to 0.035. If the building material cannot be determined, Ct is set to 0.035. C_t is in units of seconds/feet$^{3/4}$ or in units of seconds/meter$^{3/4}$. C_t < 0.42 if the units are in feet, and C_t > 0.42 if the units are in meter.
8. Due to the abstractness of the expression "Height above foundation", in STAAD, height "h" is measured above supports. If supports are staggered all over the vertical elevations of the structure, it is not possible to calculate "h" if one doesn't have a clear elevation level from where to measure

"h". Also, the code deals with distributing the forces only on regions above the foundation. If there are lumped weights below the foundation, it is not clear as to how one should determine the lateral forces for those regions.

The following example shows the commands required to enable the program to generate the lateral loads. Users may refer to the LOAD GENERATION section of the Reference Manual for this information.

```
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
UBC  LOAD  X 0.75
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
UBC LOAD  Z 0.75
```

Notes:

The UBC / IBC input can be provided in 2 or more lines using the continuation mark (hyphen) as shown in the following example :

```
DEFINE UBC ACCIDENTAL LOAD
ZONE 3.000 -
I 1.00 RWX 1.100 -
RWZ 1.200 STYP 5.000 NA 1.40 NV 1.50 CT -
1.300 PX 2.100 PZ 2.200
```

5.31.2.2 UBC 1994 or 1985 Load Definition

Purpose

This set of commands may be used to define the parameters for generation of UBC-type equivalent static lateral loads for seismic analysis. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate UBC equation(s).

See Sections 1.17.2, 5.32.12 and Examples manual problem no. 14

$$V = \frac{ZIC}{Rw} W \quad \text{(per UBC 1994)} \qquad (1)$$

$$V = ZIKCSW \quad \text{(per UBC 1985)} \qquad (2)$$

Note:
1) All symbols and notations are per UBC
2) Base shear V may be calculated by STAAD using either the 1994 procedure (equation 1) or the 1985 procedure (equation 2). The user should use the appropriate "ubc-spec" (see General Format below) to instruct the program accordingly.

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. User provides seismic zone co-efficient and desired "ubc-spec" (1985 or 1994) through the DEFINE UBC LOAD command.

2. Program calculates the structure period T.
3. Program calculates C from appropriate UBC equation(s) utilizing T.
4. Program calculates V from appropriate equation(s). W is obtained from the weight data (SELFWEIGHT, JOINT WEIGHT(s), etc.) provided by the user through the DEFINE UBC LOAD command. The weight data must be in the order shown.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per UBC procedures.

General format:

```
DEFINE UBC (ACCIDENTAL) LOAD

        ZONE f1       ubc-spec

        SELFWEIGHT
        JOINT WEIGHT
              joint-list  WEIGHT w
        MEMBER  WEIGHT
                         ( UNI    v1  v2  v3 )
              mem-list   (                   )
                         ( CON    v4  v5     )
        ELEMENT  WEIGHT

              plate-list   PRESS  p1

        FLOOR  WEIGHT

              YRANGE ...
        (see Section 5.32.4 for input description)

                      (  I       f2    )
                      (  RWX     f3    )
                    * (  RWZ     f4    )                       * (  K     f6   )
ubc-spec =            (  S       f5    )  ubc-spec =             (  I     f7   )
for UBC 1994          ( (CT      f9)   )  for UBC 1985           ( (TS    f8)  )
                      ( (PX      f10)  )
                      ( (PZ      f11)  )
```

where,

f_1 = seismic zone coefficient (0.2, 0.3 etc.). Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.075, 0.15, 0.2, 0.3, 0.4, etc.

f_2 = importance factor

f_3 = numerical co-efficient R_w for lateral load in X-direction

f_4 = numerical co-efficient R_w for lateral load in Z-directions

f_5 = site co-efficient for soil characteristics

f_6 = horizontal force factor

f_7 = importance factor

f_8 = site characteristic period (Referred to as Ts in the UBC code). Default value is 0.5.

f_9 = Value of the term C_t which appears in the equation of the period of the structure per Method A. It's default value is 0.035. See note 7 in section 5.31.2.1.

f_{10} = Period of structure (in seconds) in the X- direction.

f_{11} = Period of structure (in seconds) in the Z direction (or Y if SET Z UP is used).

w = joint weight associated with list

UNI = specifies a uniformly distributed load with a value of v_1 starting at a distance of v_2 (from the start of the member) and ending at a distance of v_3 (from the start of the member). If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

CON = specifies a concentrated force with a value of v_4 applied at a distance of v_5 (from the start of the member). If v_5 is omitted, the load is assumed to act at the center of the member.

p = weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (see section 5.32.4 for details of the Floor Load input).

Notes

1) If the option ACCIDENTAL is used, the accidental torsion will be calculated per UBC specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the weight input (SELFWEIGHT, JOINT WEIGHTs, etc.) specified by the user.
2) In "ubc-spec" for 1985 code, specification of TS is optional. If TS is specified, resonance co-efficient S is determined from the building period T and user provided TS using UBC equations. If TS is not specified, the default value of 0.5 is assumed.
3) By providing either PX or PZ or both, you may override the period calculated by STAAD for Method B of the UBC Code. The user defined value will then be used instead of the one recommended by UBC per equation 28.5 of UBC 94. If you do not define PX or PZ, the period for Method B will be calculated by the program per equation 28.5.
4) Some of the items in the output for the UBC analysis are explained below.

   #### CALC / USED PERIOD

   The CALC PERIOD is the period calculated using the Rayleigh method (Method B as per UBC code). For UBC in the x-direction, the USED PERIOD is PX. For the UBC in the z-direction, the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one substituted into the critical equation of the UBC code to calculate the value of C.

5) In the analysis for UBC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.

Example

```
DEFINE UBC LOAD
ZONE 0.2 I 1.0 RWX 9 RWZ 9 S 1.5 CT 0.032
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 2.5
49 TO 64 WEIGHT 1.25
LOAD 1
UBC LOAD X 0.75
SELFWEIGHT Y -1.0
JOINT LOADS
17 TO 48 FY -2.5
```

5.31.2.3 Colombian Seismic Load

Purpose

The purpose of this command is to define and generate static equivalent seismic loads as per Colombian specifications using a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

See Sections 1.17.2, 5.32.12 and Examples manual problem no. 14

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology :

Seismic zone coefficient and parameter values are supplied by the user through the DEFINE COLOMBIAN LOAD command.

Program calculates the natural period of building T utilizing clause 1628.2.2 of UBC 1994.

Design spectral coefficient (Sa) is calculated utilizing T as,

$$
\begin{aligned}
Sa &= Aa\ I\ (1.0 + 5.0\ T) && \text{when, } 0 \le T \le 0.3 \text{ sec} \\
&= 2.5\ Aa\ I && \text{when, } 0.3 < T \le 0.48\ S \text{ in sec} \\
&= 1.2\ Aa\ S\ I / T && \text{when, } 0.48\ S < T \le 2.4\ S \text{ in sec} \\
&= Aa\ I / 2 && \text{when, } 2.4\ S < T
\end{aligned}
$$

where,

Aa = Seismic Risk factor (user input)
S = Soil Site Coefficient (user input)
I = Coefficient of Importance (user input)

Base Shear, Vs is calculated as

$Vs = W * Sa$

Where,

W = Total weight on the structure

Total lateral seismic load, Vs is distributed by the program among different levels as,

$Fx = Cvx * Vs$

Where,

$Cvx = (Wx * hx^K) / \Sigma_{i=1}^{n} (Wx * hx^K)$

Where,

Wx = Weight at the particular level
hx = Height of that particular level
K = 1.0 when, $T \le 0.5$ sec
= 0.75 + 0.5 * T when, $0.5 < T \le 2.5$ sec
= 2.0 when, $2.5 < T$

General Format

DEFINE COLOMBIAN LOAD
ZONE f1 *ubc-spec*
SELFWEIGHT
JOINT WEIGHT
Joint-list **WEIGHT w**

[See Section 5.31.2.2 for complete weight input definition]

ubc-spec **= (I f2, S f3)**

Where, f1, f2 and f3 are Seismic Risk factor, Soil Site Coefficient and Coefficient of Importance.

General format to provide Colombian Seismic load in any load case:

```
LOAD i
COLOMBIAN LOAD    {X/Y/Z}    (f)
```

where i and f are the load case number and factor to multiply horizontal seismic load respectively. Choose horizontal directions only.

Example

```
DEFINE COLOMBIAN LOAD
ZONE 0.38  I  1.0  S  1.5
JOINT  WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720

LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
COLOMBIAN  LOAD  X
```

5.31.2.4 Japanese Seismic Load

Purpose

The purpose of this command is to define and generate static equivalent seismic loads as per Japanese specifications using a static equivalent approach similar to those outlined by UBC. The implementation is as per Article 88 in the 'Building Codes Enforcement Ordinance 2006'.

See Sections 1.17.2, 5.32.12 and Examples manual problem no. 14

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology :

Seismic zone coefficient and parameter values are supplied by the user through the DEFINE AIJ LOAD command.

Program calculates the natural period of building T utilizing the following equation

$$T = h\,(0.02 + 0.01\,\alpha)$$

where,

h = height of building
α = ratio of steel part

Design spectral coefficient (Rt) is calculated utilizing T and Tc as follows

$$R_t = 1.0 \quad \text{when} \quad T < T_c$$
$$= 1 - 0.2\,(T/T_c - 1)^2 \quad \text{when} \quad T_c \le T \le 2T_c$$
$$= 1.6\,T_c / T \quad \text{when} \quad 2T_c \le T$$

αi is calculated from the weight provided by the user in Define AIJ Load command.

Seismic coefficient of floor Ci is calculated using appropriate equations

Ci	=	Z Rt Ai Co

Where,

Z	=	zone factor
Co	=	normal coefficient of shear force
Ai	=	$1 + (1 / \sqrt{\alpha i} - \alpha i)\, 2T / (1 + 3T)$

The total lateral seismic load is distributed by the program among different levels.

General Format

DEFINE AIJ LOAD
ZONE f1 *ubc-spec*
SELFWEIGHT
JOINT WEIGHT
Joint-list **WEIGHT w**

[See Section 5.31.2.2 for complete weight input definition]

ubc-spec **= (I f2, CO f3, TC f4)**

Where, f1, f2, f3and f4 are Zone factor, Ratio of Steel Part, Normal coefficient of shear force and Value needed for calculation of Rt.

General format to provide Japanese Seismic load in any load case:

LOAD i
AIJ LOAD {X/Y/Z} (f)

Where, i and f are load case number and factor to multiply horizontal seismic load respectively. Choose horizontal directions only.

Example

```
DEFINE AIJ LOAD
ZONE 0.8  I  0.0  CO 0.2  TC 0.6
JOINT  WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X)
AIJ  LOAD  X
```

5.31.2.5 Definition of Lateral Seismic Load per Indian IS:1893 (Part 1) – 2002 Code

Purpose

This feature enables one to generate seismic loads per the IS:1893 specifications using a static equivalent approach.

Description

The seismic load generator can be used to generate lateral loads in the X and Z directions only. Y is the direction of gravity loads. This facility has not been developed for cases where the Z axis is set to be the vertical direction using the "SET Z UP" command.

Methodology

The design base shear is computed by STAAD in accordance with the IS: 1893(Part 1)-2002 equation 7.5.3.

$$\mathbf{V} = \mathbf{A_h . W} \qquad \text{where, } \mathbf{A_h} = \frac{Z}{2} . \frac{I}{R} . \frac{Sa}{g}$$

Note:
All symbols and notations in the above equation are as per IS: 1893(Part 1)-2002.

STAAD utilizes the following procedure to generate the lateral seismic loads.

See Section 5.32.12

1. User provides seismic zone co-efficient and desired "1893(Part 1)-2002specs" through the DEFINE 1893 LOAD command.
2. Program calculates the structure period (T).
3. Program calculates Sa/g utilizing T.
4. Program calculates V from the above equation. W is obtained from SELFWEIGHT, JOINT WEIGHT(s) and MEMBER WEIGHT(S) provided by the user through the DEFINE 1893 LOAD command.

5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per the IS: 1893 procedures.

General format:

DEFINE 1893 LOAD

ZONE f1 1893-spec

SELFWEIGHT
JOINT WEIGHT
joint-list WEIGHT w
MEMBER WEIGHT

$$\textbf{mem-list} \quad \left\{ \begin{array}{l} \textbf{UNI} \quad \textbf{v1 v2 v3} \\ \\ \textbf{CON} \quad \textbf{v4 v5} \end{array} \right\}$$

CHECK SOFT STORY

1893-Spec= {RF f2, I f3, SS f4, (ST f5), DM f6, (PX f7), (PZ f8), (DT f9)}

where,

Zone f1 = Seismic zone coefficient . Refer to Table 2 of IS:1893(Part 1)-2002.

RF f_2 = Response reduction factor. Refer Table 7 of IS: 1893 (Part 1) -2002.

I f_3 = Importance factor depending upon the functional use of the structures, characterized by hazardous consequences of its failure, post-earthquake functional needs, historical value, or economic importance. Refer Table 6 of IS: 1893(Part 1)-2002.

SS f_4 = Rock or soil sites factor (=1 for hard soil, 2 for medium soil, 3 for soft soil). Depending on type of soil, average response acceleration coefficient S_a/g is calculated corresponding to 5% damping. Refer Clause 6.4.5 of IS: 1893 (Part 1) -2002.

ST f_5 = Optional value for type of structure (=1 for RC frame building, 2 for Steel frame building, 3 for all other buildings). If this parameter is mentioned the program will calculate natural period as per Clause 7.6 of IS:1893(Part 1)-2002.

DM f6 = Damping ratio to obtain multiplying factor for calculating Sa/g for different damping. If no damping is specified 5% damping (default value 0.05) will be considered corresponding to which multiplying factor is 1.0. Refer Table 3 of IS:1893(Part 1)-2002.

PX f_7 = Optional period of structure (in sec) in X direction. If this is defined this value will be used to calculate S_a/g for generation of seismic load along X direction.

PZ f8 = Optional period of structure (in sec) in Z direction. If this is defined this value will be used to calculate Sa/g for generation of seismic load along Z direction.

DT f9 = Depth of foundation below ground level. It should be defined in current unit. If the depth of foundation is 30m or more, the value of A_h is taken as half the value obtained. If the foundation is placed between the ground level and 30m depth, this value is linearly interpolated between A_h and $0.5A_h$.

w = joint weight associated with joint list

<u>UNI</u> .= specifies a uniformly distributed load with a value of v_1 starting at a distance of v_2 (from the start of the member) and ending at a distance of v_3 (from the start of the member). If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

<u>CON</u> = specifies a concentrated force with a value of v4 applied at a distance of v5 (from the start of the member). If v5 is omitted, the load is assumed to act at the center of the member.

CHECK SOFT STORY indicates that soft story checking will be performed. If omitted from input, there will be no soft story checking. See below for details on soft storey checking.

Note: By default STAAD calculates natural periods of the structure in both X and Z directions respectively which are used in calculation for base shear. If, however, PX and PZ are mentioned the program will consider these values for calculation of average response acceleration coefficient. If instead of PX and PZ values ST is mentioned the program will calculate natural period depending upon the empirical expression given in IS: 1893 (Part 1)-2002.

Soft Storey Checking

As per the IS1893-2002 code Clause 7.1, to perform well during an earthquake a building must have simple and regular configuration, adequate lateral strength, stiffness and ductility. This is because a building with simple regular geometry and uniformly distributed mass and stiffness in plan as well as in elevation, will suffer much less damage than buildings with irregular configurations.

According to this standard, a building can be considered irregular, if at least one of the conditions given in Table 4 - Plan Irregularities and Table 5 - Vertical Irregularities, of IS1893-2002 is applicable.

STAAD has implemented the methodology to find vertical stiffness irregularities, as given in Table 5 Sl No. (1) i) a) and Sl No. (1) i) b), in the form of soft storey checking.

Stiffness Irregularities – Soft Storey

As per this provision of the code, a soft storey is one in which the lateral stiffness is less than 70 percent of that in the storey above or less than 80 percent of the average lateral stiffness of the three storey above.

<u>Stiffness Irregularities – Extreme Soft Storey</u>

As per this provision of the code, a extreme soft storey is one in which the lateral stiffness is less than 60 percent of that in the storey above or less than 70 percent of the average lateral stiffness of the three storey above.

Thus, if any storey of a building is found to be soft or extremely soft, the building is likely to suffer much damage in an earthquake than a similar type of building but has more regular vertical stiffness.

Note:

STAAD identifies column and shear wall (without opening) as vertical component for the purpose of computing lateral stiffness of the storey. The vertical stiffness of a column is calculated as $12EI / L^3$ where E is the Young's modulus, I is the moment of inertia and L is the length of the column respectively and that for a shear wall (without opening) is calculated as $Ph^3/3EI + 1.2Ph/AG$ (i.e. summation of flexural stiffness and shear stiffness, obtained as deflection of a cantilever wall under a single lateral load P at its top) where h is the height, A is the cross-sectional area and G is the shear modulus of the wall (E and I carry usual meaning). The summation of lateral stiffness of all columns and shear walls at a particular floor level constitute the total lateral stiffness of that particular storey or floor level. The program checks soft storey of a building along both global X and Z directions respectively.

Identification of floor level

Following two ways can identify floor level.

1. Program calculated
2. User defined

Program calculated

In general, STAAD identifies floor levels in order of increasing magnitude of Y-coordinates of joints. The program sorts different values of Y-coordinates, from minimum to maximum values, in ascending order and consider each Y-coordinate value as each floor level. This is the method used by the DEFINE UBC or similar load generation features.

This feature has been enhanced to identify the beam-column junctions at each floor level, as identified by the method above. If no beam-column junctions are identified at that level, a floor level will not be considered at that level in the structure. Where beam-column junctions are found, the program identifies two beams, at the same level which span in two different directions from the same beam-column junction. If this true, this identified floor level will be considered as truly existing floor level in the structure.

User defined
Floor heights should be defined before using the DEFINE 1893 or any primary response spectrum load case.

General format:

FLOOR HEIGHT

$\mathbf{h_1; h_2; h_3; \ldots\ldots\ldots\ldots\ldots; h_i - h_{i+1}; \ldots\ldots\ldots\ldots\ldots; h_n}$

where,

$\mathbf{h_1}$ **......** $\mathbf{h_n}$ are the different floor heights in current length unit and **n** is the number of floor levels.

For regular building which has well defined floors (i.e. does not contain shear wall, staggered flooring, etc) STAAD can identify floor level on its own. However, if floor level is not so well defined it is better to define floor height to have more accurate result for the purpose of torsion and soft storey checking.

Example

See section 5.32.12 for an example of the correct usage of this command.

5.31.2.6 IBC 2000/2003 Load Definition

Description

See Sections 1.17.2, 5.32.12 and Examples manual problem no. 14

The specifications of the IBC 2000 and 2003 codes for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The implementation details of the respective codes are as follows:

IBC 2000

On a broad basis, the rules described in section 1617.4 of the IBC 2000 code document have been implemented. These are described in pages 359 thru 362 of that document. The specific section numbers, those which are implemented, and those which are not implemented, are as follows:

Implemented sections of IBC 2000	Omitted sections of IBC 2000
1617.4.1	1617.4.4.1
1617.4.1.1	1617.4.4.2
1617.4.2	1617.4.4.3
1617.4.2.1	1617.4.4.5
1617.4.3	1617.4.5
1617.4.4	1617.4.6
1617.4.4.4	

IBC 2003

On a broad basis, the rules described in section 1617.4 of the IBC 2003 code document have been implemented. This section directs the engineer to Section 9.5.5 of the ASCE 7 code. The specific section numbers of ASCE 7-2002, those which are implemented, and those which are not implemented, are shown in the table below. The associated pages of the ASCE 7-2002 code are 146 thru 149.

Implemented sections of IBC 2003 (ASCE 7-02)	Omitted sections of IBC 2003 (ASCE 7-02)
9.5.5.2	9.5.5.5.1
9.5.5.2.1	9.5.5.5.2
9.5.5.3	9.5.5.6
9.5.5.3.1	9.5.5.7
9.5.5.3.2	
9.5.5.4	
9.5.5.5	
Some parts of 9.5.5.5.2	

Methodology

The design base shear is computed in accordance with the equation shown below.

$V = C_S W$ .... Eqn 16-34 of IBC 2000
Eqn 9.5.5.2-1 of ASCE 7-02

The seismic response coefficient, C_s, is determined in accordance with the following equation:

$$C_s = \frac{S_{DS}}{\left[\frac{R}{I_E}\right]} \text{ .... Eqn 16-35 of IBC 2000, Eqn 9.5.5.2.1-1 of ASCE 7-02}$$

C_s need not exceed the following:

$$C_s = \frac{S_{D1}}{\left[\frac{R}{I_E}\right]T} \text{ .... Eqn 16-36 of IBC 2000, Eqn 9.5.5.2.1-2 of ASCE 7-02}$$

C_s shall not be taken less than:

$$C_S = 0.044 S_{DS} I_E \text{ ....Eqn 16-37 of IBC 2000, Eqn 9.5.5.2.1-3 of ASCE 7-02}$$

In addition, for structures for which the 1-second spectral response, S_1, is equal to or greater than 0.6g, the value of the seismic response coefficient, C_s, shall not be taken less than:

$$C_s = \frac{0.5 S_1}{\left[R/I_E\right]} \text{ ....Eqn 16-38 of IBC 2000, Eqn 9.5.5.2.1-4 of ASCE 7-02}$$

For an explanation of the terms used in the above equations, please refer to the IBC 2000 and ASCE 7-02 codes.

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE IBC 2000 or 2003 LOAD command.

General Format

DEFINE IBC ({ 2000 | 2003 }) (ACCIDENTAL) LOAD
SDS f1 *ubc-spec*
SELFWEIGHT
JOINT WEIGHT
***Joint-list* WEIGHT w**

[See Section 5.31.2.2 for complete weight input definition]

***ubc-spec* = { SD1 f2 S1 f3 IE f4 RX f5 RZ f6**
SCLASS f7 (CT f8) (PX f9) (PZ f10) }

where,

f1 = Design spectral response acceleration at short periods. See equation 16-18, Section 1615.1.3 of IBC 2000 and equation 9.4.1.2.5-1 of ASCE7-02

f2 = Design spectral response acceleration at 1-second period. See equation 16-19, Section 1615.1.3. of IBC 2000 and equation 9.4.1.2.5-2 of ASCE7-02

f3 = Mapped spectral acceleration for a 1-second period. See equation 16-17 of IBC 2000, and 9.4.1.2.4-2 of ASCE 7-02

f4 = Occupancy importance factor determined in accordance with Section 1616.2 of IBC 2000 and 2003, and section 9.1.4 (page 96) of ASCE 7-02

f5 = The response modification factor for lateral load along the X direction. See Table 1617.6 of IBC 2000 (pages 365-368) and Table 1617.6.2 of IBC 2003 (page 334-337). It is used in equations 16-35, 16-36 & 16-38 of IBC 2000

f6 = The response modification factor for lateral load along the Z direction. See Table 1617.6 of IBC 2000 (pages 365-368) and Table 1617.6.2 of IBC 2003 (page 334-337). It is used in equations 16-35, 16-36 & 16-38 of IBC 2000.

f7 = Site class as defined in Section 1615.1.1 of IBC 2000 (page 350) & 2003 (page 322). Enter 1 through 6 in place of A through F, see table below).

f8 = Optional CT value to calculate time period. See section 1617.4.2.1, equation 16-39 of IBC 2000 and section 9.5.5.3.2, equation 9.5.5.3.2-1 of ASCE 7-02.

If the value of Ct is not specified, the program scans the Modulus of Elasticity (E) values of all members and plates to determine if the structure is made of steel, concrete or any other material. If the average E is smaller than 2000 ksi, Ct is set to 0.016(0.044) [0.016 is the value in FPS units, 0.044 is the value in Metric units]. If the average E is between 2000 & 10000 ksi, Ct is set to 0.02(0.055). If the average E is greater than 10000 ksi, Ct is set to 0.028(0.068). If the building material cannot be determined, Ct is set to 0.02(0.055).

In Table 9.5.5.3.2 on page 147 of ASCE 7-02, a structure type called "Eccentrically braced steel frames" is listed, with a Ct of 0.03(0.07). STAAD does not choose this Ct on its own presently. If one wishes to use this value, Ct should be specified as an input accordingly.

f9 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

f10 = Optional Period of structure (in sec) in Z or Y direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

The Soil Profile Type parameter SCLASS can take on values from 1 to 6. These are related to the values shown in Table 1615.1.1, Site Class Definitions of the IBC 2000/2003 code in the following manner :

STAAD Value	IBC code value
1	A
2	B
3	C
4	D
5	E
6	F

Example

```
DEFINE IBC 2003 LOAD
SDS 0.6 SD1 .36 S1 .3 I 1.0 RX 3 RZ 4 SCL 4 CT 0.032
SELFWEIGHT
JOINT  WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
```

Steps to calculate base shear are as follows:

1. Time Period of the structure is calculated based on section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02. This is reported in the output as Ta.
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as T.
3. The user may override the Rayleigh based period by specifying a value for PX or PZ (Items f7 and f8) depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above-mentioned two periods, and the additional guidance provided in clause 1617.4.2 of IBC 2000, and section

9.5.5.3 of ASCE 7-02. The resulting value is reported as "Time Period used".

5. The Design Base Shear is calculated based on equation 16-34 of IBC 2000 and equation 9.5.5.2-1 of ASCE 7-02. It is then distributed at each floor using the rules of clause 1617.4.3, equations 16-41 and 16-42 of IBC 2000, and, clause 9.5.5.4, equations 9.5.5.4-1 & 9.5.5.4-2 of ASCE 7-02.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (clause 1617.4.4.4 of IBC 2000 and, 9.5.5.5.2 of ASCE 7-02). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.

The following example shows the commands required to enable the program to generate the lateral loads. Users may refer to Section 5.32.12 of the Technical Reference Manual for this information.

Example

```
LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
IBC  LOAD  X 0.75
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
IBC LOAD  Z 0.75
```

The Examples manual contains examples illustrating load generation involving IBC and UBC load types.

5.31.2.7 CFE (Comision Federal De Electricidad) Seismic Load

Purpose

The purpose of this command is to define and generate static equivalent seismic loads as per MANUAL DE DISEÑO POR SISMO - SEISMIC DESIGN HANDBOOK COMISION FEDERAL DE ELECTRICIDAD - ELECTRIC POWER FEDERAL COMISSION - October 1993 (Chapters 3.1, 3.2, 3.3 and 3.4) specifications. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s). This is a code used in the country of Mexico.

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

Seismic zone coefficient and parameter values are supplied by the user through the DEFINE CFE LOAD command.

Program calculates the natural period of building T utilizing Raleigh-Quotient method. If time period is provided in the input file, that is used in stead of calculated period.

c = Seismic coefficient is extracted from table 3.1
a_0, T_a, T_b, and r are obtained form table 3.1

The acceleration a is calculated according to the following:

$$a = a_0 + (c - a_0) \times T/T_a \qquad \text{if} \qquad T < T_a$$
$$a = c \qquad \text{if} \qquad T_a \le T \le T_b$$

$a = c\,(Tb/T)^r$ if $T > T_b$

The ductility reduction factor Q' is calculated according to section 3.2.5.

$Q' = Q$ if $T \geq T_a$
$Q' = 1 + (T/T_a)\,(Q-1)$ if $T < T_s$

If not regular Q' = Q' x 0.8

If the period T_s of the soil is known and the soil type II or III T_a and T_b will be modified according to section 3.3.2.

Lateral loads for each direction are calculated for:

$T \leq Tb$ – Eq. 4.5. Section 3.4.4.2

$$P_n = W_n h_n \frac{\sum_{n=1}^{N}(W_n)\ a}{\sum_{n=1}^{N}(W_n h_n)\ Q'}$$

$T > Tb$ – Eq. 4.6/7/8. Section 3.4.4.2

$$P_n = W_n\, a / Q\, (K_1 h_i + K_2 hi^2)$$

Being:

$K_1 = q\ \{1 - r\,(1-q)\}\ \Sigma W_i / \Sigma (W_i/h_i)$
$K_2 = 1.5\ r\ q\ (1-q)\ \Sigma W_i / \Sigma (W_i/h_i^2)$
$q = (Tb/T)^r$

The base shear are distributed proportionally to the height if T≤ Tb or with the quadratic equation mentioned if T > Tb.

The distributed base shears are subsequently applied as lateral loads on the structure.

General Format

DEFINE CFE LOAD
ZONE f1 *cfe-spec*
SELFWEIGHT
JOINT WEIGHT
Joint-list **WEIGHT w**

[See Section 5.31.2.2 for complete weight input definition]

cfe-spec = {
QX f2
QZ f3
GROUP f4
STYP f5
(REGULAR)
(TS f6)
(PX f7)
(PZ f8)
}

where,

f1 = Zone number specified in number such as 1, 2, 3 or 4
f2 = seismic behavior factor of the structure along X direction as a parameter according 3.2.4.
f3 = seismic behavior factor of the structure along Z direction as a parameter according 3.2.4.
f4 = Group of structure entered as A or B
f5 = Soil type entered as 1 or 2 or 3
REGULAR optional parameter is entered to consider the structure as a regular structure. By default, all structures are considered as irregular.
f6 = site characteristic period

f7 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value calculated by the program using Raleigh-Quotient method

f8 = Optional Period of structure (in sec) in Z direction (or Y if SET Z UP is used) to be used as fundamental period of the structure instead of the value calculated by the program using Raleigh-Quotient method

General format to provide RPA Seismic load in any load case:

LOAD i
CFE LOAD {X/Y/Z} (f)

where i and f are the load case number and factor to multiply horizontal seismic load respectively. Choose horizontal directions only.

Examples

```
UNIT KGS METER
DEFINE CFE LOAD
ZONE 2 QX .5 QZ 0.9 STYP 2 GROUP B TS 0.2
SELFWEIGHT
MEMBER WEIGHT
1 TO 36 41 TO 50 UNI 300
JOINT  WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
FLOOR WEIGHT
YRA 11.8 12.2 FLOAD 400 -
XRA -1 11 ZRA -1 21

LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
CFE  LOAD  X 1.0
LOAD 2 ( SEISMIC LOAD IN -Z DIRECTION )
CFE  LOAD  Z -1.0
```

5.31.2.8 NTC (Normas Técnicas Complementarias) Seismic Load

Purpose

The purpose of this command is to define and generate static equivalent seismic loads as per Code of the México Federal District (Reglamento de Construcciones del Distrito Federal de México) and Complementary Technical Standards for Seismic Design (y Normas Técnicas Complementarias (NTC) para Diseño por Sismo -Nov. 1987) (Chapters 8.1 8.2 8.6 and 8.8) specifications. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

The design base shear is computed in accordance with Sections 8.1 or 8.2 of the NTC as decided by the user.

A. Base Shear is given as

Vo / Wo = c / Q

c: obtained by the program from the following table

Seismic Coeficient c	GROUP A	GROUP B
Z-I	0.24	0.16
Z-II not shadowed	0.48	0.32
ZIII y Z_II shadowed	0.60	0.40

Q: is entered by the user as a parameter.

B. Base shear is given as

$$Vo / Wo = a / Q'$$

Where Reduction of Shear Forces are requested

Time Period T of the structure is:
calculated by the program based on using Raleigh quotient technique.

The user may override the period that the program calculates by specifying these in the input

a and Q' are calculated according to the sections 3 and 4 of the NTC, that is to say:

$a = (1+3T/Ta)c/4$ if $T<Ta$

$a=c$ if $Ta \leq T \leq Tb$

$a=q*c$ if $T>Tb$

where,

$q= (Tb/T)^r$

$Q'= Q$ if $T \geq Ta$

$Q'= 1 + (T/Ta) (Q-1)$ if $T < Ta$

If not regular

$Q' = Q' \times 0.8$

Ta, Tb and r are taken from table 3.1.

Table 3.1 Values of Ta, Tb and r			
ZONE	Ta	Tb	r
I	0.2	0.6	1/2
II not shaded	0.3	1.5	2/3
III y II shaded	0.6	3.9	1.0

a shall not be less than c/4

Vo for each direction is calculated

$Vo = Wo\ a/Q'$ if $T \leq Tb$

$Vo = \Sigma Wi\ a/Q'\ (K1\ hi + K2\ hi^2)$ if $T > Tb$

where

$K1 = q\ (1 - r\ (1-q))\ \Sigma\ Wi/(\Sigma\ Wi/hi)$

$K2 = 1.5\ r\ q\ (1-q)\ \Sigma\ Wi/(\Sigma\ Wi/hi^2)$

Wi and hi the weight and the height of the i th mass over the soil or embedment level.

The base shear are distributed proportionally to the height if $T \leq Tb$ or with the cuadratic equation mentioned if $T > Tb$.
The distributed base shears are subsequently applied as lateral loads on the structure.

General Format

DEFINE NTC LOAD
ZONE f1 *ntc-spec*
SELFWEIGHT
JOINT WEIGHT
Joint-list **WEIGHT w**

[See Section 5.31.2.2 for complete weight input definition]

$$
ntc\text{-}spec = \left\{ \begin{array}{ll} \text{QX} & \text{f2} \\ \text{QZ} & \text{f3} \\ \text{GROUP} & \text{f4} \\ \text{(SHADOWED)} & \\ \text{(REGULAR)} & \\ \text{(REDUCE)} & \\ \text{(PX} & \text{f6)} \\ \text{(PZ} & \text{f7)} \end{array} \right\}
$$

where,

f1 = Zone number specified in number such as 1, 2, 3 or 4
f2 = seismic behavior factor of the structure along X direction as a parameter according 3.2.4.
f3 = seismic behavior factor of the structure along Z direction as a parameter according 3.2.4.
f4 = Group of structure entered as A or B
REGULAR optional parameter is entered to consider the structure as a regular structure. By default, all structures are considered as irregular.
SHADOWED optional parameter is used to define the shaded zone II as the site of the structure. By default regular zone II is used.
REDUCE optional parameter allows to reduce the seismic factors as described above. Otherwise the following formula is used to calculate base shear,

$$V = \frac{c}{Q'} \sum_{n=1}^{N} W_n$$

f6 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value calculated by the program using Raleigh-Quotient method

f7 = Optional Period of structure (in sec) in Z (or Y if Z Up) direction to be used as fundamental period of the structure instead of the value calculated by the program using Raleigh-Quotient method

General format to provide NTC Seismic load in any load case:

LOAD i
NTC LOAD {X/Y/Z} (f)

where i and f are the load case number and factor to multiply horizontal seismic load respectively. Choose horizontal directions only.

Examples

```
UNIT KGS METER
DEFINE NTC LOAD
ZONE 2 QX .5 QZ 0.9 GROUP B
SELFWEIGHT
ELEMENT  WEIGHT
1577 TO 1619 PRESSURE 275

LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
NTC  LOAD  X 1.0
LOAD 2 ( SEISMIC LOAD IN Z DIRECTION )
NTC  LOAD  Z 1.0
```

5.31.2.9 RPA (Algerian) Seismic Load

Purpose

The purpose of this command is to define and generate static equivalent seismic loads as per RPA specifications using a static equivalent approach similar to those outlined by RPA. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

Methodology

The design base shear is computed in accordance with section 4.2.3 of the RPA 99 code. The primary equation, namely 4-1, as shown below, is checked.

$$V = \frac{A.D.Q}{R} W \text{ ....Eqn. 4.1 of RPA 99}$$

where

W = total weight on the structure
A = zone coefficient
D = average dynamic amplification factor
R = lateral R factor
Q = structural quality factor

Seismic zone coefficient and parameter values are supplied by the user through the DEFINE RPA LOAD command.

Program calculates the natural period of building T utilizing clause 4.2.4 of RPA 99.

Design spectral coefficient (D) is calculated utilizing T as,

$$
\begin{aligned}
D &= 2.5\,\eta \text{ when, } 0 \le T \le T_2 \\
&= 2.5\,\eta\,.\,(T_2/T)^{2/3} \text{ when, } T_2 < T \le 3.0\,\text{sec} \\
&= 2.5\,\eta\,.\,(T_2/3)^{2/3}\,.\,(3/T)^{5/3} \text{ when, } T > 3.0\,\text{sec}
\end{aligned}
$$

where

η = factor of damping adjustment .... Eq. 4.3

T_2 = specific period ....Table 4.7

Total lateral seismic load, V is distributed by the program among different levels.

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE RPA LOAD command.

General Format

DEFINE RPA LOAD
A f1 *rpa-spec*
SELFWEIGHT
JOINT WEIGHT
Joint-list **WEIGHT w**

(See section 5.31.2.2 for complete weight input definition) *rpa-spec* =

Q	**f2**
RX	**f3**
RZ	**f4**
STYP	**f5**
CT	**f6**
CRDAMP	**f7**
(PX	**f8)**
(PZ	**f9)**

where

f1 = Seismic zone coefficient. Instead of using an integer value like 1, 2, 3 or 4, use the fractional value like 0.08, 0.15, 0.2, 0.3, 0.05, etc.
f2 = Importance factor
f3 = Coefficient R for lateral load in X direction – table 4.3
f4 = Coefficient R for lateral load in Z direction – table 4.3
f5 = Soil Profile Type
f6 = Coefficient from table 4.6 of RPA 99
f7 = Damping factor
f8 = Optional Period of structure (in sec) in X direction
f9 = Optional Period of structure (in sec) in Z direction (or Y if SET Z UP is used) to be used as fundamental period of the structure instead of the value calculated by the program using Raleigh method

General format to provide RPA Seismic load in any load case:

LOAD i
RPA LOAD {X/Y/Z} (f)

where, I and f are the load case number and factor to multiply horizontal seismic load respectively. Choose horizontal directions only.

Example

```
DEFINE RPA LOAD
A 0.15 Q 1.36 STYP 2 RX 3 RZ 4 CT 0.0032 –
CRDAMP 30 PX .027 PZ 0.025
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720

LOAD 1 ( SEISMIC LOAD IN X DIRECTION )
RPA LOAD X 1.0
```

5.31.2.10 Canadian Seismic Code (NRC) - 1995

Purpose

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per National Building Code(NRC/CNRC) of Canada- 1995 edition. Depending on this definition, equivalent lateral loads will be generated in horizontal direction(s).

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The minimum lateral seismic force or base shear (V) is automatically calculated by STAAD using the appropriate equation(s).

See Sections 1.17.2, 5.32.12 and Example Problem Examp14_NRC.std

$$\mathbf{V} = \frac{0.6 \times V_e}{R}$$ (per sentence 4, section 4.1.9.1)

Where V_e , the equivalent lateral seismic force representing elastic response is given by

$$\mathbf{V_e} = v \times S \times I \times F \times W$$ (per sentence 5,section 4.1.9.1)

Where,

v = Zonal velocity ratio per appendix C
S = Seismic Response Factor per table 4.9.1.A
I = Seismic importance factor per sentence 10 section 4.1.9.1
F = Foundation factor conforming to Table 4.9.1.C and sentence 11 section 4.1.9.1
W = Total load lumped as weight per sentence 2 section 4.1.9.1

R = Force modification factor conforming to Table 4.9.1.B that reflects the capability of a structure to dissipate energy through inelastic behaviour.

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. User provides seismic zone co-efficient and desired "nrc-spec" (1995) through the DEFINE NRC LOAD command.
2. Program calculates the fundamental period(T) of the structure by

   (a) finding out whether the structure being analysed is a moment resisting frame made primarily of steel or of concrete or it is a structure of any other type. Alternatively, the software uses the optional parameter CT if provided. The calculation is done per sentence 7(a) & 7(b) of section 4.1.9.1.
   (b) using the Rayleigh method or using the optional parameters PX , PZ –if provided. The stipulations of sentence 7(c) of section 4.1.9.1 are also considered while calculating.
   (c) taking the conservative value of T between those calculated by methods (a) and (b) above.
3. Program finds out the value of Seismic Response Factor(S) per table 4.9.1.A utilizing the values of T as calculated above and the values of ZA & ZV input by the user.
4. Program calculates V per sentence 4 section 4.1.9.1. W is obtained from the weight data (SELFWEIGHT, JOINT WEIGHT(s), etc.) provided by the user through the DEFINE NRC LOAD command. The weight data must be in the order shown.

a) The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per applicable NRC guidelines like sentence 13(a) section 4.1.9.1.

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE NRC LOAD command.

General format:

<u>DEF</u>INE <u>NRC</u> <u>LOA</u>D

***nrc-spec**

<u>SEL</u>FWEIGHT
<u>JOI</u>NT <u>WEI</u>GHT
joint-list <u>WEI</u>GHT w
<u>MEM</u>BER <u>WEI</u>GHT

$$\text{mem-list} \left\{ \begin{array}{ll} \underline{\text{UNI}} & v_1 \; v_2 \; v_3 \\ \underline{\text{CON}} & v_4 \; v_5 \end{array} \right\}$$

<u>ELE</u>MENT <u>WEI</u>GHT

plate-list PRESS p_1

<u>FLO</u>OR <u>WEI</u>GHT

<u>YR</u>ANGE ...
(see Section 5.32.4 for input description)

where

***nrc-spec =**

$$* \left| \begin{array}{ll} \underline{\text{v}} & f_1 \\ \underline{\text{ZA}} & f_2 \\ \underline{\text{ZV}} & f_3 \\ \underline{\text{RX}} & f_4 \\ \underline{\text{RZ}} & f_5 \\ \underline{\text{I}} & f_6 \\ \underline{\text{F}} & f_7 \\ (\underline{\text{CT}} & f_8) \\ (\underline{\text{PX}} & f_9) \\ (\underline{\text{PZ}} & f_{10}) \end{array} \right|$$

where,

f_1 = Zonal velocity ratio per Appendix C
f_2 = Factor for acceleration related seismic zone per Appendix C
f_3 = Factor for velocity related seismic zone per Appendix C
f_4 = Force modification factor along X-direction that reflects the capability of a structure to dissipate energy through inelastic behaviour. Please refer Table 4.1.9.1B
f_5 = Force modification factor along Z-direction that reflects the capability of a structure to dissipate energy through inelastic behaviour. Please refer Table 4.1.9.1B
f_6 = Seismic importance factor per sentence 10 section 4.1.9.1
f_7 = Foundation factor conforming to Table 4.1.9.1C and sentence 11 section 4.1.9.1
f_8 = Factor to be used to calculate the fundamental period of structure .This is an optional parameter.
f_{10} = Period of structure (in seconds) in the X- direction. This is an optional parameter.
f_{11} = Period of structure (in seconds) in the Z- direction (or Y if SET Z UP is used). This is an optional parameter.

w = joint weight associated with list

UNI = specifies a uniformly distributed load with a value of v_1 starting at a distance of v_2 (from the start of the member) and ending at a distance of v_3 (from the start of the member). If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

CON = specifies a concentrated force with a value of v_4 applied at a distance of v_5 (from the start of the member). If v_5 is omitted, the load is assumed to act at the center of the member.
p = weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (see section 5.32.4 of STAAD Technical Reference Manual for details of the Floor Load input).

The weights have to be input in the order shown.

Generation of NRC Load

The load so defined as above is applied on the structure in the NRC loadcases. These loadcases have to be the first loadcases in the input file. Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per the relevant code specifications.

The following general format should be used to generate loads in a particular direction.

LOAD i
NRC LOAD {Dir}(f1)

Where Dir = The direction in which the NRC load is generated—X,Y or Z.
i = load case number
f1 = factor to be used to multiply the NRC Load (default = 1.0). May be negative.

Notes

1) By providing either PX or PZ or both, you may override the period calculated by STAAD using Rayleigh method. If you do not define PX or PZ, the period for Method 2(b) above will be calculated by the program using Rayleigh method and the stipulations of sentence 7(c) of section 4.1.9.1
2) Some of the items in the output for the NRC analysis are explained below.

   T_a = Time period calculated per sentence 7(a) or 7(b) of section 4.1.9.1

T_c = Time period calculated per sentence 7(c) of section 4.1.9.1

CALC / USED PERIOD

The CALC PERIOD is the period calculated using the Rayleigh method. For NRC in the x-direction, the USED PERIOD is PX. For the NRC in the z-direction (or Y direction if SET Z UP is used), the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one utilized to find out the value of S.

3) In the analysis for NRC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.

Example

```
DEFINE NRC LOAD
V 0.2 ZA 4 ZV 4 RX 4 RZ 4 I 1.3 F 1.3 CT 0.35 PX 2 PZ 2
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5
LOAD 1 EARTHQUAKE ALONG X
NRC LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

5.31.2.11 Canadian Seismic Code (NRC) – 2005 Volume I

Purpose

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per the National Building Code (NRC/CNRC) of Canada- 2005 Volume 1 Section 4.1.8. Equivalent lateral loads will be generated in the horizontal direction(s).

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

See Example Problem Examp14_NRC_2005.std

The minimum lateral seismic force or base shear (V) is automatically calculated by STAAD using the appropriate equation(s).

(A) **V =** $S(Ta)M_v I_E W/(R_d R_o)$ (as per section 4.1.8.11(2) of NBC of Canada 2005 Volume 1)

except that V shall not be less than,

$S(2.0)M_V I_E W/(R_d R_o)$ (i.e. lower limit of V)

and for an R_d = >1.5,

V need not be greater than

$$\frac{2}{3}S(0.2)I_E W/(R_d R_o) \qquad \text{(i.e. upper limit of V)}$$

T_a is the fundamental lateral period in the direction under consideration and is determined as:

(a) for moment-resisting frames that resist 100% of the required lateral forces and where the frame is not enclosed by or adjoined by more rigid elements that would tend to prevent the frame from resisting lateral forces, and is calculated by the empirical formulae as described below provided h_n is in meter:

i. $0.085(h_n)^{3/4}$ for steel moment frames,

ii. $0.075(h_n)^{3/4}$ for concrete moment frames, or

(b) other established methods of mechanics using a structural model that complies with the requirements of Sentence 4.1.8.3.(8) , except that

(i) for moment resisting frames, T_a shall not be greater than 1.5 times that determined in Clause (a).

(ii) for braced frames, T_a shall not be greater than 2.0 times that determined in Clause (b).

(iii) for shear wall structures, T_a shall not be greater than 2.0 times that determined in Clause (c), and

(iv) for the purpose of calculating the deflections, the period without the upper limit specified is referred from Appendix A.

$S(T_a)$ is the design spectral acceleration and is determined as follows, using linear interpolation for intermediate values of T_a:

$$\begin{aligned} S(T_a) &= F_a S_a(0.2) \text{ for } T_a <= 0.2 \text{ s} \\ &= F_v S_a(0.5) \text{ or } F_a S_a(0.2)\text{, whichever is smaller for } T_a = 0.5 \text{ s} \\ &= F_v S_a(1.0) \text{ for } T_a = 1.0 \text{ s} \\ &= F_v S_a(2.0) \text{ for } T_a = 2.0 \text{ s} \\ &= F_v S_a(2.0)/2 \text{ for } T_a >= 4.0 \text{ s} \end{aligned}$$

The above terms $S_a(0.2)$, $S_a(0.5)$, $S_a(1.0)$ and $S_a(2.0)$ are the Seismic Data and are obtained as user input from the *Table C-2.*

Based on the above values of $S_a(T_a)$, F_a and F_v, the acceleration and velocity based site coefficients are determined from the Tables 4.1.8.4.B and 4.1.8.4.C, using linear interpolation for intermediate values of $S_a(0.2)$ and $S_a(1.0)$. These are the user inputs based on the site classes from A to E and the desired $S_a(0.2)$ and $S_a(1.0)$ values required as per the above equations.

M_v is the factor to account for higher mode effect on base shear and the associated base overturning moment reduction factor is J obtained as user input from Table 4.1.8.11. To get this higher mode factor (M_v) and numerical reduction coefficient for base overturning moment (J), user has to get the ratios of $S_a(0.2)/S_a(2.0)$ and the “Type of Lateral Resisting System”.

For values of M_v between fundamental lateral periods, T_a of 1.0 and 2.0 s, the product $S(T_a)$. M_v shall be obtained by linear interpolation.

Values of J between fundamental lateral periods, T_a of 0.5 and 2.0 s shall be obtained by linear interpolation.

I_E is the earthquake importance factor of the structure and is determined from the Table 4.1.8.5. This is an user input depending on Importance Category and ULS / SLS

W is the weight of the building and shall be calculated internally using the following formula:

$$W = \sum_{i=1}^{n} W_i$$

Where W_i is the portion of W that is located at or assigned to level i.

R_d is the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9.

R_o is the over-strength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9.

These R_d and R_o values are the user inputs depending on the type of SFRS

(B) As per 4.1.8.11(6), the total lateral seismic force, V, shall be distributed such that a portion F_t shall be concentrated at the top of the building, where,

$F_t = 0.07 T_a V$ but F_t is not greater than 0.25V and F_t = 0 when T_a is not greater than 0.7 s.

The remainder (V- F_t), shall be distributed along the height of the building, including the top level, in accordance with the following formula:

$$F_x = (V - F_t) W_x h_x / (\sum_{i=1}^{n} W_i h_i)$$ (as per section 4.1.8.11(6))

Where,

F_x is the lateral force applied to level x.

F_t is the portion of V to be concentrated at the top of the structure.

W_i,W_x are the portion of W that is located at or assigned to level i or x respectively.

h_i,h_x are the height above the base(i=0) to level i or x respectively.

level i is any level in the building, i=1 for first level above the base.

level n is uppermost in the main portion of the structure.

STAAD utilizes the following format to generate the lateral seismic loads.

General format:

```
DEFINE NRC 2005 LOAD
            *nrc-spec
             SELFWEIGT
             JOINT WEIGHT
                    Joint-list  WEIGHT w
            MEMBER  WEIGHT
                    mem-list     {UNI    v1 v2 v3 }
                                 {CON    v4 v5    }
            ELEMENT WEIGHT
                    Plate-list   PRESS p1
            FLOOR WEIGHT
                    YRANGE…….
```

(see Section 5.32.4 for input description)

where,

***nrc-spec =**

[SA1	**f_1]**
[SA2	**f_2]**
[SA3	**f_3]**
[SA4	**f_4]**
[MVX	**f_5]**
[MVZ	**f_6]**
[JX	**f_7]**
[JZ	**f_8]**
[IE	**f_9]**
[RDX	**f_{10}]**
[ROX	**f_{11}]**
[RDZ	**f_{12}]**
[ROZ	**f_{13}]**
[SCLASS	**f_{14}]**

This nrc-spec is written in the following format in the input file:

DEFINE NRC 2005 LOAD

SA1 f_1 SA2 f_2 SA3 f_3 SA4 f_4 MVX f_5 MVZ f_6 JX f_7 JZ f_8 IE f_9 RDX f_{10} ROX f_{11} RDZ f_{12} ROZ f_{13} SCLASS f_{14}

where,

f_1, f_2, f_3 and f_4 = Seismic Data as per Table C-2.

f_5, f_6 = The higher mode factor along X and Z directions respectively. Please refer Table 4.1.8.11.

f_7, f_8 = The numerical reduction coefficient for base overturning moment along X and Z directions respectively. Please refer Table 4.1.8.11.

f_9 = the earthquake importance factor of the structure. This is an user input depending on Importance Category and ULS / SLS. Please refer Table 4.1.8.11.

f_{10}, f_{12} = are the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behaviour as described in article 4.1.8.9. along X and Z directions respectively. Please refer Table 4.1.8.9.

f_{11}, f_{13} = are the overstrength-related force modification factor accounting for the dependable portion of reserve strength in a structure designed according to the provision of the Article 4.1.8.9. along X and Z directions respectively. Please refer Table 4.1.8.9.

f_{14} = is the Site Class starting from A to E. F_a and F_v are determined based on Site Class as per Table 4.1.8.4.B and Table 4.1.8.4.C.

w = joint weight associated with list

UNI = specifies a uniformly distributed load with a value of v_1 starting at a distance of v_2 (from the start of the member) and ending at a distance of v_3 (from the start of the member). If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

CON = specifies a concentrated force with a value of v_4 applied at a distance of v_5 (from the start of the member). If v_5 is omitted, the load is assumed to act at the center of the member.

p = weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (see section 5.32.4 of STAAD Technical Reference Manual for details of the Floor Load input).

Example

```
DEFINE NRC 2005 LOAD
SA1 .33 SA2 .25 SA3 .16 SA4 .091 MVX 1.2 MVZ 1.5 JX .7 JZ .5 IE 1.3 -
RDX 4.0 ROX 1.5 RDZ 3.0 ROZ 1.3 SCLASS 4
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5

LOAD 1 EARTHQUAKE ALONG X
NRC LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

5.31.2.12 Turkish Seismic Code – Specification for Structures to be built in Disaster Areas; Part – III – Earthquake Disaster Prevention

Purpose

This set of commands may be used to define the parameters for generation of equivalent static lateral loads for seismic analysis per the specifications laid out in "*Specification for Structures to be Built in Disaster Areas Part – III – Earthquake Disaster Prevention*" Amended on 2.7.1998, Official Gazette No. 23390 (English Translation). This is referred to as the Turkish Seismic Provisions.

Description

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The minimum lateral seismic force or base shear (Vt) is automatically calculated by STAAD using the appropriate equation(s).

See Example Problem Examp14_Turkish.std

(A) $\mathbf{V_t = W\ A(T_1) / R_a(T_1)}$

(as per section 6.7.1.1 equation 6.4 of the above mentioned code),

where T_1 is fundamental time period of the structure.

Except that $\mathbf{V_t}$ shall not be less than,

$0.10\ A_o\ I\ W$ (i.e. lower limit of Vt)

Seismic Load Reduction Factor, $R_a(T_1)$ in above equation is determined based on the following equations 6.3a and 6.3b in the code:-

$R_a(T_1) = 1.5 + (R - 1.5)\ T1 / T_{\boldsymbol{A}}$ $(0 \le T1 \le T_A)$

$R_a(T_1) = R$ $(T1 > T_A)$

Structural Behavior Factor, R is provided through user input along the direction of calculation. *Spectrum Characteristics Period*, T_A and T_B are also provided by user through the parameters.

The description of the terms of the equation to calculate Vt:

T_1 is *the fundamental Lateral period* in the direction under consideration and is determined as:

(a) calculated by the empirical formulae as described below provided hn is in meter:

$T_{\boldsymbol{1}} = C_T\ [hn]^{3/4}$

Where C_T is assumed to be 0.075 for steel moment frames, 0.085 for concrete moment frames, or any user specified value.

(b) The period is also calculated in accordance with the *Raleigh* method but could be overridden by user specified time period (PX, PZ).

The time period calculated based on method (a) is used in further calculation unless it is greater than 1.0 sec and 1.3 times of this is greater than the same calculated based on method (b). In that case time period calculated based on method (b) is used.

A(T_1) is the *Spectral Acceleration Coefficient* is determined as follows as per eq. 6.1,

$A(T_1) = Ao\ I\ S(T_1)$

Ao and I in above equation are *Effective Ground Acceleration Coefficient and Building Importance Factor* are provided by the user through the load definition parameter and could be found in table 6.2 and 6.3 respectively in the code.

$S(T_1)$ is the *Spectrum Coefficient* is found by following equations, could be found in eq. 6.2a, 6.2b and 6.3c in original code

$S(T_1) = 1 + 1.5\ T_1 / T_A$ $\quad (0 \leq T \leq T_A)$

$S(T_1) = 2.5$ $\quad (T_A < T \leq T_B)$

$S(T_1) = 2.5\ (T_B / T_1)^{0.8}$ $\quad (T > T_B)$

W is the weight of the building and shall be calculated internally using the following formula:

$$W = \sum_{i=1}^{N} W_i$$

Where W_i is the portion of W that is located at or assigned to level i.

(B) As per 4.1.8.11(6), the total lateral seismic force, Vt, shall be distributed such that a portion, F_t shall be concentrated at the top of the building, where,

$\Delta F_N = 0.07\ T_1\ V_t$

but ΔF_N is not greater than $0.20V_t$

and $\Delta F_N = 0$ when $H_N <= 25$ m.

The remainder (V- ΔF_N), shall be distributed along the height of the building, including the top level, in accordance with the following formula:

$F_i = (V_t - \Delta F_N)\ w_i\ H_i / \Sigma\ w_j\ H_j$ (as per eq. 6.9)

Where,

F_i is the lateral force applied to level i.

ΔF_N is the portion of Vt to be concentrated at the top of the structure.

Wi,Wj are the portion of W that is located at or assigned to level i or j respectively.

level i is any level in the building, i=1 for first level above the base.
level N is uppermost in the main portion of the structure.

STAAD utilizes the following format to generate the lateral seismic loads.

General format:

DEFINE TURKISH LOAD

```
*tur-spec
SELFWEIGT
JOINT WEIGHT
Joint-list  WEIGHT w
MEMBER  WEIGHT
mem-list    {UNI   v1 v2 v3 }
            {CON     v4 v5   }
ELEMENT WEIGHT
Plate-list   PRESS p1
FLOOR WEIGHT
YRANGE.......
```

(see Section 5.32.4 for input description)

Where,

***tur-spec =**
[A f_1]
[TA f_2]
[TB f_3]
[I f_4]
[RX f_5]
[RZ f_6]
([CT f_7])
([PX f_8])
([PZ f_9])

This tur-spec is written in the following format in the editor:

DEFINE TURKISH LOAD
A f1 TA f2 TB f3 I f4 RX f5 RZ f6 CT f7 PX f8 PZ f9

where,

f1 = Effective Ground Acceleration Coefficient, Ao. Refer table 6.2

f2, f3 = Spectrum Characteristic Periods, TA and TB. These are user input and found in Table 6.4

f5 and f6 = Structural Behavior Factors (R) along X and Z directions respectively. These are user input and please refer Table 6.5

f4 = the earthquake importance factor of the structure.

f10, f12 = are the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior as described in article 4.1.8.9. along X and Z directions respectively. These are user input and please refer Table 4.1.8.9.

f8 = Optional CT value to calculate time period. See section 1617.4.2.1, equation 16-39 of IBC 2000 and section 9.5.5.3.2, equation 9.5.5.3.2-1 of ASCE 7-02.

f9 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

f10 = Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure instead of the value derived from section 1617.4.2 of IBC 2000, and section 9.5.5.3 of ASCE 7-02.

w = joint weight associated with list

UNI = specifies a uniformly distributed load with a value of v_1 starting at a distance of v_2 (from the start of the member) and ending at a distance of v_3 (from the start of the member). If v_2 and v_3 are omitted, the load is assumed to cover the entire length of the member.

CON = specifies a concentrated force with a value of v_4 applied at a distance of v_5 (from the start of the member). If v_5 is omitted, the load is assumed to act at the center of the member.

p = weight per unit area for the plates selected. Assumed to be uniform over the entire plate.

Element Weight is used if plate elements are part of the model, and uniform pressures on the plates are to be considered in weight calculation.

Floor Weight is used if the pressure is on a region bounded by beams, but the entity which constitutes the region, such as a slab, is not defined as part of the structural model. It is used in the same sort of situation in which one uses FLOOR LOADS (see section 5.32.4 of STAAD Technical Reference Manual for details of the Floor Load input).

Example

```
DEFINE TUR LOAD
A  0.40 TA 0.10 TB 0.30 I 1.4 RX 3.0 RZ 3.0
SELFWEIGHT
JOINT WEIGHT
17 TO 48 WEIGHT 7
49 TO 64 WEIGHT 3.5

LOAD 1 EARTHQUAKE ALONG X
TUR LOAD X 1.0
PERFORM ANALYSIS PRINT LOAD DATA
CHANGE
```

5.31.2.13 IBC 2006 Seismic Load Definition

Description

The specifications of the seismic loading chapters of the International Code Council's IBC 2006 code and the ASCE 7-05 code for seismic analysis of a building using a static equivalent approach have been implemented as described in this section. Depending on the definition, equivalent lateral loads will be generated in the horizontal direction(s).

The seismic load generator can be used to generate lateral loads in the X & Z directions for Y up or X & Y for Z up. Y up or Z up is the vertical axis and the direction of gravity loads (See the SET Z UP command). All vertical coordinates of the floors above the base must be positive and the vertical axis must be perpendicular to the floors.

The rules described in section 1613 of the ICC IBC-2006 code (except 1613.5.5) have been implemented. This section directs the engineer to the ASCE 7-2005 code. The specific section numbers of ASCE 7- those which are implemented, and those which are not implemented, are shown in the table below.

Implemented sections of IBC 2006 (ASCE 7-2005)	**Omitted sections of IBC 2006 (ASCE 7-2005)**
11.4	12.8.4.1
11.5	12.8.4.3 and onwards
12.8	

Methodology

The design base shear is computed in accordance with the equations shown below.

$$V = C_S W$$ ....Eqn12.8-1 of ASCE 7-05

The seismic response coefficient, C_s, is determined in accordance with the following equation:

$$C_s = \frac{S_{DS}}{\left[\dfrac{R}{I_E}\right]}$$ ....Eqn12.8-2 of ASCE 7-05

For IBC 2006, C_s need not exceed the following limits defined in ASCE 7-05:-

$$C_s = \frac{S_{D1}}{T\left(\dfrac{R}{I}\right)} \text{ for } T \le T_L$$ ....Eqn (12.8-3) of ASCE 7-05

$$C_s = \frac{S_{D1}\,T_L}{T^2\left(\dfrac{R}{I}\right)} \text{ for } T > T_L$$ .... Eqn.(12.8-4) of ASCE 7-05

C_s shall not be less than

$$C_s = 0.01$$ .... Eqn (12.8-5) of ASCE 7-05

In addition, for structures located where S_1 is equal to or greater than 0.6g, C_s shall not be less than

$$C_s = \frac{0.5 S_1}{\left(\dfrac{R}{I}\right)}$$ .... Eqn (12.8-6) of ASCE 7-05

For an explanation of the terms used in the above equations, please refer to IBC 2006 and ASCE 7-05 codes.

Command

There are 2 stages of command specification for generating lateral loads. This is the first stage and is activated through the DEFINE IBC 2006 LOAD command.

For IBC 2006 using a known zip code, the command is thus:-

<u>DEF</u>INE <u>IBC</u> 2006 (<u>ACC</u>IDENTAL) <u>LOA</u>D

<u>ZIP</u> f11 <u>RX</u> f5 <u>RZ</u> f6 <u>I</u>E f4 <u>TL</u> f15 <u>SCL</u>ASS f7 (<u>CT</u> f8) (<u>PX</u> f9) (<u>PZ</u> f10) (<u>K</u> f16) (<u>FA</u> f17) (<u>FV</u> f18)

***Weight spec* (See section 5.31.2.2 of the Technical Reference manual for complete weight specification)**

For IBC 2006 using a known longitude and latitude, the command is thus:-

<u>DEF</u>INE <u>IBC</u> 2006 (<u>ACC</u>IDENTAL) <u>LOA</u>D

<u>LAT</u> f12 <u>LON</u>G f13 <u>RX</u> f5 <u>RZ</u> f6 <u>I</u>E f4 <u>TL</u> f15 <u>SCL</u>ASS f7 (<u>CT</u> f8) (<u>PX</u> f9) (<u>PZ</u> f10) (<u>K</u> f16) (<u>FA</u> f17) (<u>FV</u> f18)

***Weight spec* (See section 5.31.2.2 of the Technical Reference manual for complete weight specification)**

For IBC 2006 using specific SS and S1 values, the command is thus:-

<u>DEF</u>INE <u>IBC</u> 2006 (<u>ACC</u>IDENTAL) <u>LOA</u>D

<u>SS</u> f14 <u>S1</u> f3 <u>RX</u> f5 <u>RZ</u> f6 <u>I</u>E f4 <u>TL</u> f15 <u>SCL</u>ASS f7 (<u>CT</u> f8) (<u>PX</u> f9) (<u>PZ</u> f10) (<u>K</u> f16) (<u>FA</u> f17) (<u>FV</u> f18)

***Weight spec* (See section 5.31.2.2 of the Technical Reference manual for complete weight specification)**

where,

S1 f3 = the mapped MCE spectral response acceleration at a period of 1 second as determined in accordance with Section 11.4.1 ASCE7-05

IE f4 = Occupancy importance factor. ***(IBC 2006 Clause 1604.5, ASCE 7-05 Table 11.5-1)***

RX f5 = The response modification factor for lateral load along the X direction, ***(ASCE Table 12.2.1).*** *This is the value used in "R" in the equations shown above for calculating Cs.*

RZ f6 = The response modification factor for lateral load along the Z direction, ***(ASCE Table 12.2.1)*** *This is the value used in "R" in the equations shown above for calculating Cs.*

SCLASS f7 = Site class. Enter 1 through 6 in place of A through F, see table below. ***(IBC 2006 Table 1613.5.2, ASCE 7-05 Section 20.3)***

The Soil Profile Type parameter SCLASS can take on values from 1 to 6. These relate to the values shown in Site Class Definitions Table in the following manner:-

IBC Class	**SCLASS value**
A	1
B	2
C	3
D	4
E	5
F	6

CT f8 = Optional CT value to calculate time period. ***(ASCE 7-05 Table 12.8-2).***

PX f9 = Optional Period of structure (in sec) in X-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. ***(ASCE 7-05 Table 12.8-2).***

PZ f10 = Optional Period of structure (in sec) in Z-direction to be used as fundamental period of the structure. If not entered the value is calculated from the code. ***(ASCE 7-05 Table 12.8-2).***

ZIP f11 = The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. ***(ASCE 7-05 Chapter 22).***

LAT f12 = The latitude of the site used with the longitude to determine the Ss and S1 factors. ***(ASCE 7-05 Chapter 22).***

LONG f13 = The longitude of the site used with the latitude to determine the Ss and S1 factors. ***(ASCE 7-05 Chapter 22).***

SS f14 = Mapped MCE for 0.2s spectral response acceleration. ***(IBC 2006 Clause 1613.5.1, ASCE 7-05 Clause 11.4.1).***

TL f15 = Long-Period transition period in seconds. ***(ASCE 7-05 Clause 11.4.5 and Chapter 22).***

K f16 = Exponent used in equation 12.8-7, ASCE 7. ***(ASCE 7-2005 table 12.8-2 page 129).***

FA f17 = Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e. 6). ***(IBC 2006 Clause 1613.5.3, ASCE 7-05 Section 11.4.3).***

FV f18 = Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e. 6). ***(IBC 2006 Clause 1613.5.3, ASCE 7-05 Section 11.4.3).***

Example

```
DEFINE IBC 2006
LAT 38.0165 LONG -122.105 I 1.25 RX 2.5 RZ 2.5 SCLASS 4 TL 12 FA 1 FV 1.5
SELFWEIGHT
JOINT  WEIGHT
51 56 93 100 WEIGHT 650
MEMBER WEIGHT
151 TO 156 158 159 222 TO 225 324 TO 331 UNI 45
```

Steps used to calculate and distribute the base shear are as follows:

1. The Time Period of the structure is calculated based on section 12.8.1 of ASCE 7-05 (IBC 2006). This is reported in the output as Ta.
2. The period is also calculated in accordance with the Rayleigh method. This is reported in the output as T.
3. The user may override the Rayleigh based period by specifying a value for PX or PZ (Items f7 and f8) depending on the direction of the IBC load.
4. The governing Time Period of the structure is then chosen between the above two periods, and the additional guidance provided in section 12.8.2.1 of ASCE 7-05 (IBC 2006). The resulting value is reported as "Time Period used" in the output file.
5. The Design Base Shear is calculated based on equation 12.8-1 of ASCE 7-05 (IBC 2006). It is then distributed at each floor using the rules of clause 12.83, equations 12.8-11, 12.8-12 and 12.8-13 of ASCE 7-05.
6. If the ACCIDENTAL option is specified, the program calculates the additional torsional moment. The lever arm for calculating the torsional moment is obtained as 5% of the building dimension at each floor level perpendicular to the direction of the IBC load (section 12.8.4.2 of ASCE 7-05 for IBC 2006). At each joint where a weight is located, the lateral seismic force acting at that joint is multiplied by this lever arm to obtain the torsional moment at that joint.

7. The amplification of accidental torsional moment, as described in Section 12.8.4.3 of the ASCE 7-05 code, is not implemented.
8. The story drift determination as explained in Section 12.8.6 of the ASCE 7-05 code is not implemented in STAAD.

The following example shows the commands required to enable the program to generate the lateral loads. Users may refer to Section 5.32.12 of the Technical Reference Manual for this information.

Example

```
LOAD 1 (SEISMIC LOAD IN X DIRECTION)
IBC LOAD X 0.75
LOAD 2 (SEISMIC LOAD IN Z DIRECTION)
IBC LOAD Z 0.75
```

5.31.3 Definition of Wind Load

Purpose

This set of commands may be used to define some of the parameters for generation of wind loads on the structure. See section 5.32.12, Generation of Wind Loads, for the definition of wind direction and the possible surfaces to be loaded. Section 1.17.3 of this manual describes the two types of structures on which this load generation can be performed.

See Section 1.17.3, 5.32.12 and Examples manual problem no. 15

The wind load generator can be used to generate lateral loads in the horizontal - X and Z (or Y if Z up) - directions only.

General Format:

```
DEFINE WIND LOAD
   TYPE j
   INTENSITY p1 p2 p3 ... pn HEIGHT h1 h2 h3 ... hn

   EXPOSURE     { e1 JOINT joint-list              }
                { e1 YRANGE  f1  f2  if Y UP        }
                {        or                         }
                { e1 ZRANGE  f1  f2  if Z UP        }

   EXPOSURE      e2    - same definition as e1 -
      . . .
   EXPOSURE      em     -ditto-
```

where,

j =	wind load system type number (integer)
$p_1,p_2,p_3...p_n$	wind intensities (pressures) in force/area. Up to 100 different intensities can be defined in the input file per type.
$h_1,h_2,h_3...h_n$	corresponding heights in global vertical direction up to which the above intensities occur.

$e_1, e_2, e_3 \ldots e_m$	exposure factors. A value of 1.0 means that the wind force may be applied on the full influence area associated with the joint(s) if they are also exposed to the wind load direction. Limit: 99 factors.
joint-list	Joint list associated with Exposure Factor (joint numbers or “TO” or “BY”) or enter only a group name.
f_1 and f_2	global coordinate values to specify Y (or Z if Z UP) vertical range for Exposure Factor.

If the command EXPOSURE is not specified or if a joint is not listed in an Exposure, the exposure factor for those joints is chosen as 1.0.

Description

All loads and heights are in current unit system. In the list of intensities, the first value of intensity acts from the ground level up to the first height. The second intensity (p_2) acts in the Global vertical direction between the first two heights (h_1 and h_2) and so on. The program assumes that the ground level has the lowest global vertical coordinate of any joint entered for the structure.

See Section 1.17.3 and 5.32.12

Only exposed surfaces bounded by members (not by plates or solids) will be used. The joint influence areas are computed based on surface member selection data entered in section 5.32.12 and based on the wind direction for a load case. Only joints actually exposed to the wind and connected to members will be loaded. The individual bounded areas must be planar surfaces, to a close tolerance, or they will not be loaded.

Exposure factor (e) is the fraction of the influence area associated with the joint(s) on which the load may act if it is also exposed to the wind load. Total load on a particular joint is calculated as follows.

Joint load = (Exposure Factor) X (Influence Area) X (Wind Intensity)
The exposure factor may be specified by a joint-list or by giving a vertical range within which all joints will have the same exposure. If an exposure factor is not entered or not specified for a joint,

then it defaults to 1.0 for those joints; in which case the entire influence area associated with the joint(s) will be considered.

For load generation on a closed type structure defined as a PLANE FRAME, influence area for each joint is calculated considering unit width perpendicular to the plane of the structure. The user can accommodate the actual width by incorporating it in the Exposure Factor as follows.

Exposure Factor (User Specified) = (Fraction of influence area) X (influence width for joint)

Notes

All intensities, heights and ranges must be provided in the current unit system.

If necessary, the INTENSITY, EXPOSURE, command lines can be continued on to additional lines by ending all but last line with a space and hyphen (-). Use up to 11 lines for a command.

Example

```
UNIT FEET
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.15 HEIGHT 12 24
EXPOSURE 0.90 YRANGE 11 13
EXPOSURE 0.85 JOINT 17 20 22
LOAD 1 WIND LOAD IN X-DIRECTION
WIND LOAD X 1.2 TYPE 1
```

For additional examples, see section 5.32.12 and example 15 in the Examples manual.

The Intensity line can be continued in up to 12 lines.

So the following

```
INT 0.008 0.009 0.009 0.009 0.01 0.01 0.01 0.011 0.011 0.012 0.012 0.012 HEIG 15 20
25 30 40 50 60 70 80 90 100 120
```

could be split as

```
INT 0.008 0.009 0.009 0.009 0.01 0.01 0.01 0.011 0.011 0.012 0.012 0.012 –
HEIG 15 20 25 30 40 50 60 70 80 90 100 120
```

or

```
INT 0.008 0.009 0.009 0.009 0.01 0.01-
0.01 0.011 0.011 0.012 0.012 0.012 HEIG 15 20 25 -
30 40 50 60 70 80 90 100 120
```

etc.

Persistency of Parameters used to Generate ASCE Wind Load in the STAAD Input File

In the graphical environment, under the General-Load-Definitions-Wind Definitions page, there is a facility to generate the pressure versus height table per the ASCE 7-02 wind load specifications. However, once the table is generated, the parameters which go into the derivation of this table were not retained by the graphical environment. As a result, the data had to be re-entered if any modification in the parameters was needed.

With effect from STAAD.Pro 2006, these parameters are saved in the STAAD input file. An example of it is shown below.

Example

```
DEFINE WIND LOAD
TYPE 1
<! STAAD PRO GENERATED DATA DO NOT MODIFY !!!
ASCE-7-2002:PARAMS 85.000 MPH 0 1 0 0 0.000 FT 0.000 FT 0.000 FT 1 -
1 40.000 FT 30.000 FT 25.000 FT 2.000 0.010 0 -
0 0 0 0 0.761 1.000 0.870 0.850 0 -
0 0 0 0.866 0.800 0.550
!> END GENERATED DATA BLOCK
INT 0.0111667 0.0111667 0.0113576 0.0115336 0.0116972 0.0118503 0.0119944 -
 0.0121307 0.0122601 0.0123834 0.0125012 0.0126141 0.0127226 0.012827 0.0129277 -
HEIG  0  15  16.9231  18.8461  20.7692  22.6923  24.6154  26.5385  28.4615 -
 30.3846  32.3077  34.2308  36.1538  38.0769  40
```

5.31.4 Definition of Time History Load

Purpose

This set of commands may be used to define parameters for Time History loading on the structure.

General format:

```
DEFINE TIME HISTORY (DT x)

TYPE i   { ACCELERATION        }  (SCALE f7 ) (SAVE)
         { FORCE or MOMENT     }

         { READ fn ( f8 )
         {        - OR -
         { t1 p1 t2 p2 .... tn pn
         {        - OR -
         { function-spec

where function-spec =

FUNCTION   { SINE   }
           { COSINE }

AMPLITUDE f0 { FREQUENCY } f2 (PHASE f3) CYCLES f4 { SUBDIV f6 }
             { RPM       }                        { STEP   f5 }
```

Repeat TYPE and Amplitude vs. Time sets until all are entered, then:

See Sections 1.18.3 and 5.32.10.2

```
ARRIVAL TIME
a1 a2 a3 ....... an

{ DAMPING d }
{ CDAMP     }
{ MDAMP     }
```

x = solution time step used in the step-by-step integration of the uncoupled equations. Values smaller than 0.00001 will be reset to the default DT value of 0.0013888 seconds.

i = type number of time varying load (integer). Up to 136 types may be provided. ACCELERATION indicates that the time varying load type is a ground motion. FORCE or MOMENT indicates that it is a time varying force or moment. This number should be sequential.

Scale f_7 = The scale factor option multiplies all forces, accelerations, and amplitudes entered, read or generated within this Type. Primarily used to convert acceleration in g's to current units (9.80665, 386.08858, etc.). Default is 1.0.

Save = The save option results in the creation of two files (input file name with "Tim" and "Frc" extensions) containing the history of the displacements of every node (on "Tim") and the history of the 12 end forces of every member and the history of the support reactions (on "Frc") of the structure at every time step . Syntax: TYPE 1 FORCE SAVE

t_1 p_1 t_2 p_2 ...= values of time(sec.) and corresponding force (current force unit) or acceleration (current length unit/sec^2) depending on whether the time varying load is a forcing function or a ground motion. If the data is specified through the input file, up to 499 pairs can be provided for each type in the ascending value of time. More than one line may be used if necessary. However, if the data is provided through an external file, an unlimited number of time-force pairs may be specified. If the first point is not at zero time, then the forces before the first time (but after the arrival time) will be determined by extrapolation using the first two points entered. If the first point has a nonzero force, there will be a sudden application of that force over a single integration step (DT) at that time. Zero force will be assumed for all times after the last data point.

a_1 a_2 a_3 ... a_n = Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here. The arrival times and the times from the time-force pairs will be added to get the times for a particular set of joints in the TIME LOAD data (see section 5.10.32.2). The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to Section 5.32.10.2 for information on input specification for application of the forcing function and/or ground motion loads. Up to 999 arrival time values may be specified.

d = Modal damping ratio (i.e. [Percent of critical damping]/100). Default value is 0.05 (5% of critical damping). To enter different damping for each mode, enter a Define Damp command elsewhere and MDAMP here; or for composite damping, enter CDAMP here and CDAMP ratios for all of the members / elements/ springs in the Constants and Set commands. **The default is 0.05 if no value is entered or if 0.0 is entered or if a very small number is entered.**

The "function-spec" option may be used to specify harmonic loads. Both "sine" and "cosine" harmonic functions may be specified. The program will automatically calculate the harmonic load time history based on the following specifications -

f_0 - Max. Amplitude of the forcing function in current units.

f_2 - If FREQUENCY, then cyclic frequency (cycles / sec.) If RPM, then revolutions per minute.

f_3 - Phase Angle in degrees, default = 0
f_4 - No. of cycles of loading.
f_5 - time step of loading, default = one twelfth of the period corresponding to the frequency of the harmonic loading. It is best to use the default; or
f_6 - subdivide a ¼ cycle into this many integer time steps. Default = 3.

f_5 or f_6 is used only to digitize the forcing function. It is not the DT used to integrate for the responses. More subdivisions or smaller step size will make the digitized force curve more closely match a sine wave. The default is usually adequate.

Please be aware that if a Cosine function or Sine with nonzero phase angle is entered, the force at the arrival time will be nonzero; there will be a sudden application of force over a single integration step (DT) at that time.

Example

```
UNIT . . .
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.6 5.0 2.8
TYPE 2 ACCELERATION
0.0 2.5 0.5 2.7 1.0 3.2 1.5 3.8
2.0 4.2 2.5 4.5 3.0 4.5 3.5 2.8
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075
```

Notes

The 'READ fn' command is to be provided only if the history of the time varying load is to be read from an external file. fn is the file name and may be up to 72 characters long. If the data on the file consists only of amplitudes, then enter f_8 as the delta time spacing.

The data in the external file must be provided as one or more time-force pairs per line as shown in the following example.

Data in Input file

```
UNIT  . . .
DEFINE TIME HISTORY
TYPE 1 FORCE
READ THFILE
ARRIVAL TIME
0.0
DAMPING 0.075
```

Data in the External file "THFILE"

```
0.0 1.0   1.0 1.2
2.0 1.8
3.0 2.2
4.0 2.6
```

Example for Harmonic Loading Generator

```
UNIT  . . .
DEFINE TIME HISTORY
TYPE 1 FORCE
*Following lines for Harmonic Loading Generator
FUNCTION SINE
AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100 STEP 0.02
ARRIVAL TIME
0.0
DAMPING 0.075
```

To define more than one sinusoidal load, the input specification is as follows :

```
DEFINE TIME HISTORY
TYPE 1 FORCE
FUNCTION SINE
AMPLITUDE 1.925 RPM 10794.0 CYCLES 1000
TYPE 2 FORCE
FUNCTION SINE
AMPLITUDE 1.511 RPM 9794.0 CYCLES 1000
TYPE 3 FORCE
FUNCTION SINE
AMPLITUDE 1.488 RPM 1785.0 CYCLES 1000
ARRIVAL TIME
0.0 0.0013897 0.0084034
DAMPING 0.04
```

Notes

1) By default the response (displacements, forces etc.) will contain the contribution of only those modes whose frequency is less than or equal to 108 cps. Use the CUT OFF FREQUENCY command to change this limit. Contributions of modes with frequency greater than the Cut Off Frequency are not considered.
2) Results are the individual maximums over the time period. Thus, derived quantities such as section forces and stresses, plate surface stresses and principal stresses should not be used.
3) Results from harmonic input are the maximum over the time period including the start-up transient period. These results are not the steady-state results.
4) By default, the results do not include the time period after the time loads end. Use the CUT OFF TIME command to lengthen (or shorten) the time period. If an intense short-term loading is used, the loading should be continued until after the expected peak response is reached.

5.31.5 Definition of Snow Load

Purpose

This set of commands may be used to define some of the parameters for generation of snow loads on the structure. See section 5.32.13, Generation of Snow Loads, for the definition of additional parameters and the surfaces to be loaded.

General Format:

Also see Section 5.32.13 and 1.17.4

DEFINE SNOW LOAD

TYPE f_1 **PG** f_2 **CE** f_3 **CT** f_4 **IM** f_5

Where

f1 - Type No: (limit of 100)
The “Type No.” is an integer value (1, 2, 3, etc.) which denotes a number by which the snow load type will be identified. Multiple snow load types can be created in the same model. Include as many types as needed.

f2 - Ground Snow Load (Default = 0.0)
The pressure or, weight per unit area, to be used for the calculation of the design snow load. Use a negative value to indicate loading acting towards the roof (upwards) as per section 7.2 of SEI/ASCE 7-02.

f3 - Exposure Factor (Default = 1.0)
Exposure factor as per Table 7-2 of the SEI/ASCE-7-02 code. It is dependent upon the type of exposure of the roof (fully exposed/partially exposed/sheltered) and the terrain category, as defined in section 6.5.6 of the code.

f4 - Thermal Factor (Default = 0.0)
Thermal factor as per Table 7-3 of the SEI/ASCE-7-02 code. It is dependent upon the thermal condition.

f5 - Importance Factor (Default = 1.0)
Importance factor as per Table 7-4 of the SEI/ASCE-7-02 code. This value depends on the category the structure belongs to, as per section 1.5 and Table 1-1 of the code.

Example

```
START GROUP DEFINITION
FLOOR
_ROOFSNOW 102 TO 153 159 160 TO 170 179 195 TO 197
END GROUP DEFINITION

UNIT FEET POUND
DEFINE SNOW LOAD
TYPE 1 PG 50 CE 0.7 CT 1.1 IM 1.1
```

5.31.6 Reference Load Types – Definition

Purpose

Large models can include multiple load cases which do not require analysis in their own right and are simply the building blocks for inclusion in primary load cases. This is similar to a REPEAT LOAD command, but has the added benefit of not being solved in its own right.

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

In Section 5.33 of this manual, the procedure for specifying the reference load information in active load cases is described.

General Format

DEFINE REFERENCE LOADS
LOAD R(i) LOADTYPE (type) TITLE any_load_title_you_choose
(Load items)
....
END DEFINE REFERENCE LOADS

Example

```
DEFINE REFERENCE LOADS
LOAD R1 LOADTYPE Dead  TITLE REF DEAD
SELFWEIGHT Y -1
JOINT LOAD
4071 4083 4245 4257 FY -4.04
4090 FY -0.64
ELEMENT LOAD
```

```
378 TO 379 406 TO 410 422 TO 426 PR GY -1.44
MEMBER LOAD
5006 TO 5229 UNI GY -0.64
PMEMBER LOAD
1 TRAP GY -0.347 -0.254 35.5 42
LOAD R2 LOADTYPE Live  TITLE REF LIVE
JOINT LOAD
4209 FY -6.63
4071 4083 4245 4257 FY -1.71
LOAD R3 LOADTYPE Snow  TITLE REF SNOW
JOINT LOAD
4109 FY -8.69
4071 4083 4245 4257 FY -3.29
LOAD R4 LOADTYPE Soil  TITLE REF SOIL
ELEMENT LOAD
1367 TO 1394 1396 1398 1522 1539 TO 1574 -
1575 TRAP JT -0.78 -0.78 -0.719167 -0.719167
END DEFINE REFERENCE LOADS
```

5.31.7 Definition of Direct Analysis Members (Available effective STAAD.Pro Build 03)

Purpose

This set of commands may be used to define the members whose flexural stiffness or axial stiffness is considered to contribute to the lateral stability of the structure. Also the initial value of τ_b for each member can be set here. See Appendix 7 ANSI/AISC 360-05.

Members listed with FLEX will have their EI factored by 0.80 times τ_b while performing the global solution. The final member forces and code check will be with 100% of the flexural stiffness. Members listed with AXIAL will have their EA factored by 0.80 while performing the global solution. The final member forces and code check will be with 100% of the axial stiffness.

The FYLD value here will only be used for the calculation of τ_b.

General Format:

```
DEFINE DIRECT
                              *  { XR f4 f5       }
FLEX      (f1)                   { YR f4 f5       }
AXIAL                            { ZR f4 f5       }
FYLD      (f2)                   { MEM memb-list  }
                                 { LIST memb-list }
                                 { ALL            }

NOTIONAL LOAD FACTOR      (f3)
```

where:

f1 Tau-b value. Default is 1.0. See Appendix 7 ANSI/AISC 360-05

f2 Yield Strength in current units. Default is 36.0 KSI.

f3 Notional Load Factor. Default is 0.002. If 0.003 or greater is entered, then τ_b will be set to 1.0 and no iterations will be performed.

5.32 Loading Specifications

Purpose

This section describes the various loading options available in STAAD. The following command may be used to initiate a new load case.

LOADING i1 (LOADTYPE a) (REDUCIBLE) (TITLE any_load_title)

i1 = any unique integer number (up to five digits) to identify the load case. This number need not be sequential with the previous load number.

a = one of the following.

Dead
Live
Wind
Seismic
Snow
Rain Water/Ice
Dust
Traffic
Temperature
Imperfection
Accidental
Mass

The words "LOADTYPE a" are necessary only if one intends to perform a graphics based operation called Automatic Load Combination generation. For details, see Section AD.2003.28.8 of the Software release report for STAAD.Pro 2003 second edition, and, Sections 2.2.12 and 6.9 of the GUI manual

The keyword REDUCIBLE should be used only when the loadtype is LIVE. It instructs the program to reduce a floor live load according to the provisions of UBC 1997, IBC 2000 and IBC 2003 codes. For details, see section 5.32.4 of this manual.

Under this heading, all different loads related to this loading number can be input. These different kinds of loads are described in the remaining sub-sections below.

Note: For Mass Model Loading in Dynamics please read Mass Modeling in Section 1.18.3 carefully. For the purpose of entering the mass distribution for the first dynamic load case, use the following sections, 5.32.1 through 5.32.4 plus 5.32.9.

The purpose of the mass modeling step is to create lumped masses at the joints that the eigensolution can use. The member/element loading is only a convenience in generating the joint masses. Analytically the masses are not in the elements but are lumped at the joints.

The absolute value of joint loads or loads distributed to joints from member/element loadings will be treated as weights. The moments applied to member/elements or computed at joints as a result of member/element loadings will be ignored. Only moments (actually weight moment of inertia, force-length^2 units) applied in the Joint

Load command will be used in defining the weight moment of inertias at joints. For slave joint directions, the associated joint weight or weight moment of inertia will be moved to the master. In addition, the translational weights at slave joint directions will be multiplied by the square of the distance to the master to get the additional weight moment of inertia at the master. Cross product weight moment of inertias at the master will be ignored.

5.32.1 Joint Load Specification

Purpose

This set of commands may be used to specify JOINT loads on the structure. For dynamic mass modeling see sections 5.32 and 1.18.3.

General format:

```
JOINT LOAD
                        * { FX   f1 }
                          { FY   f2 }
        joint-list        { FZ   f3 }
                          { MX   f4 }
                          { MY   f5 }
                          { MZ   f6 }
```

FX, FY and FZ specify a force in the corresponding global direction (even at inclined support joints).
MX, MY and MZ specify a moment in the corresponding global direction.
f_1, f_2 ... f_6 are the values of the loads.

See Section 1.16.1

Example

```
JOINT LOAD
3 TO 7 9 11 FY -17.2 MZ 180.0
5 8 FX 15.1
12 MX 180.0 FZ 6.3
```

Notes

- Joint numbers may be repeated where loads are meant to be additive in the joint.
- UNIT command may be on lines in between joint-list lines.
- If moments are for dynamic mass, then the units are assumed to be force-length^2.

5.32.2 Member Load Specification

Purpose

This set of commands may be used to specify MEMBER loads on frame members.

General format:

MEMBER LOAD

$$\textbf{member-list} \begin{Bmatrix} \underline{\textbf{UNI}} \textbf{ or } \underline{\textbf{UMO}}\textbf{M} & \textbf{direction-spec} & f_1, f_2, f_3, f_4, f_{14} \\ \underline{\textbf{CON}} \textbf{ or } \underline{\textbf{CMO}}\textbf{M} & \textbf{direction-spec} & f_5, f_6, f_4, f_{14} \\ \underline{\textbf{LIN}} & \textbf{direction-spec} & f_7, f_8, f_9 \\ \underline{\textbf{TRA}}\textbf{P} & \textbf{direction-spec} & f_{10}, f_{11}, f_{12}, f_{13} \end{Bmatrix}$$

$$\textbf{direction-spec} = \begin{Bmatrix} \underline{\textbf{X}} \\ \underline{\textbf{Y}} \\ \underline{\textbf{Z}} \\ \underline{\textbf{GX}} \\ \underline{\textbf{GY}} \\ \underline{\textbf{GZ}} \\ \underline{\textbf{PX}} \\ \underline{\textbf{PY}} \\ \underline{\textbf{PZ}} \end{Bmatrix}$$

UNI or UMOM specifies a uniformly distributed load or moment with a value of f_1, at a distance of f_2 from the start of the member to the start of the load, and a distance of f_3 from the start of the member to the end of the load. The load is assumed to cover the full member length if f_2 and f_3 are omitted. UMOM is not available for tapered members.

CON or CMOM specifies a concentrated force or moment with a value of f_5 applied at a distance of f_6 from the start of the member. f_6 will default to half the member length if omitted.

---

f_4 = Perpendicular distance from the member shear center to the local plane of loading. The value is positive in the general direction of the parallel (or close to parallel) local axis. The local x component of force is not offset. If global or projected load is selected, then the local Y component of load is offset the f_4 distance; the local Z component is offset the f_4 distance; and the local X component is not offset.

-OR-

f_4 = If f_{14} is not blank, then f_4 is the perpendicular to local y distance from the member shear center to the local z-plane of loading.

f_{14} = Perpendicular to local z distance from the member shear center to the local y-plane of loading.
The local Y component of load is offset the f_4 distance; the local Z component is offset the f_{14} distance; and the local X component is not offset.

LIN specifies a linearly decreasing or increasing, or a triangular load. If the load is linearly increasing or decreasing then f_7 is the value at the start of the member and f_8 is the value at the end. If the load is triangular, then f_7 and f_8 are input as zero and f_9 is the value of the load in the middle of the member.

TRAP specifies a trapezoidal linearly varying load that may act over the full or partial length of a member and in a local, global or projected direction. The starting load value is given by f_{10} and the ending load value by f_{11}. The loading location is given by f_{12}, the loading starting point, and f_{13}, the stopping point. Both are measured from the start of the member. If f_{12} and f_{13} are not given, the load is assumed to cover the full member length.

X, Y, & Z in the direction-spec and local-spec specify the direction of the load in the local (member) x, y and z-axes.

GX, GY, & GZ in the direction-spec specify the direction of the load in the global X, Y, and Z-axes.

PX, PY and PZ may be used if the load is to be along the projected length of the member in the corresponding global direction. Load start and end distances are measured along the member length and not the projected length.

Notes

See Section 1.16.2

In earlier versions of STAAD, the LINear type of member load could be applied only along the local axis of the member. It has been modified to allow for global and projected axes directions also.

If the member being loaded has offset distances (see MEMBER OFFSET specification), the location of the load is measured not from the coordinates of the starting node but from the offset distance.

Trapezoidal loads are converted into a uniform load and 8 or more concentrated loads.

A UNIT command may be on lines in between member-list lines.

If a load location is less than zero, it is reset to 0.0.

If a load location is greater than the length, it is reset to the length.

Example

```
MEMBER LOAD
619 CON GY  -2.35 5.827
68 TO 72 UNI GX -0.088 3.17 10.0
186  TRAP  GY -0.24 -0.35 0.0 7.96
3212 LIN X -5.431 -3.335
41016 UNI PZ -0.075
3724 LIN GY -6.2 -7.8
```

5.32.3 Element Load Specifications

This set of commands may be used to specify various types of loads on plate and solid elements.

5.32.3.1 Element Load Specification - Plates

Purpose

This command may be used to specify various types of ELEMENT LOADS for plates. Plate element loads must be applied following the expression

ELEMENT LOAD (PLATE)

using the format explained under the following options.

OPTION 1

$$\textbf{element-list} \quad \left[\underline{\text{PRE}}\text{SSURE} \begin{Bmatrix} \underline{\text{GX}} \\ \underline{\text{GY}} \\ \underline{\text{GZ}} \end{Bmatrix} \quad p_1 \ (x_1 \ y_1 \ x_2 \ y_2) \right]$$

This is for specifying a pressure of magnitude p_1 in one of the global axis directions on the full element or a small rectangular part of an element. If applied on a small part, (x1,y1,x2 and y2) define the corners of the rectangular region where the load is applied. If only x_1, y_1 is provided, the load is assumed as a concentrated load applied at the specified point defined by (x1,y1). If (x1,y1,x2,y2) is not provided, the load is assumed to act over the full area of the element. (x1,y1, x2 and y2) are measured from the center of the element in the local axis system (see figure later in this section). There is no option to apply the load over a projected area.

P_1 has units of force per square of length for pressure and units of force for concentrated load.

GX, GY and GZ represent the global axis directions.

OPTION 2

element-list <u>PRE</u>SSURE (Z) p_1 (x_1 y_1 x_2 y_2)

This is for specifying a constant pressure of magnitude p_1 acting perpendicular to the plane of the element on the full element or a small rectangular part of an element. This coincides with the element's local Z axis. If applied on a small part, (x1,y1,x2 and y2) define the corners of the rectangular region where the load is applied. If only x_1, y_1 is provided, the load is assumed as a concentrated load applied at the specified point defined by (x1,y1). If (x1,y1,x2,y2) is not provided, the load is assumed to act over the full area of the element. (x1,y1, x2 and y2) are measure from the center of the element in the local axis system (see figure later in this section). There is no option to apply the load over a projected area.

p_1 has units of force per square of length for pressure and units of force for concentrated load.

OPTION 3

element-list <u>PRE</u>SSURE $\left\{\begin{matrix}\textbf{LX}\\\textbf{LY}\end{matrix}\right\}$ $\left\{\textbf{p2}\right\}$

This is for specifying a constant pressure of magnitude p_2 along the local X (LX) or Y (LY) axis of the element (parallel to the element surface). An example of this type of load is friction load. With this option, a load can be applied only on the full area of the element.

p_2 has units of force per square of length.

OPTION 4

$$\textbf{element-list}\ \ \underline{\textbf{TRA}}\textbf{P}\ \left\{\begin{array}{l}\underline{\textbf{GX}}\\ \underline{\textbf{GY}}\\ \underline{\textbf{GZ}}\\ \underline{\textbf{LX}}\\ \underline{\textbf{LY}}\end{array}\right\}\ \left\{\begin{array}{l}\textbf{X}\\ \textbf{Y}\end{array}\right\}\ \ \textbf{f1}\quad \textbf{f2}$$

This is for applying a trapezoidally varying load with the following characteristics:

a) The direction of action of the load is global (GX, GY or GZ), parallel to the surface (LX or LY as in friction type loads), or normal to the element surface (local Z). The last becomes the automatic direction of action if the global or tangential directions are not specified).

b) The load varies along the local X or Y directions (imagine the wall of a tank with hydrostatic pressure where pressure at the lower nodes is higher than at the upper nodes, and hence the load varies as one travels from the bottom edge of the elements to the top edge.)

This type of load has to be applied over the full area of the element. f1 is the intensity at the I-J (or J-K) edge, and f2 is the intensity at the K-L (or L-I) edge depending on whether the load varies along "X" or "Y".

OPTION 5

$$\textbf{element-list}\ \ \underline{\textbf{TRA}}\textbf{P}\ \left\{\begin{array}{l}\underline{\textbf{GX}}\\ \underline{\textbf{GY}}\\ \underline{\textbf{GZ}}\\ \underline{\textbf{LX}}\\ \underline{\textbf{LY}}\end{array}\right\}\quad \underline{\textbf{JT}}\ \ \textbf{f3 f4 f5 f6}$$

This is for specifying a trapezoidally varying load over the full area of the element where one happens to know the intensity at the joints (JT) of the element. The load is defined by intensities of f3, f4, f5 and f6 at the 4 corners of a 4-noded element. For triangular elements, f6 is not applicable. The load can act along the global directions (GX, GY and GZ) or along the local X and Y directions (LX and LY, like a friction load).

Description for options 1 and 2

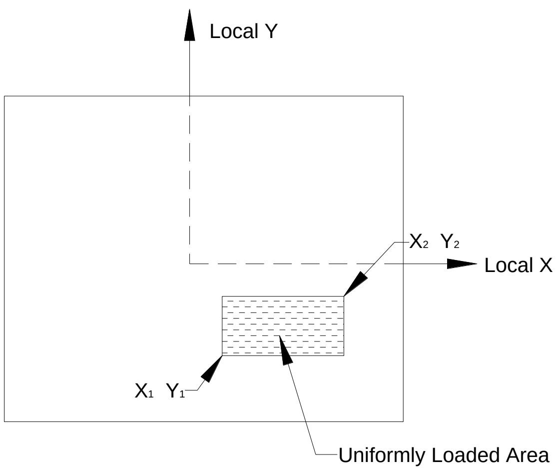


Figure 5.23

x_1,y_1 & x_2,y_2 Co-ordinate values in the local co-ordinate system

Description for option 4

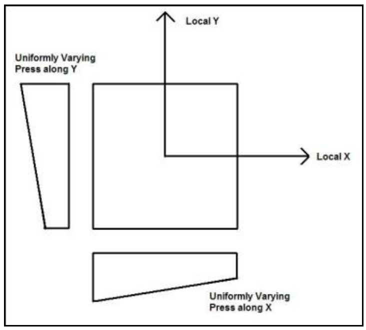


Figure 5.24

Center of the element is considered as the origin defining the rectangular area on which the pressure is applied.

The TRAP option should be used when a linearly varying pressure needs to be specified. The variation must be provided over the entire element.

X or Y	Direction of variation of element pressure. The TRAP X/Y option indicates that the variation of the Trapezoid is in the local X or in the local Y direction. The load acts in the global or local direction if selected, otherwise along the local Z axis.
f_1	Pressure intensity at start.
f_2	Pressure intensity at end.

Notes

1. “Start” and “end” defined above are based on positive directions of the local X or Y axis.
2. Pressure intensities at the joints allows linear variation of pressure in both the X and Y local directions simultaneously.
3. The TRAP load with global directions may be used to apply a volumetric type of pressure. For example, consider a grain silo with a sloping wall. In the event of modelling it using non-uniform elements, by which we mean elements whose 3 or 4 nodes are all at different elevations, the grain height at each node will depend on the elevation of the node. One can apply the pressure by specifying the intensity at each node of each element.

Example

```
LOAD 4
ELEMENT LOAD
1  7  TO  10  PR  2.5
11  12  PR  2.5  1.5  2.5  5.5  4.5
15  TO  25  TRAP  X  1.5  4.5
15  TO  20  TRAP  GY  JT  1.5  4.5  2.5  5.5
34  PR  5.0  2.5  2.5
35  TO  45  PR  -2.5
15  25  TRAP  GX  Y  1.5  4.5
29 95 TRAP LX Y 3.7 8.7
```

5.32.3.2 Element Load Specification - Solids

Purpose

Two types of loads can be assigned on the individual faces of solid elements:

1. A uniform pressure
2. A volumetric type of pressure on a face where the intensity at one node of the face can be different from that at another node on the same face. An example of such a load is the weight of water on the sloping face of a dam. If the dam is modelled using solids, for the individual elements, the water height at the lower elevation nodes will be larger than those at the higher elevation nodes.

The syntax is as follows.

General format:

ELEMENT LOAD SOLIDS

element-list FACE i_1 PRESSURE $\left\{ \begin{array}{c} \mathbf{GX} \\ \mathbf{GY} \\ \mathbf{GZ} \end{array} \right\}$ $\mathbf{f_1\ f_2\ f_3\ f_4}$

Description

The first option above loads the solid by specifying one or more of the 6 faces to receive pressure.

The PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be normal to the face and a positive pressure is into the solid. The loads are proportional to the area, not the projected area.

f_1 f_2 f_3 f_4...Pressure values at the joints for each 3 or 4 joint face defined. Only **f_1** needs to be specified for uniform pressure. In any case the pressure is provided over the entire face.

i_1 is one of six face numbers to receive the pressure for the solids selected. See figure 1.15 in section 1.6.2 for the following face definitions. Enter a pressure on all 4 joints even if the face is collapsed to 3 points.

FACE NUMBER	SURFACE JOINTS			
	f_1	**f_2**	**f_3**	**f_4**
1 front	Jt 1	Jt 4	Jt 3	Jt 2
2 bottom	Jt 1	Jt 2	Jt 6	Jt 5
3 left	Jt 1	Jt 5	Jt 8	Jt 4
4 top	Jt 4	Jt 8	Jt 7	Jt 3
5 right	Jt 2	Jt 3	Jt 7	Jt 6
6 back	Jt 5	Jt 6	Jt 7	Jt 8

Example

```
LOAD 4
ELEMENT LOAD SOLIDS
11 12 FACE 3 PR 2.5 1.5 2.5 5.5
```

5.32.3.3 Element Load Specification - Joints

Purpose

This command may be used to specify various types of element like loads for joints. Three or four joints are specified that form a plane area; pressure is specified for that area; then STAAD computes the equivalent joint loads. This command may be used as an alternative or supplement for the Area Load, Floor Load, and the other Element Load commands.

General format:

ELEMENT LOAD JOINTS

i_1 (by i_2) i_3 (by i_4) i_5 (by i_6) i_7 (by i_8) FACETS –

$$j_1 \;\; \mathbf{PRESSURE} \left\{ \begin{array}{c} \underline{GX} \\ \underline{GY} \\ \underline{GZ} \end{array} \right\} \quad f_1 \;\; f_2 \;\; f_3 \;\; f_4$$

If this data is on more than one line, the hyphens must be within the joint data.

Description

f_1 f_2 f_3 f_4...Pressure values at the joints for each 3 or 4 joint facet defined. Only f_1 needs to be specified for uniform pressure. In any case the pressure is provided over the entire element.

The PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be in the local Z direction as if the joints defined a plate. The loads are proportional to the area, not the projected area.

There will be j_1 facets loaded. The first facet is defined by joints i_1 i_3 i_5 i_7 then each joint number is incremented by the BY value (1 if omitted).

Example

```
LOAD 4
ELEMENT LOAD JOINT
1 by 1  2 by 1  32 by 1  31 by 1 –
FACETS  5  PR  GY  10  10  15  15
```

The above data is equivalent to the following :

```
LOAD 4
ELEMENT LOAD JOINT
1 2 32 31 FACETS 1 PRESSURE GY 10 10 15 15
2 3 33 32 FACETS 1 PRESSURE GY 10 10 15 15
3 4 34 33 FACETS 1 PRESSURE GY 10 10 15 15
4 5 35 34 FACETS 1 PRESSURE GY 10 10 15 15
5 6 36 35 FACETS 1 PRESSURE GY 10 10 15 15
```

So, the value following the word **FACETS** is like a counter for generation, indicating how many element faces the load command must be created for. Thus a value of 5 for facets means, a total of 5 imaginary element faces have been loaded.

BY is the value by which the individual corner node number is being incremented during the generation. In this example, the value is 1, which is same as the default. Instead, if it had been say,

```
1 by 2 2 by 3 32 by 1 31 by 1 -
FACETS 5 PRESSURE GY 10 10 15 15
```

we would have obtained

```
1 2 32 31 FACETS 1 PRESSURE GY 10 10 15 15
3 5 33 32 FACETS 1 PRESSURE GY 10 10 15 15
5 8 34 33 FACETS 1 PRESSURE GY 10 10 15 15
7 11 35 34 FACETS 1 PRESSURE GY 10 10 15 15
9 14 36 35 FACETS 1 PRESSURE GY 10 10 15 15
```

Notes:

If a pressure or volumetric load is acting on a region or surface, and the entity which makes up the surface, like a slab, is not part of the structural model, one can apply the pressure load using this facility. The load is defined in terms of the pressure intensity at the 3 or 4 nodes which can be treated as the corners of the triangular or quadrilateral plane area which makes up the region. This command may be used as an alternative or supplement for the Area Load, Floor Load, Wind Load, and other pressure load situations.

In other words, the element pressure load can be applied along a global direction on any surface, without actually having elements to model that surface. Thus, for a sloping face of a building, if one wants to apply a wind pressure on the sloping face, one can do so by specifying the joints which make up the boundary of that face. Three or four joints are specified that form a plane area; pressure is specified for that area; then STAAD computes the equivalent joint loads.

5.32.3.4 Surface Loads Specification

The following loading options are available for surface entities:

Uniform pressure on full surface

General Format

```
LOAD n
SURFACE LOAD
                                  GX
surface-list PRESSURE           { GY }  w
                                  GZ
```

GX, GY and GZ : Global X, Y and Z directions. If the direction is omitted, the load will act along the local Z axis of the surface

w = Value of pressure. Negative value indicates load acts opposite to the direction of the axis specified.

Pressure Load on partial area of the surface

General Format

```
LOAD n
SURFACE LOAD
                         GX
surface-list PRESSURE  { GY }  w x1 y1 x2 y2
                         GZ
```

w = Value of pressure. Negative value indicates load acts opposite to the direction of the axis specified.

GX, GY and GZ : Global X, Y and Z directions. If the direction is omitted, the load will act along the local Z axis of the surface x1, y1 = Local X and Y coordinates of the corner nearest to the surface origin of the loaded region. Measured from the origin of the surface, in the local coordinate system of the surface.

x2, y2 = Local X and Y coordinates of the corner farthest from the surface origin of the loaded region. Measured from the origin of the surface, in the local coordinate system of the surface.

Concentrated force on Surface

General Format

LOAD n
SURFACE LOAD
surface-list PRESSURE { GX | GY | GZ } p x y

GX, GY and GZ : Global X, Y and Z directions. If the direction is omitted, the load will act along the local Z axis of the surface

p = Value of the concentrated force. Negative sign indicates load acts opposite to the direction of the axis specified.

x, y = Local X and Y coordinates of the point of action of the force. Measured from the origin of the surface, in the local coordinate system of the surface.

Examples

```
LOAD 1 uniform pressure on full surface
SURFACE LOAD
1 2 PRESS GX 0.002
3 GX -0.0025

LOAD 2 concentrated force
SURFACE LOAD
10 12 PRE GZ 400 3.5 4.5
```

```
LOAD 3 Partial Area Load
SURFACE LOAD
23 25 PRE GY -250 4 4.3 8 9.5
```

The attributes associated with surfaces, and the sections of this manual where the information may be obtained, are listed below:

Attributes	**Related Sections**
Surfaces incidences -	5.13.3
Openings in surfaces -	5.13.3
Local coordinate system for surfaces -	1.6.3
Specifying sections for stress/force output -	5.13.3
Property for surfaces -	5.21.2
Material constants -	5.26.3
Surface loading -	5.32.3.4
Stress/Force output printing -	5.42
Shear Wall Design -	3.8.2, 5.54

5.32.4 Area Load/Oneway Load/Floor Load Specification

Purpose

These commands may be used to specify AREA LOADs, ONEWAY LOADs or FLOOR LOADs on a structure based on members only. They are used mostly when the entity transmitting the load, such as a slab, is not part of the structural model. The AREA LOAD or ONEWAY LOAD may be used for modeling one-way distribution and the FLOOR LOAD may be used for modeling two-way distribution.

General format for AREA LOAD:

<u>ARE</u>A <u>LOA</u>D

member-list <u>ALO</u>AD f_1 **(** $\left\{ \begin{array}{c} \underline{GX} \\ \underline{GY} \\ \underline{GZ} \end{array} \right\}$ **)**

See Section 1.16.3

f_1 = The value of the area load (units of weight over square length unit). If Global direction is omitted, then this load acts along the positive local y-axis [for the members of a floor analysis, the local y direction will coincide with global vertical axis in most cases]. If Global direction is included, then the load acts in that direction. The magnitude of the loads calculated is the same as if the positive local y axis option was selected. (For detailed description, refer to Section 1.)

Example

```
AREA LOAD
2 4 TO 8 ALOAD -.250
12 16 ALOAD -.500
```

Note

Area load should not be specified on members declared as MEMBER CABLE, MEMBER TRUSS or MEMBER TENSION.

General Format for ONEWAY LOAD:

ONEWAY LOAD

YRA f_1 f_2 ONELOAD f_3 (XRA f_4 f_5 ZRA f_6 f_7) * { GX | GY | GZ } TOWARDS f8

XRA f_1 f_2 ONELOAD f_3 (YRA f_4 f_5 ZRA f_6 f_7) * { GX | GY | GZ } TOWARDS f8

ZRA f_1 f_2 ONELOAD f_3 (XRA f_4 f_5 YRA f_6 f_7) * { GX | GY | GZ } TOWARDS f8

General Format for FLOOR LOAD:

FLOOR LOAD

YRA f_1 f_2 FLOAD f_3 (XRA f_4 f_5 ZRA f_6 f_7) * { GX | GY | GZ }

XRA f_1 f_2 FLOAD f_3 (YRA f_4 f_5 ZRA f_6 f_7) * { GX | GY | GZ }

ZRA f_1 f_2 FLOAD f_3 (XRA f_4 f_5 YRA f_6 f_7) * { GX | GY | GZ }

FLOOR_GROUP_NAME FLOAD f_3 * { GX | GY | GZ }

The input rules are the same for these commands.

where:

f_1 f_2 Global coordinate values to specify Y, X, or Z range. The floor/oneway load will be calculated for all members lying in that global plane within the first specified global coordinate range.

f_3 The value of the floor/oneway load (unit weight over square length unit). If Global direction is omitted, then this load acts parallel to the positive global Y if command begins with YRA and based on the area projected on a X-Z plane. Similarly, for commands beginning with XRA, the load acts parallel to the positive global X and based on the area projected on a Y-Z plane. Similarly, for commands beginning with ZRA, the load acts parallel to the positive global Z and based on the area projected on a X-Y plane.

f_4 - f_7 Global coordinate values to define the corner points of the area on which the specified floor load (f_3) acts. If not specified, the floor load will be calculated for all members in all floors within the first specified global coordinate range.

f8 Member number that signifies the direction along which the load should be applied. By default, the load is applied on the longer members of a rectangular panel. If the load must be applied on the shorter members instead, specify the member number of one of the shorter members for f8.

GX,GY,GZ: If a Global direction is included, then the load is re-directed to act in the specified direction(s) with a magnitude of the loads which is based on the area projected on a plane as if the Global direction was omitted. The Global direction option is especially useful in mass definition.

FLOORGROUP NAME: Please see section 5.16 of this manual for the procedure for creating FLOORGROUP names. The member-list contained in this name will be the candidates that will receive the load generated from the floor pressure.

Notes

1. The structure has to be modeled in such a way that the specified global axis remains perpendicular to the floor plane(s).
2. For the FLOOR LOAD specification, a two-way distribution of the load is considered. For the ONEWAY and AREA LOAD specification, a one-way action is considered. For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated for those panels. For such panels, use the FLOOR LOAD type.
3. FLOOR LOAD from a slab is distributed on the adjoining members as trapezoidal and triangular loads depending on the length of the sides as shown in the diagram. Internally, these loads are converted to multiple point loads.

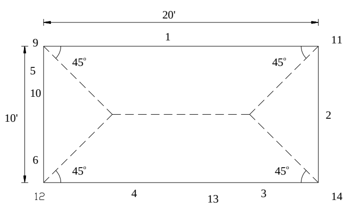


Figure 5.25

Members 1 and 2 get full trapezoidal and triangular loads respectively. Members 3 and 4 get partial trapezoidal loads and 5 and 6 get partial triangular load.

4. The load per unit area may not vary for a particular panel and it is assumed to be continuous and without holes.
5. If the floor has a shape consisting of a mixture of convex and concave edges, then break up the floor load command into several parts, each for a certain region of the floor. This will force the program to localize the search for panels and the solution will be better. See illustrative example at the end of this section.
6. At least one quadrilateral panel bounded on at least 3 sides by "complete" members has to be present within the bounds of the user-defined range of coordinates (XRANGE, YRANGE and ZRANGE) in order for the program to successfully generate member loads from the FLOOR/ONEWAY LOAD specification. A "complete" member is defined as one whose entire length between its start and end coordinates borders the specified panel.

The load distribution pattern depends upon the shape of the panel. If the panel is Rectangular, the distribution will be Trapezoidal and triangular as explained in the following diagram.

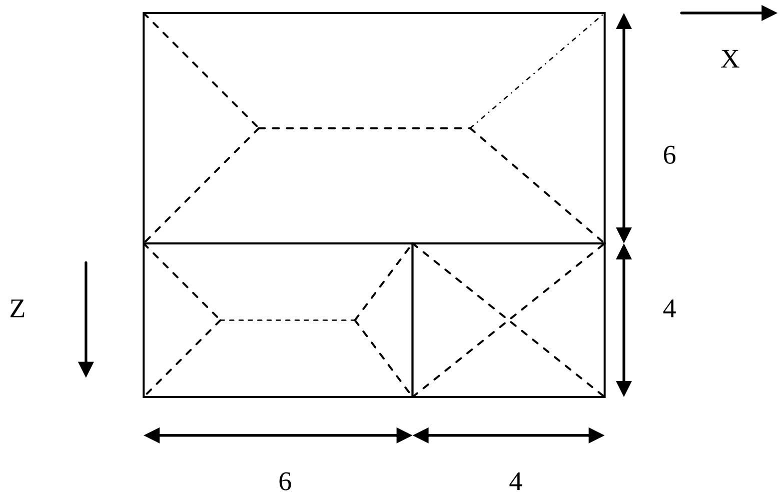


Figure 5.26

For a panel that is not rectangular, the distribution is described in following diagram.

First, the CG of the polygon is calculated. Then, each corner is connected to the CG to form triangles as shown. For each triangle, a vertical line is drawn from the CG to the opposite side. If the point of intersection of the vertical line and the side falls outside the triangle, the area of that triangle will be calculated and an equivalent uniform distributed load will be applied on that side. Otherwise a triangular load will be applied on the side.

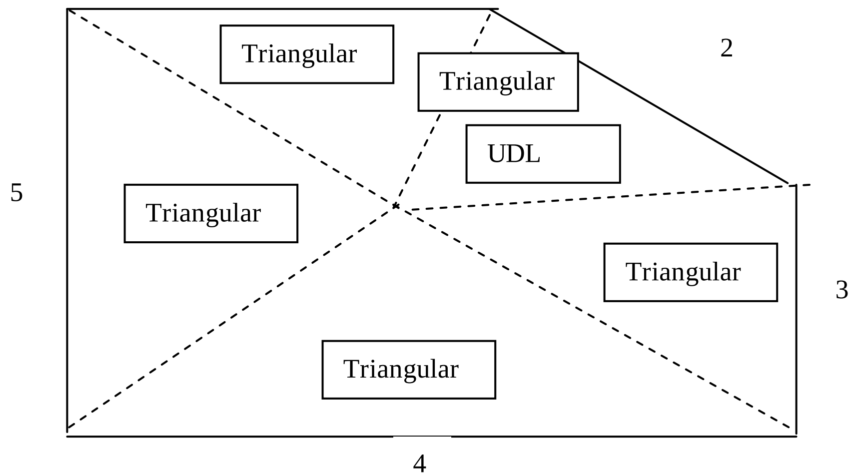


Figure 5.27

Example

The input for FLOOR LOAD is explained through an example.

Let us consider the following floor plan at y = 12.

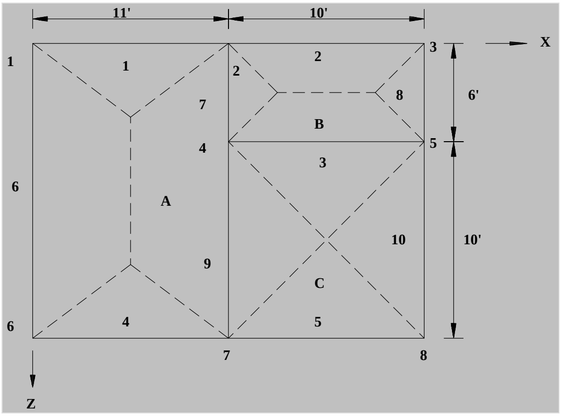


Figure 5.28

If the entire floor has a load of 0.25 (force/unit area), then the input will be as follows:

```
. . .
LOAD 2
FLOOR LOAD
YRA 12.0 12.0 FLOAD -0.25
  . . .
```

If in the above example, panel A has a load of 0.25 and panels B and C have a load of 0.5, then the input will be as follows:

Note the usage of XRANGE, YRANGE and ZRANGE specifications.

```
   ...
LOAD 2
FLOOR LOAD
YRA 11.9 12.1 FLOAD -0.25 XRA 0.0 11.0 ZRA 0.0 16.0
YRA 11.9 12.1 FLOAD -0.5 XRA 11.0 21.0 ZRA 0.0 16.0
LOAD 3
   ...
```

The program internally identifies the panels (shown as A, B and C in the figure). The floor loads are distributed as trapezoidal and triangular loads as shown by dotted lines in the figure. The negative sign for the load signifies that it is applied in the downward global Y direction.

Illustration of Notes Item (6) for FLOOR LOAD

The attached example illustrates a case where the floor has to be sub-divided into smaller regions for the floor load generation to yield proper results. The internal angle at node 6 between the sides 108 and 111 exceeds 180 degrees. A similar situation exists at node 7 also. As a result, the following command

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD –0.35
```

will not yield acceptable results. Instead, the region should be subdivided as shown in the following example

```
LOAD 1
FLOOR LOAD
YRANGE 11.9 12.1 FLOAD –0.35 XRA –0.1 15.1 ZRA –0.1 8.1
YRANGE 11.9 12.1 FLOAD –0.35 XRA 4.9 10.1  ZRA  7.9 16.1
```

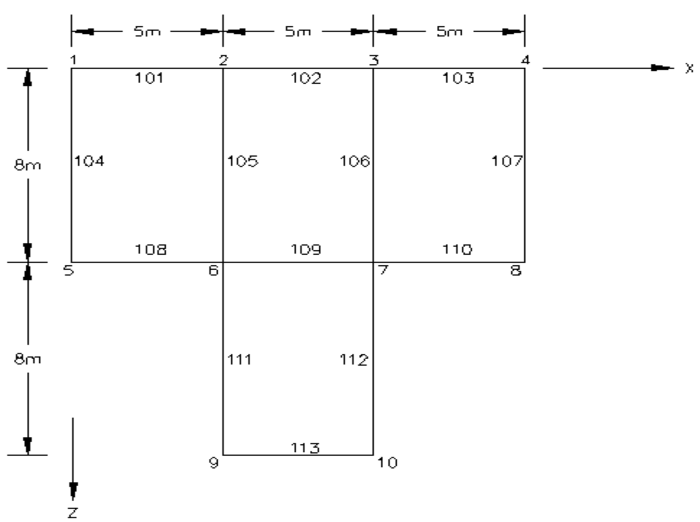


Figure 5.29

3) The global horizontal direction options (GX and GZ) enables one to consider AREA LOADs, ONEWAY LOADSs and FLOOR LOADs for mass matrix for frequency calculations.
4) For ONE WAY loads, the program attempts to find the shorter direction within panels for load generation purposes. So, if any of the panels are square in shape, no load will be generated on the members circumscribing those panels. In such cases, one ought to use the FLOOR LOAD type.

Applying floor load on members grouped under a floor group name

When applying a floor load using XRANGE, YRANGE and ZRANGE, there are two limitations that one may encounter:

a) If panels consist of members whose longitudinal axis cross each other in an X type, and if the members are not connected to each other at the point of crossing, the panel identification and hence the load generation in that panel may fail. A typical such situation is shown in the plan drawing shown in the next figure.

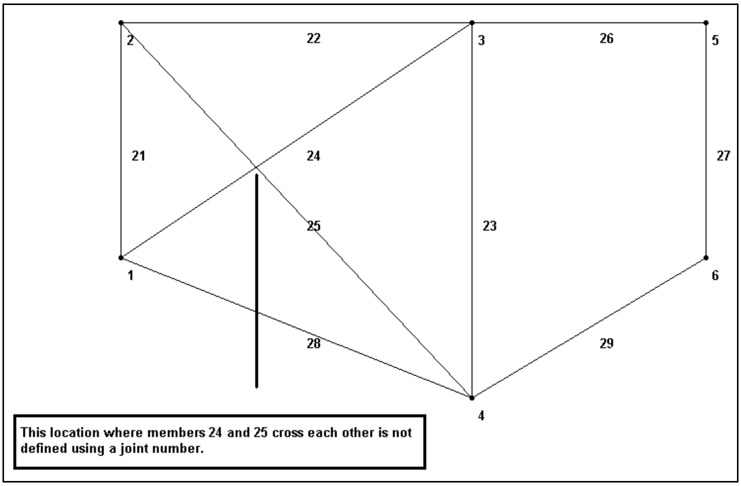


Figure 5.30

b) After the load is specified, if the user decides to change the geometry of the structure (X, Y or Z coordinates of the nodes of the regions over which the floor load is applied), she/he has to go back to the load and modify its data too, such as the XRANGE, YRANGE and ZRANGE values. In other words, the 2 sets of data are not automatically linked.

The above limitations may be overcome using a FLOOR GROUP. A GROUP name is a facility which enables us to cluster a set of entities – nodes, members, plates, solids, etc. into a single moniker through which one can address them. Details of this are available in section 5.16 of this manual.

The syntax of this command, as explained earlier in this section is:

FLOOR LOAD

Floor-group-name FLOAD f3 $\left\{\begin{matrix}\underline{GX} \\ \underline{GY} \\ \underline{GZ}\end{matrix}\right\}$ **INCLINED**

where f3 = pressure on the floor

To create equal loads in all 3 global directions for mass definition or other reasons, then enter direction labels for each direction desired; GY first then GX and/or GZ.

Example

```
START GROUP DEFINITION
FLOOR
_PNL5A  21 22 23 28
END GROUP DEFINITION

LOAD 2 FLOOR LOAD on intermediate panel @ Y = 10 ft
FLOOR LOAD
_PNL5A  FLOAD -0.45 GY
```

```
LOAD 3 FLOOR WEIGHTS FOR MASS DEFINITION
FLOOR LOAD
_PNL5A  FLOAD 0.45 GX GY GZ
```

INCLINED - This option must be used when a FLOOR LOAD is applied on a set of members that form a panel(s) which is inclined to the global XY, YZ or ZX planes.

Example

```
LOAD 5 LOAD ON SLOPING ROOF
FLOOR LOAD
_SLOPINGROOF FLOAD -0.5 GY INCLINED
```

Live load reduction per UBC and IBC Codes

The UBC 1997, IBC 2000 and IBC 2003 codes permit reduction of floor live loads under certain situations. The provisions of these codes have been incorporated in the manner described further below.

To utilize this facility, the following conditions have to be met when creating the STAAD model.

1. The live load must be applied using the FLOOR LOAD or ONEWAY LOAD option. This option is described earlier in this section of this manual, and an example of its usage may be found in example problem 15 of the Examples manual.

2. As shown in section 5.32, the load case has to be assigned a Type called Live at the time of creation of that case. Additionally, the option called **Reducible**, also has to be specified as shown.

   LOAD n LOADTYPE Live REDUCIBLE
   where n is the load case number

The following figures show the load generated on members for the two situations.

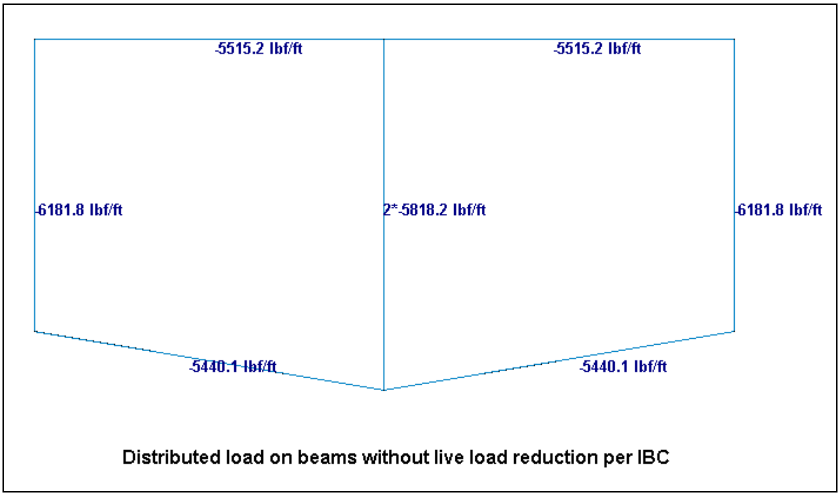


Figure 5.31

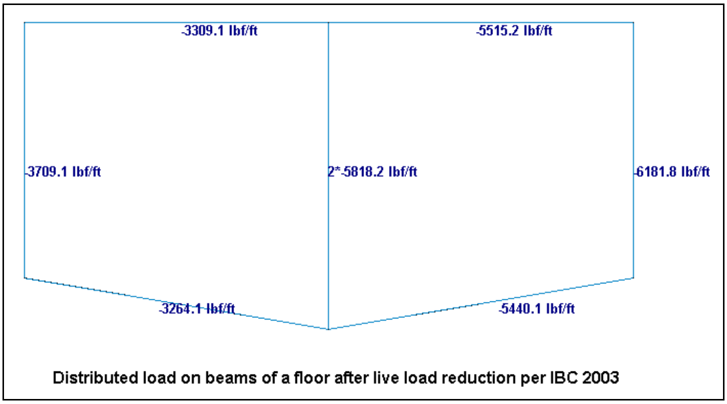


Figure 5.32

Details of the code implementation:

Code name	Section of code which has been implemented	Applicable equations
UBC 1997	1607.5, page	Equation 7-1 R = r(A-150) for FPS units R = r(A-13.94) for SI units
IBC 2000	1607.9.2, page 302	Equation 16-2 R = r(A-150) for FPS units R = r(A-13.94) for SI units
IBC 2003	1607.9.2, page 277	Equation 16-22 R = r(A-150) for FPS units R = r(A-13.94) for SI units

In the above equations,

A = area of floor supported by the member
R = reduction in percentage
R = rate of reduction equal to 0.08 for floors.

Notes:

1. Only the rules for live load on **Floors** have been implemented. The rules for live load on **Roofs** have not been implemented.

2. Since the medium of application of this method is the FLOOR LOAD or ONEWAY LOAD feature, and since STAAD performs load generation on beams only, the rules of the above-mentioned sections of the code for vertical members (columns) has not been implemented. The distributed load on those members found to satisfy the requirements explained in the code would have a lowered value after the reduction is applied.

3. Equation (7-2) of UBC 97, (16-3) of IBC 2000 and (16-23) of IBC 2003 have not been implemented.

4. In the IBC 2000 and 2003 codes, the first note says “A reduction shall not be permitted in Group A occupancies.” In STAAD, there is no direct method for conveying to the program that the occupancy type is Group A. So, it is the user’s responsibility to ensure that when he/she decides to utilize the live load reduction feature, the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

5. In the UBC 97 code, the last paragraph of section 1607.5 states that “The live load reduction shall not exceed 40 percent in garages for the storage of private pleasure cars having a capacity of not more than nine passengers per vehicle.” Again, there is no method to convey to STAAD that the structure is a garage for storing private pleasure cars. Hence, it is the user’s responsibility to ensure that the structure satisfies this requirement. If it does not, then the reduction should not be applied. STAAD does not check this condition by itself.

6. Because all the three codes follow the same rules for reduction, no provision is made available in the command syntax for specifying the code name according to which the reduction is to be done.

5.32.5 Prestress Load Specification

Purpose

This command may be used to specify PRESTRESS loads on members of the structure.

General Format:

$$\underline{\mathbf{MEMB}}\mathbf{ER} \quad \left\{ \begin{array}{l} \underline{\mathbf{PRE}}\mathbf{STRESS} \\ \underline{\mathbf{POST}}\mathbf{STRESS} \end{array} \right\} \quad (\underline{\mathbf{LOA}}\mathbf{D})$$

$$\textbf{member-list} \quad \underline{\mathbf{FORC}}\mathbf{E}\ \mathbf{f_1} \quad {}^{*}\left\{ \begin{array}{ll} \underline{\mathbf{ES}} & \mathbf{f_2} \\ \underline{\mathbf{EM}} & \mathbf{f_3} \\ \underline{\mathbf{EE}} & \mathbf{f_4} \end{array} \right\}$$

f_1 = Prestressing force. A positive value indicates precompression in the direction of the local x-axis. A negative value indicates pretension.

ES = specifies eccentricity of the prestress force at the start of the member at a distance f_2 from the centroid.

EM = specifies eccentricity of the prestress force at the mid-point of the member at a distance f_3 from the centroid.

EE = specifies eccentricity of the prestress force at the end of the member at a distance f_4 from the centroid.

Description

See Section 1.16.5

The first option, (MEMBER PRESTRESS LOAD), considers the effect of the prestressing force during its application. Thus, transverse shear generated at the ends of the member(s) subject to the prestressing force is transferred to the adjacent members.

The second option, (MEMBER POSTSTRESS LOAD), considers the effect of the existing prestress load after the prestressing operation. Thus, transverse shear at the ends of the member(s) subject to the prestressing force is not transferred to the adjacent members.

Example

```
MEMBER PRESTRESS
2 TO  7  11 FORCE 50.0
MEMBER POSTSTRESS
8 FORCE  30.0 ES  3.0  EM  -6.0  EE  3.0
```

In the first example, a prestressing force of 50 force units is applied through the centroid (i.e. no eccentricity) of members 2 to 7 and 11. In the second example, a poststressing force of 30 force units is applied with an eccentricity of 3 length units at the start, -6.0 at the middle, and 3.0 at the end of member 8.

One of the limitations in using this command is that under any one load case, on any given member, a prestress or poststress load may be applied only once. If the given member carries multiple stressed cables or has a PRESTRESS and POSTSTRESS load condition, such a situation will have to be specified through multiple load cases for that member. See example below.

Incorrect input

```
LOAD 1
MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
6 FORCE 150 ES 3 EM -6 EE 3
PERFORM ANALYSIS
```

Correct Input

```
LOAD 1
MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
LOAD 2
MEMBER PRESTRESS
6 FORCE 150 ES 3 EM -6 EE 3
LOAD COMBINATION 3
1 1.0 2 1.0
PERFORM ANALYSIS
```

Examples for Modeling Techniques

The following examples describe the partial input data for the members and cable profiles shown below.

Example 1

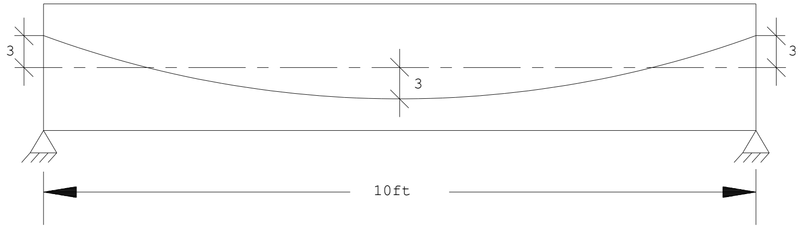


Figure 5.33

```
JOINT COORD
1 0 0 ; 2 10 0
MEMB INCI
1 1 2
.
..
UNIT  . . .
LOAD 1
MEMBER POSTSTRESS
1 FORCE 100 ES 3 EM -3 EE 3
PERFORM ANALYSIS
```

Example 2

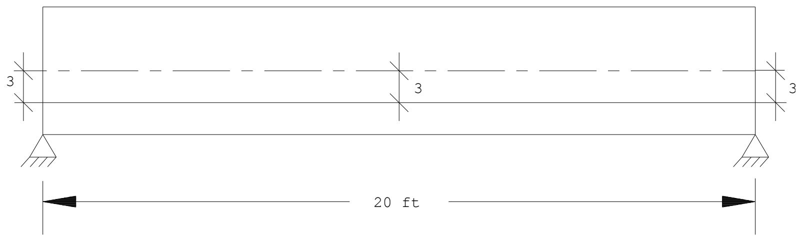


Figure 5.34

```
JOINT COORD
1 0 0 ; 2 20 0
MEMB INCI
1 1 2
.
.
.
UNIT  . . .
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES -3 EM -3 EE -3
PERFORM ANALYSIS
```

Example 3

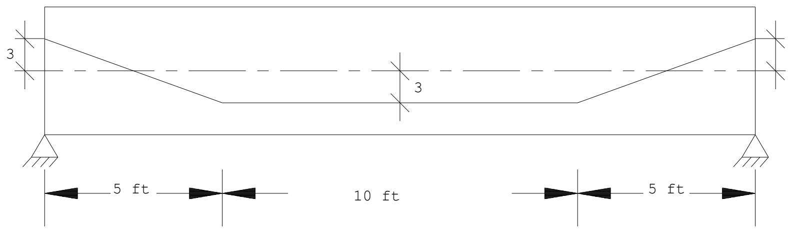


Figure 5.35

```
JOINT COORD
1 0 0 ; 2 5 0 ; 3 15 0 0 ; 4 20 0
MEMB INCI
1 1 2 ; 2 2 3 ; 3 3 4
.
.
.
UNIT  . . .
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM 0 EE -3
2 FORCE 100 ES -3 EM -3 EE -3
3 FORCE 100 ES -3 EM 0 EE 3
PERFORM ANALYSIS
```

Example 4

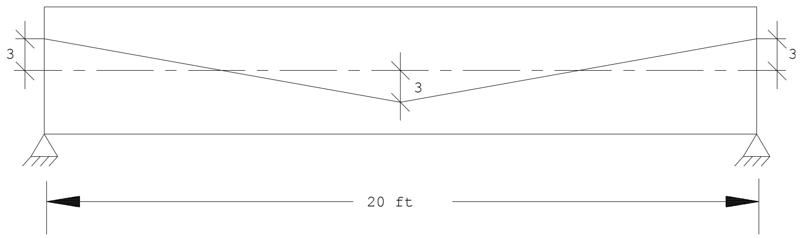


Figure 5.36

```
JOINT COORD
1 0 0 ; 2 10 0 ; 3 20 0 0
MEMB INCI
1 1 2 ; 2 2 3
.
.
.
UNIT  . . .
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM 0 EE -3
2 FORCE 100 ES -3 EM 0 EE 3
PERFORM ANALYSIS
```

Example 5

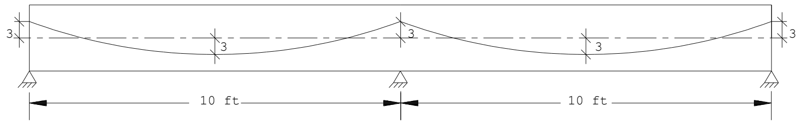


Figure 5.37

```
JOINT COORD
1 0 0 ; 2 10 0 ; 3 20 0 0
MEMB INCI
1 1 2 ; 2 2 3
.
.
.
UNIT  . . .
LOAD 1
MEMBER PRESTRESS
1 FORCE 100 ES 3 EM -3 EE 3
2 FORCE 100 ES 3 EM -3 EE 3
PERFORM ANALYSIS
```

5.32.6 Temperature Load Specification for Members, Plates, and Solids

Purpose

This command may be used to specify TEMPERATURE loads or strain loads on members, plates, and solids; or strain loads on members.

General format:

TEMPERATURE LOAD

$$\textbf{memb/elem-list} \begin{Bmatrix} \textbf{TEMP} & f_1 \; f_2 \; f_4 \\ \textbf{STRAIN} & f_3 \\ \textbf{STRAINRATE} & f_5 \end{Bmatrix}$$

f_1 = The average change in temperature (from ambient “stress-free” temperature) in the member/element which will cause axial elongation in the members or uniform volume expansion in plates and solids. The temperature unit is the same as the unit chosen for the coefficient of thermal expansion ALPHA under the CONSTANT command. (Members/Plates/Solids)

f_2 = The temperature differential from the top to the bottom of the member or plate ($T_{top\ surface}$-$T_{bottom\ surface}$). If f_2 is omitted, no bending will be considered. (Local Y axis Members/ Local Z axis Plates). Section depth must be entered for prismatic.

f_4 = The temperature differential from side to side of the member. (Local Z axis) (Members only). Section or flange width must be entered for prismatic.

See Section 1.16.6

f_3 = Initial axial elongation (+)/ shrinkage (-) in member due to misfit, etc. in length unit (Members only).

f_5 = Initial axial elongation (+)/ shrinkage (-) per unit length, of members only.

Example

```
UNIT MMS
TEMP LOAD
1 TO 9 15 17 TEMP 70.0
18 TO 23 TEMP 90.0 66.0
8 TO 13 STRAIN 3.0
15 27 STRAINRATE 0.4E -4
```

Note

It is not necessary or possible to specify the units for temperature or for ALPHA. The user must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for the temperature load. (see Section 5.26).

If ALPHA was provided by a material name (STEEL, CONCRETE, ALUMINUM) then the temperature here must be in degree Fahrenheit units (if English length units were in effect when Alpha was defined) or degree Celsius units (if metric length units were in effect when Alpha was defined).

5.32.7 Fixed-End Load Specification

Purpose

This command may be used to specify FIXED-END loads on members (beams only) of the structure.

General format:

FIXED (END) LOAD

Member_list FXLOAD f_1, f_2, ... f_{12}

member_list = normal Staad member list rules (TO and BY for generation; and - to continue list to next line).

See Section 1.16.4

f_1 ... f_6 = Force-x, shear-y, shear-z, torsion, moment-y, moment-z (all in local coordinates) at the start of the member.
f_7 ... f_{12} = Same as above except at the end of the member.

If less than 12 load values are entered, zero values will be added to the end. The loads may be extended to one additional line by ending the first load line with a - (hyphen).

These loads are given in the member local coordinate system and the directions are opposite to the actual load on the member

Old input format:

Member_number f_1, f_2, ... f_{12}

Member_number = enter only one member number per line for this format. [If you want to define the same load for many members, then use the member list form of this command shown at the beginning of this section.]

5.32.8 Support Joint Displacement Specification

Purpose

This command may be used to specify DISPLACEMENTs (or generate loads to induce specified displacements) in supported directions (pinned, fixed, enforced, or spring).

General format:

<u>SUP</u>PORT <u>DIS</u>PLACEMENT

$$\textbf{support joint-list} \quad \left\{ \begin{array}{l} \underline{\textbf{FX}} \\ \underline{\textbf{FY}} \\ \underline{\textbf{FZ}} \\ \underline{\textbf{MX}} \\ \underline{\textbf{MY}} \\ \underline{\textbf{MZ}} \end{array} \right\} \quad \mathbf{f_1}$$

See Section 1.16.7

FX, FY, FZ specify translational displacements in X, Y, and Z directions respectively. MX, MY, MZ specify rotational displacements in X, Y, and Z directions.

f_1 = Value of the corresponding displacement. For translational displacements, the unit is in the currently specified length unit, while for rotational displacements the unit is always in degrees.

Description

There are two distinct modes of usage for this command. If any “Enforced” specifications were used in the Support command then the “Displacement mode” is used; otherwise the “Load mode” is used. Despite the name of this command, if displacements are specified in spring directions, the displacement is at the joint not at the grounded end of the support spring. Displacement cannot be specified in a direction that does not have a support or a spring.

DISPLACEMENT MODE

With this mode, the support joint displacement is modeled as an imposed joint displacement. The joint directions where displacement may be specified must be defined (same for all cases) in the SUPPORT command, see section 5.27.1. Any beam members, springs or finite elements will be considered in the analysis. Other loading, inclined supports, and master/slave are all considered. Any number of cases may have displacements entered. However, all cases will have zero displacements at the enforced directions if no displacement values are entered for that case. At inclined supports the displacement specification is assumed to be in the inclined direction. Displacements may not be specified at slave directions.

If some cases are to have spring supports and others enforced displacements at the same joint directions, then two PERFORM ANALYSES must be used with the CHANGE command in between. The first perform analysis could have the SUPPORTS with springs, no enforced directions, and with the load cases without displacements. The second perform analysis would then have SUPPORTS without springs but with enforced directions and the cases with displacements.

Displacement Mode Restrictions

The Support Displacement command may be entered only once per case. Spring directions and Enforced directions may not both be specified at the same joint direction in the same Perform Analysis step.

LOAD MODE

With this mode, the support joint displacement is modeled as a load. Only beam members (not springs or finite elements) are considered in computing the joint load distribution necessary to cause the displacement. Other loading, inclined supports, and master/slave are also not considered. These unconsidered factors, if entered, will result in displacements other than those entered (results are superimposed). Only those cases with displacements entered will be affected.

Load Mode Restrictions

Support Displacements can be applied in up to 4 load cases only.
The Support Displacement command may be entered only once per case.
Finite elements should not be entered.
Inclined supports must not be entered.
Spring supports are not considered in calculating the load so their use will lead to displacements different from the input values.

Example

```
UNIT . . .
SUPPORT DISPL
5 TO 11 13 FY –0.25
19 21 to 25 MX 15.0
```

In this example, the joints of the first support list will be displaced by 0.25 units in the negative global Y direction. The joints of the second support list will be rotated by 15 degrees about the global X-axis.

5.32.9 Selfweight Load Specification

Purpose

This command may be used to calculate and apply the SELFWEIGHT of the structure for analysis.

General format:

$$\underline{\text{SEL}}\text{FWEIGHT} \quad \begin{Bmatrix} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{Bmatrix} \quad f_1$$

This command is used if the self-weight of the structure is to be considered. The self-weight of every active member is calculated and applied as a uniformly distributed member load.

X, Y, & Z represent the global direction in which the selfweight acts.

f_1 = The factor to be used to multiply the selfweight.

This command may also be used without any direction and factor specification. Thus, if specified as "SELFWEIGHT", loads will be applied in the negative global Y direction with a factor of unity.

Notes

Density must be provided for calculation of the self weight.

The selfweight of finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.

The selfweight of a plate is placed at the joints, regardless of plate releases.

5.32.10 Dynamic Loading Specification

Purpose

The command specification needed to perform response spectrum analysis and time-history analysis is explained in the following sections.

Related topics can be found in the following sections:

CUT OFF MODE -	5.30.1
CUT OFF FREQUENCY -	5.30.1
CUT OFF TIME -	5.30.1
MODE SELECTION -	5.30.2

The Steady State Analysis input can be found in Section 5.37.6.

5.32.10.1 Response Spectrum Analysis

Various methods for performing response spectrum analysis have been implemented in STAAD. They include a generic method that is described in most text books, as well as code based methods like those required by the Eurocode 8, IS 1893, etc. These are described in the following sections.

5.32.10.1.1 Response Spectrum Specification - Generic Method

Purpose

This command may be used to specify and apply the RESPONSE SPECTRUM loading for dynamic analysis.

General Format:

```
SPECTRUM  { SRSS }   * { X f1 }  { ACC }  (SCALE f4)
          { ABS  }     { Y f2 }  {     }
          { CQC  }     { Z f3 }  { DIS }
          { ASCE }
          { TEN  }

   { DAMP f5 }   ( { LIN } )  (MIS f6)   (ZPA f7)  (FF1 f8)   (FF2 f9)   (SAVE)
   { CDAMP   }     {     }
   { MDAMP   }     { LOG }
```

The data in the first line above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of 3 lines).

Starting on the next line, enter Spectra in one of these two input forms:

```
{ P1 V1; P2 V2; P3 V3;......  }      (with DAMP, CDAMP, or MDAMP)
{        or                   }
{ FILE fn                     }      (with CDAMP or MDAMP)
```

SRSS , ABS , CQC, ASCE4-98 & TEN Percent are methods of combining the responses from each mode into a total response. The CQC&ASCE4 methods require damping, ABS, SRSS, and TEN do not use damping unless Spectra-Period curves are made a function of damping (see File option below). CQC, ASCE and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE & CQC are more sophisticated and realistic methods and are recommended.

X Y Z f1, f2, f3 are the factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.

ACC or DIS indicates whether Acceleration or Displacement spectra will be entered.

SCALE f4 = Scale factor by which the spectra data will be multiplied. Usually to factor g's to length/sec^2 units.

DAMP , CDAMP , MDAMP. Select source of damping input.

f5 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).
DAMP indicates to use the f5 value for all modes.
CDAMP indicates to use Composite modal damping if entered, otherwise same as MDAMP.

MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.

LIN or LOG. Select linear or logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. LIN is default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve.
For LIN or LOG , when FILE fn is entered, the interpolation along the damping axis will be linear. <u>The last LIN-LOG parameter entered on all of the spectrum cases will be used for all spectrum cases.</u>

MIS f6 = Use Missing Mass method to include the static effect of the masses not represented in the modes. The spectral acceleration for this missing mass mode is the f6 value entered in length/sec^2 (this value is not multiplied by SCALE). If f6 is zero, then the spectral acceleration at the ZPA f7 frequency is used. If f7 is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results. If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.

ZPA f7 = For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f6 is entered.

FF1 f8 = The f1 parameter defined in the ASCE 4-98 standard in Hz units. For ASCE option only. Defaults to 2 Hz if not entered.

FF2 f9 = The f2 parameter defined in the ASCE 4-98 standard in Hz units. For ASCE option only. Defaults to 33 Hz if not entered.

Save = The save option results in the creation of a file (file name with "Acc" extension) containing the joint accelerations in g's and radians/sec^2

SPECTRA data. Choose one of these two input forms.

1. P1, V1; P2, V2; .... ; Pn, Vn. Data is part of input, immediately following SPECTRUM command. Period – Value pairs (separated by semi colons) are entered to describe the Spectrum curve. Period is in seconds and the corresponding Value is either acceleration (current length unit/sec^2) or displacement (current length unit) depending on the ACC or DIS chosen. Continue the curve data onto as many lines as needed (up to 500 spectrum pairs). Spectrum pairs must be in ascending (or descending) order of period. Note, if data is in g acceleration units, then set SCALE to a conversion factor to the current length unit (9.807, 386.1, etc.). Also note, do not end these lines with a hyphen (-). Each SPECTRUM command must be followed by Spectra data if this input form is used.

2. FILE fn data is in a separate file.

Spectra Curves on FILE fn: When the **File fn** command has been provided, then you must have the spectra curve data on a file named "fn" prior to starting the analysis. The format of the **FILE** spectra data allows spectra as a function of damping as well as period. If the FILE fn command is entered, it must be with the first spectrum case and will be used for all spectrum cases. No FILE fn command needs to be entered with the remaining spectrum cases. The format is shown below. fn may not be more than 72 characters in length.

Modal Combination Description

SRSS Square Root of Summation of Squares method.
CQC Complete Quadratic Combination method. Default.
ASCE ASCE4-98 method.
ABS Absolute sum. (Very conservative worst case)

TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (1976).

Description

See Sections 1.18.3, 5.30, and 5.34

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. Maximum response spectrum load cases allowed in one run is 4.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections 5.32 and 1.18.3. An illustration of mass modeling is available, with explanatory comments, in Example Problem No.11.

Example

```
LOAD 2 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
```

Multiple Response Spectra

If there is more than one response spectrum defined in the input file, the load data should accompany the first set of spectrum data only. In the subsequent load cases, only the spectra should be defined. See example below.

```
LOAD 1 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
SPECTRUM SRSS X 1.0 ACC SCALE 32.2
0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47
1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43
*
LOAD 2 SPECTRUM IN Y-DIRECTION
SPECTRUM SRSS Y 1.0 ACC SCALE 32.2
0.20 0.1 ; 0.40 0.15 ; 0.60 0.33 ; 0.80 0.45 ; 1.00 0.48
1.20 0.51 ; 1.4 0.63 ; 1.6 0.67 ; 1.8 0.54 ; 2.0 0.42
```

FILE FORMAT FOR SPECTRA DATA

The format of the **FILE** spectra data allows spectra as a function of damping as well as period. The format is:

Data set 1	MDAMPCV	NPOINTCV	(no of values = 2)
Data set 2	Damping Values in ascending order		(no of values = Mdampcv)
Data set 3a	Periods		(no of values = Npointcv)
3b	Spectra		(no of values = Npointcv)

Repeat Data set 3 Mdampcv times (3a,3b , 3a,3b , 3a,3b , etc.) (i.e. for each damping value).

Data sets 3a and 3b must have exactly Npointcv values each. Blanks or commas separate the values. The data may extend to several lines. Do not end lines with a hyphen (-). No comment lines (*) or semi-colons. Multiple values may be entered per line.

MDAMPCV = Number of damping values for which there will be separate Spectra vs. Period curves.

NPOINTCV = Number of points in each Spectra vs. Period curve. If NPOINTCV is negative, then the period-spectra values are entered as pairs.

5.32.10.1.2 Response Spectrum Specification in Conjunction with the Indian IS: 1893 (Part 1)-2002 Code for Dynamic Analysis

Methodology

The design lateral shear force at each floor in each mode is computed by STAAD in accordance with the Indian IS: 1893 (Part 1)-2002 equations 7.8.4.5c and 7.8.4.5d.

$$\mathbf{Q_{ik} = A_k * \phi_{ik} * P_k * W_k \text{ and } V_{ik} = \sum_{i=i+1}^{n} Q_{ik}}$$

Note: All symbols and notations in the above equation are as per IS: 1893(Part 1)-2002.

STAAD utilizes the following procedure to generate the lateral seismic loads.

1. User provides the value for $\frac{Z}{2} * \frac{I}{R}$ as factors for input spectrum
2. Program calculates time periods for first six modes or as specified by the user.
3. Program calculates S_a/g for each mode utilizing time period and damping for each mode.
4. The program calculates design horizontal acceleration spectrum A_k for different modes.
5. The program then calculates mode participation factor for different modes.
6. The peak lateral seismic force at each floor in each mode is calculated.
7. All response quantities for each mode are calculated.
8. The peak response quantities are then combined as per method (CQC or SRSS or ABS or TEN or CSM) as defined by the user to get the final results.

General Format:

SPECTRUM {Method} 1893 (TOR) X f1 Y f2 Z f3 ACC (SCALE f4)
(DAMP f5 or MDAMP or CDAMP) (MIS f6) (ZPA f7)
SOIL TYPE f8

The data in the first line above must be on the first line of the command, the second line of data can be on the first or subsequent lines with all but last ending with a hyphen (limit of 3 lines). The last line (Soil Type parameter) must be in a separate line.

where,

Method = SRSS or CQC or ABS or CSM or TEN

SRSS , CQC, ABS, CSM and **TEN** are methods of combining the responses from each mode into a total response. SRSS stands for square root of summation of squares, CQC for complete quadratic combination and ABS for Absolute method. CSM is a closely-spaced modes grouping method where the peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined by Absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method. TEN PERCENT method combines modal responses as published in US Nuclear Regulatory Guide 1.92 Revision 1 of February 1976.

Note : CQC, SRSS and CSM Grouping methods are recommended by IS:1893 (Part 1) –2002.

If SRSS is given, the program will internally check whether there are any closely spaced modes or not. If it finds any such modes, it will switch over to the CSM method. In the CSM method, the program will check whether all modes are closely spaced or not. If all modes are closely spaced, it will switch over to the CQC method.

1893 indicates the analysis as per IS:1893(Part 1)-2002 procedures.

TOR indicates that the torsional moment (in the horizontal plane) arising due to eccentricity between the centre of mass and centre of rigidity needs to be considered. If TOR is entered on any one spectrum case it will be used for all spectrum cases.

X Y Z f1, f2, f3 are the factors for the input spectrum to be applied in X, Y, & Z directions. These must be entered as the product of $\frac{Z}{2} * \frac{I}{R}$. Any one or all directions can be input. Directions not provided will default to zero.

ACC indicates Acceleration spectra will be entered.

SCALE f4 = Scale factor by which design horizontal acceleration spectrum will be multiplied. This factor signifies that the structures and foundations, at which level base shear will be calculated, are placed below the ground level. Default value is 1.0.

DAMP , MDAMP and CDAMP. Select source of damping input.

f5 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP indicates to use the f5 value for all modes.
MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
CDAMP is the composite damping of the structure calculated for each mode. One must specify damping for different materials under CONSTANT specification.

MIS f6 = Use Missing Mass method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the f6 value entered in length/sec^2 (this value is not multiplied by SCALE). If f6 is zero, then the spectral acceleration at the ZPA f7 frequency is used. If f7 is zero or not entered, the spectral acceleration at 33Hz (**Zero Period Acceleration, ZPA**) is used. The results of this calculation are SRSSed with the modal combination results. If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.

ZPA f7 = For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f6 is entered.

SOIL TYPE f8 = the types of soil. f8 is 1 for rocky or hard soil, 2 for medium soil and 3 for soft soil sites. Depending upon time period, types of soil and damping, average response acceleration coefficient, Sa/g is calculated.

Notes:

1. The design base shear V_B (calculated from the Response Spectrum method) is compared with the base shear V_b (calculated by empirical formula for the fundamental time period). If V_B is less than V_b, all of the response quantities are multiplied by V_b /V_B as per Clause 7.8.2. **To have STAAD compute the empirical base shear V_b, please enter the data described in Section 5.31.2.5 and place that data before the first load case in the STAAD input.** STAAD does not calculate the fundamental frequency of the structure needed for the empirical base shear V_b calculation; **so you must enter either the ST parameter or the PX and PZ parameters in the Section 5.31.2.5 DEFINE 1893 LOAD data.**

2. The following interpolation formula is adopted for interpolation between damping values as given in Table 3.

   Interpolation and/or extrapolation of ground response acceleration for a particular mode has been made for determining the spectrum ordinates corresponding to the modal damping value for use in Response Spectrum analysis. The relationship that shall be used for this purpose is defined by :

$$Sa = Ae^{-\xi} + \frac{B}{\xi}$$

where,

Sa = Spectrum ordinate

ξ = damping ratio

A, B = Constants

The constants A and B are determined using two known spectrum ordinates Sa_1 & Sa_2 corresponding to damping ratios ξ_1 and ξ_2 respectively for a particular time period and are as follows :

$$A = \frac{Sa_1\xi_1 - Sa_2\xi_2}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}}$$

$$B = \frac{\xi_1\xi_2\left(Sa_2 e^{-\xi_1} - Sa_1 e^{-\xi_2}\right)}{\xi_1 e^{-\xi_1} - \xi_2 e^{-\xi_2}}$$

where, $\xi_1 < \xi < \xi_2$

3. The storey drift in any storey shall not exceed 0.004 times the storey height as per Clause 7.11.1. To check this, the following command should be given after the analysis command.

PRINT STOREY DRIFT

A warning message will be printed if storey drift exceeds this limitation. If any soft storey (as per definition in Table 5 of IS:1893-2002) is detected, a warning message will be printed in the output.

5.32.10.1.3 Response Spectrum Specification per Eurocode 8 1996

Purpose

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 1996 edition of Eurocode 8 (EC8) for dynamic analysis.

General Format:

```
           ┌ SRSS ┐     *    ┌ ELASTIC ┐  ┌ X  f1 ┐
SPECTRUM   │ ABS  │   EURO   │         │  │ Y  f2 │   ACC
           │ CQC  │          └ DESIGN  ┘  └ Z  f3 ┘
           │ ASCE │
           └ TEN  ┘

   ┌ DAMP  f4 ┐    ┌ LIN ┐
   │ CDAMP    │  ( │     │ )  (MIS  f5)   (ZPA f6)   (SAVE)
   └ MDAMP    ┘    └ LOG ┘
```

The data in the first line above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of 3 lines).

Starting on the next line, the response spectra is input using following standard input parameters:

```
                  ┌ A ┐
SOIL TYPE         │ B │    ALPHA  f7  Q  f8
                  └ C ┘
```

That is, unlike the custom defined response spectra the EC 8 response spectra is not input using frequency-acceleration pairs. Based on the type of Response Spectra(Elastic/Design), Soil Type, Alpha and Q the software generates the applicable response spectra curve using the guidelines of section 4.2.2 or 4.2.4 of Eurocode 8 as applicable.

SRSS , ABS , CQC, ASCE4-98 & **TEN** Percent are methods of combining the responses from each mode into a total response. CQC, ASCE and TEN include the effect of response magnification

due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE & CQC are more sophisticated and realistic methods and are recommended.
The specifier **EURO** is mandatory to denote that the applied loading is as per the guidelines of Eurocode 8.

The response spectrum loading can be based on either **ELASTIC** or **DESIGN** response spectra.

Within the scope of Eurocode 8 the earthquake motion at a given point of the surface is generally represented by an elastic ground acceleration response spectrum termed as “Elastic Response Spectrum”. STAAD generates the Elastic Response Spectra using the guidelines of section 4.2.2 and Table 4.1 of Eurocode 8.

The capacity of structural systems to resist seismic actions in the nonlinear range generally permits their design for forces smaller than those corresponding to a linear elastic response. To avoid explicit nonlinear structural analysis in design, the energy dissipation capacity of the structure through mainly ductile behavior of its elements and/or other mechanisms, is taken into account by performing a linear analysis based on a response spectrum which is a reduced form of the corresponding elastic response spectrum. This reduction is accomplished by introducing the behavior factor Q and the reduced response spectrum is termed as “Design Response Spectrum”. STAAD generates the Elastic Response Spectra using the guidelines of section 4.2.4 and Table 4.2 of Eurocode 8.

So, if the structure is supposed to resist seismic actions in the nonlinear range the Design Response Spectra is to be used.

X Y Z f1, f2, f3 are the factors for the input spectrum to be applied in X, Y, & Z directions. Any one or all directions can be input. Directions not provided will default to zero.

ACC indicates that the Acceleration spectra will be entered. Eurocode 8 does not provide displacement response spectra.

DAMP , CDAMP , MDAMP. Select source of damping input. f4 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).
DAMP indicates to use the f4 value for all modes.
CDAMP indicates to use Composite modal damping if entered, otherwise same as MDAMP.
MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.

LIN or **LOG**. Select linear or logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. LIN is default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve. For LIN or LOG , when FILE fn is entered, the interpolation along the damping axis will be linear. <u>The last LIN-LOG parameter entered on all of the spectrum cases will be used for all spectrum cases.</u>

MIS f5 = Use Missing Mass method to include the static effect of the masses not represented in the modes. The spectral acceleration for this missing mass mode is the f5 value entered in length per second squared units (this value is not multiplied by SCALE). If f5 is zero, then the spectral acceleration at the ZPA f6 frequency is used. If f6 is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results. <u>If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.</u>

ZPA f6 = For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f5 is entered.

Save = The save option results in the creation of a file (file name with "Acc" extension) containing the joint accelerations in g's and radians/sec^2

Soil Type parameter is used to define the subsoil conditions based on which the response spectra will be generated. Based on the subsoil conditions the soil types may be of three kinds
Type A : for Rock or stiff deposits of sand
Type B :- for deep deposits of medium dense sand, gravel or medium stiff clays.
Type C:- Loose cohesionless soil deposits or deposits with soft to medium stiff cohesive soil.

Please refer section 3.2 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

ALPHA f7 :- Alpha is defined to be design ground acceleration expressed in terms of acceleration due to gravity(g).For most of the application of Eurocode 8 the hazard is described in terms of a single parameter, i.e., the value of effective peak ground acceleration in rock or firm soil ; this acceleration is termed as the design ground acceleration.

Q f8 :- Q is the behavior factor used to reduce the elastic response spectra to the design response spectra. The behavior factor is an approximation of the ratio of the seismic forces, that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.

Modal Combination Description

SRSS	Square Root of Summation of Squares method.
CQC	Complete Quadratic Combination method. Default.
ASCE	ASCE4-98 method.
ABS	Absolute sum. (Very conservative worst case)
TEN	Ten Percent Method of combining closely spaced modes.

NRC Reg. Guide 1.92 (1976).

Description

See Sections 1.18.3, 5.30, and 5.34 of STAAD Technical Reference Manual

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. Maximum response spectrum load cases allowed in one run is 4.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections 5.32 and 1.18.3 of STAAD Technical Reference Manual.

Example

```
LOAD 2 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
SPECTRUM SRSS EURO ELASTIC X 1 ACC DAMP
0.05 –
LIN MIS 0 ZPA 40
SOIL TYPE A ALPHA 2 Q 1.5
```

Multiple Response Spectra

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area are generated by sources varying widely in location and other parameters. In those cases different values of ALPHA as well as Q may be required to indicate the different shapes of response spectrum for each type of earthquake.

5.32.10.1.4 Response Spectrum Specification per Eurocode 8 2004 (EN 1998-1:2004)

Purpose

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2004 edition of Eurocode 8, 'General Rules, seismic actions and rules for buildings', BS EN 1998-1:2004. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

General Format

```
             { SRSS }  *
             { ABS  }             { ELASTIC }   { RS1 }   { X f1 }
SPECTRUM     { CQC  }   EURO 2004 {         }   {     }   { Y f2 }   ACC
             { ASCE }             { DESIGN  }   { RS2 }   { Z f3 }
             { TEN  }

{ DAMP f4 }    ( { LIN } )
{ CDAMP   }    ( {     } )   (MIS f5) (ZPA f6)   (SAVE)
{ MDAMP   }    ( { LOG } )
```

The data in the first line above must be on the first line of the command, the remaining data can be on the first or subsequent lines with all but last ending with a hyphen (limit of 3 lines).

Starting on the next line, the response spectra is input using following standard input parameters:

$$\underline{\text{SOI}}\text{L } \underline{\text{TYP}}\text{E} \left\{ \begin{array}{c} \underline{\text{A}} \\ \underline{\text{B}} \\ \underline{\text{C}} \\ \underline{\text{D}} \\ \underline{\text{E}} \end{array} \right\} \quad \underline{\text{ALP}}\text{HA } f7 \qquad \text{Q } f8$$

That is, unlike the custom defined response spectra the EC 8 response spectra is not input using frequency-acceleration pairs. Based on the type of Response Spectra(Elastic/Design), Soil Type, Alpha and Q the software generates the applicable response spectra curve using the guidelines of section 3.2.2.2 or 3.2.2.3 or 3.2.2.5 of Eurocode 8: 2004 as applicable.

SRSS, ABS, CQC, ASCE4-98 & **TEN** Percent are methods of combining the responses from each mode into a total response. CQC, ASCE and TEN include the effect of response magnification due to closely spaced modal frequencies. ASCE includes more algebraic summation of higher modes. ASCE & CQC are more sophisticated and realistic methods and are recommended.

The keywords **EURO 2004** is mandatory to denote that the applied loading is as per the guidelines of Eurocode 8.

The response spectrum loading can be based on either **ELASTIC** or **DESIGN** response spectra.

Two types of response spectra curve can be generated based on either **RS1** (for response spectra type 1 curve) or **RS2** (for response spectra type 2 curve) for each **ELASTIC** and **DESIGN** option.

Within the scope of Eurocode 8 the earthquake motion at a given point of the surface is generally represented by an elastic ground acceleration response spectrum termed as “Elastic Response Spectrum”. STAAD generates the Elastic Response Spectra using the guidelines of section 3.2.2.2 and Tables 3.2 and 3.3 for horizontal elastic response spectra and section 3.2.2.3 and Table 3.4 for vertical elastic response spectra of Eurocode 8.

The capacity of structural systems to resist seismic actions in the nonlinear range generally permits their design for forces smaller than those corresponding to a linear elastic response. To avoid explicit nonlinear structural analysis in design, the energy dissipation capacity of the structure through mainly ductile behavior of its elements and/or other mechanisms, is taken into account by performing a linear analysis based on a response spectrum which is a reduced form of the corresponding elastic response spectrum. This reduction is accomplished by introducing the behavior factor Q and the reduced response spectrum is termed as "Design Response Spectrum". STAAD generates the Design Response Spectra using the guidelines of section 3.2.2.5 and Tables 3.2 and 3.3 of Eurocode 8. So, if the structure is supposed to resist seismic actions in the nonlinear range the Design Response Spectra is to be used.

X Y Z f1, f2, f3 are the factors for the input spectrum to be applied in X, Y, & Z directions. <u>Anyone or both X and Z directions can be input for generating horizontal response spectrum. For generating vertical response spectrum, Y direction input is to be given in a separate primary load case.</u> Directions not provided will default to zero.

ACC indicates that the Acceleration spectra will be entered. Eurocode 8 does not provide displacement response spectra.

DAMP, CDAMP, MDAMP. Select source of damping input.

f4 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP indicates to use the f4 value for all modes.

CDAMP indicates to use Composite modal damping if entered, otherwise same as MDAMP.

MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.

LIN or **LOG**. Select linear or logarithmic interpolation of the input Spectra versus Period curves for determining the spectra value for a mode given its period. LIN is default. Since Spectra versus Period curves are often linear only on Log-Log scales, the logarithmic interpolation is recommended in such cases; especially if only a few points are entered in the spectra curve. <u>The last LIN-LOG parameter entered on all of the spectrum cases will be used for all spectrum cases</u>.

MIS f5 = Use Missing Mass method to include the static effect of the masses not represented in the modes. The spectral acceleration for this missing mass mode is the f5 value entered in length per second squared units (this value is not multiplied by SCALE). If f5 is zero, then the spectral acceleration at the ZPA f6 frequency is used. If f6 is zero or not entered, then the spectral acceleration at 33Hz is used. The results of this calculation are SRSSed with the modal combination results. <u>If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases</u>.

ZPA f6 = For use with MIS option only. Defaults to 33 Hz if not entered. Value is printed but not used if MIS f5 is entered.

Save = The save option results in the creation of a file (file name with "Acc" extension) containing the joint accelerations in g's and radians/sec^2

SOIL TYPE parameter is used to define the subsoil conditions based on which the response spectra will be generated. Based on the subsoil conditions the soil types may be of five kinds:-

Type A : for rock or other rock-like geological formation
Type B : for deep deposits of very dense sand, gravel or very stiff clay
Type C : for deep deposits of dense or medium-dense sand, gravel or stiff clay
Type D : for deposits of loose-to-medium cohensionless soil
Type E : for a soil profile consisting of a surface of alluvium layer

Please refer Table 3.1 of Eurocode 8 for detailed guidelines regarding the choice of soil type.

ALPHA f7 :- Alpha is defined to be design ground acceleration expressed in terms of acceleration due to gravity(g).For most of the application of Eurocode 8 the hazard is described in terms of a single parameter, i.e., the value of effective peak ground acceleration in rock or firm soil ; this acceleration is termed as the design ground acceleration.

Q f8 :- Q is the behavior factor used to reduce the elastic response spectra to the design response spectra. The behavior factor is an approximation of the ratio of the seismic forces that the structure would experience, if its response was completely elastic with 5% viscous damping, to the minimum seismic forces that may be used in design- with a conventional linear model still ensuring a satisfactory response of the structure.

Modal Combination Description

SRSS Square Root of Summation of Squares method.
CQC Complete Quadratic Combination method. Default.
ASCE ASCE4-98 method.
ABS Absolute sum. (Very conservative worst case)
TEN Ten Percent Method of combining closely spaced modes. NRC Reg. Guide 1.92 (1976).

Description

See Sections 1.18.3, 5.30, and 5.34

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. Maximum response spectrum load cases allowed in one run is 4.

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. For dynamic mass modeling, see sections 5.32 and 1.18.3 of STAAD Technical Reference Manual.

Example

```
LOAD 2 SPECTRUM IN X-DIRECTION
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
JOINT LOAD
10 FX 17.5
10 FY 17.5
10 FZ 17.5
MEMBER LOADS
5 CON GX 5.0 6.0
5 CON GY 5.0 6.0
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GX 5.0 14.0
5 CON GY 5.0 14.0
SPECTRUM SRSS EURO 2004 ELASTIC RS1 X 1 ACC
DAMP 0.05 –
LIN MIS 0 ZPA 40
SOIL TYPE A ALPHA 2 Q 1.5
```

Multiple Response Spectra

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area is generated by sources varying widely in location and other parameters. In such cases different values of ALPHA as well as Q may be required to indicate the different shapes of response spectrum for each type of earthquake.

For special conditions more than one spectrum may be needed to adequately represent the seismic hazard over an area. This happens when the earthquake affecting the area is generated by sources varying widely in location and other parameters. In such cases different values of ALPHA as well as Q may be required to indicate the different shapes of response spectrum for each type of earthquake.

5.32.10.1.5 Response Spectrum Specification in accordance with IBC 2006

Purpose

This command may be used to specify and apply the RESPONSE SPECTRUM loading as per the 2006 edition of the ICC specification ‘International Building Code (IBC)’, for dynamic analysis. The graph of frequency – acceleration pairs are calculated based on the input requirements of the command and as defined in the code.

Methodology

The methodology for calculating the response spectra is defined in ASCE-7-2005, section 11.4. The following is a quick summary:

a. Input Ss and S1 (this could have been searched from the database or entered explicitly)

b. Sms= Fa x Ss and
Sm1 = Fv x S1

where, Fa and Fv are determined from the specified site classes A – E and using tables 11.4-1 and 11.4-2. For site class F, the values must be supplied by the user. For other classes, Fa and Fv could be over-ridden through user input.

c. Calculate

Sds = (2/3) Sms

And

Sd1 = (2/3) Sm1

The spectrum is generated as per section 11.4.5.

General Format:

$$\underline{\text{SPE}}\text{CTRUM} \quad (\text{ Method }) \quad \underline{\text{IBC}}\ \underline{2006} \quad \begin{Bmatrix} X & f1 \\ Y & f2 \\ Z & f3 \end{Bmatrix} \quad \underline{\text{ACC}} \quad (\underline{\text{SCA}}\text{LE} \ \ f4)$$

$$\begin{Bmatrix} \underline{\text{DAM}}\text{P} \ \ f5 \\ \underline{\text{CDA}}\text{MP} \\ \underline{\text{MDA}}\text{MP} \end{Bmatrix} \quad (\begin{Bmatrix} \underline{\text{LIN}} \\ \\ \underline{\text{LOG}} \end{Bmatrix}) \quad (\underline{\text{MIS}} \ \ f6)\ (\underline{\text{ZPA}} \ \ f7)\ (\underline{\text{SAV}}\text{E})$$

The data in the first line above must be on the first line of the command. The data shown on the second line above can be continued on the first line or one or more new lines with all but last ending with a hyphen (limit of 3 lines).

The command is completed with the following data which must be started on a new line:-

$$\begin{Bmatrix} \underline{\text{ZIP}} \ \ f8 \\ \underline{\text{LAT}} \ \ f9\ \underline{\text{LONG}} \ \ f10 \\ \underline{\text{SS}} \ \ f11\ \underline{\text{S1}} \ \ f12 \end{Bmatrix} \quad \underline{\text{SIT}}\text{E}\ \underline{\text{CLA}}\text{SS} \quad (f29) \quad (\underline{\text{FA}} \ \ f13\text{-}\ \underline{\text{FV}} \ \ f14) \quad \underline{\text{TL}}\ f15$$

where,

f29 – see under **SITECLASS** described below.

Method = **SRSS** **ABS** **CQC** **ASCE** **TEN** **or** **CSM**

SRSS , CQC, ABS, CSM and **TEN** are methods of combining the responses from each mode into a total response. SRSS stands for square root of summation of squares, **CQC** for complete quadratic combination and ABS for Absolute method. **CSM** is a closely-spaced modes grouping method where the peak response quantities for closely spaced modes (considered to be within 10 percent of each other) are combined by Absolute method. This peak response quantity for closely spaced modes is then combined with those of widely spaced modes by SRSS method. TEN PERCENT method combines modal responses as published in US Nuclear Regulatory Guide 1.92 Revision 1 of February 1976.

IBC 2006 indicates that the spectrum should be calculated as defined in the IBC 2006 specification.

X f1, Y f2, Z f3 are the factors for the input spectrum to be applied in X, Y, & Z directions. Directions not provided will be set to zero.

ACC indicates that an acceleration spectrum is defined.

(SCALE f4) = Optional Scale factor by which design horizontal acceleration spectrum will be multiplied. (Displacement spectra are not available with the IBC 2006 method.)

Source of damping input:-

DAMP f5 = indicates to use the f5 value for all modes. Default value is 0.05 (5% damping if 0 entered).

MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.

CDAMP is the composite damping of the structure calculated for each mode. One must specify damping for different materials under CONSTANT specification.

LIN, LOG. The interpolation method of the spectrum curves for determining the spectrum value for modes intermediate points. LIN is default option if not specified.

(MIS f6) = Optional parameter to use 'Missing Mass' method. The static effect of the masses not represented in the modes is included. The spectral acceleration for this missing mass mode is the f6 value entered in length/sec2 (this value is not multiplied by SCALE). If f6 is zero, then the spectral acceleration at the ZPA f7 frequency is used. If f7 is zero or not entered, the spectral acceleration at 33Hz (**Zero Period Acceleration, ZPA**) is used. The results of this calculation are SRSSed with the modal combination results. If the MIS parameter is entered on any spectrum case it will be used for all spectrum cases.
(ZPA f7) = Optional parameter for use with MIS option only. Defaults to 33 Hz if f7 is zero. The value is printed but not used if the parameter MIS f6 is entered.

(Save) = Optional parameter to save the results in a file (file name with ".ACC" extension) containing the joint accelerations in g's and radians/sec^2)

ZIP f8 = The zip code of the site location to determine the latitude and longitude and consequently the Ss and S1 factors. (ASCE 7-05 Chapter 22)

LAT f9 = The latitude of the site used with the longitude to determine the Ss and S1 factors. (ASCE 7-05 Chapter 22)

LONG f10 = The longitude of the site used with the latitude to determine the Ss and S1 factors. (ASCE 7-05 Chapter 22)

SS f11 = Mapped MCE for 0.2s spectral response acceleration. (IBC 2006 Clause 1613.5.1, ASCE 7-05 Clause 11.4.1)

S1 f12= Mapped spectral acceleration for a 1-second period. (IBC 2006 Clause 1613.5.1, ASCE 7-05 Clause 11.4.1)

SITE CLASS f29 = Enter A through F as defined in the IBC code. (IBC 2006 Clause 1613.5.2, ASCE 7-05 Section 20.3)

FA f13 = Optional Short-Period site coefficient at 0.2s. Value must be provided if SCLASS set to F (i.e. 6). (IBC 2006 Clause 1613.5.3, ASCE 7-05 Section 11.4.3)

FV f14 = Optional Long-Period site coefficient at 1.0s. Value must be provided if SCLASS set to F (i.e. 6). (IBC 2006 Clause 1613.5.3, ASCE 7-05 Section 11.4.3)

TL f15 = Long-Period transition period in seconds. (ASCE 7-05 Clause 11.4.5 and Chapter 22)

5.32.10.2 Application of Time Varying Load for Response History Analysis

Purpose

This set of commands may be used to model Time History loading on the structure for Response Time History analysis. Nodal time histories and ground motion time histories may both be provided under one load case.

General format:

See Sections 1.18.3, and 5.31.4

```
TIME LOAD
    joint list  { FX | FY | FZ | MX | MY | MZ }   It  Ia  f1

GROUND MOTION  { X | Y | Z }   It  Ia  f1
```

Where I_t = sequential position in the input data of type number of time varying load. To refer to first type number entered, use a 1 here regardless of actual type number entered. Ground Motion must have an Acceleration Type; Time Load forces must have a Force type; and Time Load moments must have a Moment Type (see Section 5.31.4).

I_a = arrival time number (see Section 5.31.4) (integer). This is the sequential number of the arrival time in the list explained in section 5.31.4. Thus the arrival time number of a_3 is 3 and of a_n is n.

f_1 = The Force, Moment, or Acceleration Amplitude at this joint and direction will be multiplied by this factor (default = 1.0). [For accelerations, if the amplitude-time curve was in g's, please use the Scale Factor in the Define Time

History command to convert g's to the acceleration units used in that command. This is recommended due to possible unit changes between that command and this command.]

Multiple loads at a joint-direction pair for a particular (l_t l_a) pair will be summed. However there can only be one (l_t l_a) pair associated with a particular joint-direction pair, the first such entry will be used. Loads at slave joint directions will be moved to the master without moment generation.

Either TIME LOAD or GROUND MOTION or both may be specified under one load case. More than one load case for time history analysis is not permitted.

For TIME LOAD data, multiple direction specifiers can be in one entry as follows (the direction specifiers must be on one line and missing values are assumed to be 1):

```
TIME LOAD
2  3  FX  1  FZ  1  4  -2.1  MX  2  2
6  7  FX  FY  FZ
```

Example

```
LOAD 1
SELFWEIGHT X  1.0
SELFWEIGHT Y  1.0
SELFWEIGHT Z  1.0
MEMBER LOADS
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GZ 7.5 10.0
TIME LOAD
2  3  FX  1  3
5  7  FX  1  6
GROUND MOTION  X  2  1
```

In the above example, the permanent masses in the structure are provided in the form of "selfweight" and "member loads" (see sections 5.32 and 1.18.3) for obtaining the mode shapes and frequencies. The rest of the data is the input for application of the time varying loads on the structure. Forcing function type 1 is applied at joints 2 and 3 starting at arrival time number 3. (Arrival time number 3 is 1.8 seconds in example shown in section 5.31.4). Similarly, forcing function type 1 is applied at joints 5 and 7 starting at arrival time number 6 (4.4 seconds). A ground motion (type 2) acts on the structure in the x-direction starting at arrival time number 1 (0.0 seconds).

5.32.11 Repeat Load Specification

Purpose

This command is used to create a primary load case using combinations of previously defined primary load cases.

General format:

REPEAT LOAD

$i_1, f_1, i_2, f_2 \ldots i_n, f_n$

where,

$i_1, i_2 \ldots i_n$ = primary load case numbers

$f_1, f_2 \ldots f_n$ = corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. Limit of 550 prior cases may be factored. Prior cases to be factored may also contain the REPEAT LOAD command.

Description

This command may be used to create a primary load case using combinations of previously defined primary load case(s). The REPEAT load differs from the load COMBINATION command (Section 5.35) in two ways:

1) A REPEAT LOAD is treated as a new primary load. Therefore, a P-Delta analysis will reflect correct secondary effects. (LOAD COMBINATIONS, on the other hand, algebraically combine the results such as displacements, member forces, reactions and stresses of previously defined primary load cases which were evaluated independently).

2) In addition to previously defined primary loads, the user can also add new loading conditions within a load case in which

the REPEAT LOAD is used.

3) The REPEAT LOAD option is available with load cases with JOINT LOADS and MEMBER LOADS. It can also be used on load cases with ELEMENT PRESSURE loads and FIXED END LOADS.

   Modal dynamic analysis load cases (Response Spectrum, Time History, Steady State) should not be used in REPEAT LOAD. It is also not available for loads generated using some of the program's load generation facilities such as MOVING LOAD Generation. However load cases with WIND LOAD may be used in Repeat Load.

   UBC cases may only be used in REPEAT LOAD if there is a PERFORM ANALYSIS and CHANGE command after each UBC case. See notes with UBC LOAD command.

   Prestress on a given member from 2 or more load cases cannot be combined.

Example

```
LOAD 1 DL + LL
SELFWEIGHT Y -1.4
MEMBER LOAD
1 TO 7 UNIFORM Y -3.5
LOAD 2 DL + LL + WL
REPEAT LOAD
1  1.10
```

4) For a load case that is defined using the REPEAT LOAD attribute, the constituent load cases themselves can also be REPEAT LOAD cases. See load case 4 below.

```
LOAD 1
SELFWEIGHT Y –1.0
LOAD 2
MEMBER LOAD
2 UNI GY –1.5
LOAD 3
REPEAT LOAD
1 1.5
LOAD 4
REPEAT LOAD
2 1.2 3 1.25
```

5.32.12 Generation of Loads

Purpose

This command is used to generate Moving Loads, UBC Seismic loads and Wind Loads using previously specified load definitions.

Primary load cases may be generated using previously defined load systems. The following sections describe generation of moving loads, UBC seismic loads and Wind Loads.

Generation of Moving Loads

See Sections 1.17 and 5.31.1

Pre-defined moving load system types may be used to generate the desired number of primary load cases, each representing a particular position of the moving load system on the structure. This procedure will simulate the movement of a vehicle in a specified direction on a specified plane on the structure.

General format:

LOAD GENERATION n (ADD LOAD i)

$$\text{TYPE}\ \ j\ x_1\ y_1\ z_1\ {}^{*}\left\{\begin{array}{ll}\text{XINC} & f_1\\ \text{YINC} & f_2\\ \text{ZINC} & f_3\end{array}\right\}\ \left(\left\{\begin{array}{l}\text{YRANGE}\\ \text{ZRANGE}\end{array}\right\} r\right)$$

where,

n = total no. of primary load cases to be generated.

i = load case no. for the previously defined load case to be added to the generated loads.

j = type no. of previously defined load system.

x_1, y_1, z_1 = x, y and z coordinates (global) of the initial position of the reference wheel.

f_1, f_2 f_3 = x, y or z (global) increments of position of load system to be used for generation of subsequent load cases. Use only XINC & ZINC if Y up; Use only XINC & YINC if Z up.

r = (Optional) defines section of the structure along global vertical direction to carry moving load. This r value is added and subtracted to the reference vertical coordinate (y_1 or z_1)in the global vertical direction to form a range. The moving load will be externally distributed among all members within the vertical range thus generated. **r** always should be a positive number. In other words, the program always looks for members lying in the range Y_1 and Y_1+ABS(r) or Z_1 and Z_1+ABS(r). The default r value is very small, so entering r is recommended.

The ADD LOAD specification may be used to add a previously defined load case to all the load cases generated by the LOAD GENERATION command. In the example below, the SELFWEIGHT specified in load case 1 is added to all the generated load cases.

Sequential load case numbers will be assigned to the series of generated primary load cases. Numbering will begin at one plus the highest previous load case number. Allow for these when specifying load cases after load case generation.

Notes

1. Primary load cases can be generated from Moving Load systems for frame members only. This feature does not work on finite elements.

2. This facility works best when the roadway, as well as the movement of the vehicle are along one of the global horizontal (X or Z) or (X or Y) directions. For bridge decks which are skewed with respect to the global axes, the load generation may not yield the most satisfactory results. In such cases, the STAAD.Beava program, which is an add-on module to STAAD.Pro, is recommended. That program works on the influence line/influence surface method, and is considerably superior to the moving load generator described in this section. It also has the advantage of being able to calculate the critical load positions on decks

modelled using plate elements, something which this facility cannot at present.

3. The x_1, y_1, z_1 values of the starting position of the reference wheel must be provided bearing in mind that the reference wheel has to be at the elevation of the deck. An improper set of values of these parameters may result in the wheels being positioned incorrectly, and consequently, no load may be generated at all.

Example

```
LOAD 1 DL ONLY
SELFWEIGHT
LOAD GENERATION 20 ADD LOAD 1
TYPE  1 0.  5.  10.  XI  10.
TYPE  2  0.  10.  10.  ZI  15.
LOAD  22  LIVE  LOAD  ON  PAVEMENT
MEMB  LOAD
10  TO  20  30  TO  40  UNI  GY  -5.0
LOAD  COMBINATION  31  10  0.75  22  0.75
PERFORM ANALYSIS
```

Generation of Seismic Loads per UBC, IBC and other codes

See Sections 1.17.2 and 5.31.2

Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per the relevant code specifications. The following general format should be used to generate loads in a particular direction.

General Format:

LOAD i

$$\textbf{UBC LOAD} \begin{Bmatrix} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{Bmatrix} \textbf{(f1)} \quad \textbf{(ACC} \quad \textbf{f2)}$$

Where UBC may be replaced by: IBC , 1893 , AIJ , COL, CFE, NTC or RPA

where i = load case number

f1 = factor to be used to multiply the UBC Load (default = 1.0). May be negative.

f2 = factor to be used to multiply the UBC, IBC, 1893, etc. Accidental torsion load (default = 1.0). May be negative.

Use only horizontal directions.

Example

```
DEFINE UBC LOAD
ZONE 0.2 K 1.0  I  1.5 TS 0.5
SELFWEIGHT
JOINT WEIGHT
1 TO 100 WEIGHT 5.0
101 TO 200 WEIGHT 7.5
LOAD 1 UBC IN X-DIRECTION
UBC LOAD X
JOINT LOAD
5 25 30 FY -17.5
PERFORM ANALYSIS
CHANGE
LOAD 2 UBC IN Z-DIRECTION
UBC LOAD Z
PERFORM ANALYSIS
CHANGE
LOAD 3 DEAD LOAD
SELFWEIGHT
LOAD COMBINATION 4
1  0.75  2  0.75  3  1.0
```

In the above example, notice that the first two load cases are UBC load cases. They are specified before any other load cases.

Notes

1) The UBC load cases should be provided as the first set of load cases. Non-UBC primary load case specified before a

UBC load case is not acceptable. Additional loads such as MEMBER LOADS and JOINT LOADS may be specified along with the UBC load under the same load case.

2) If the UBC cases are to be factored later in a Repeat Load command; or if the UBC case is to be used in a tension/compression analysis; or if Re-analysis (2 analysis commands without a CHANGE or new load case in between); then each UBC case should be followed by PERFORM ANALYSIS then CHANGE commands as shown in the example above. Otherwise the PERFORM ANALYSIS then CHANGE can be omitted. Using the CHANGE command will require the SET NL command to define the maximum number of load cases being entered in the analysis. Also LOAD LIST ALL should be entered after the last PERFORM ANALYSIS command.

3) Up to 8 UBC cases may be entered.

Incorrect usage

```
LOAD 1
SELFWEIGHT Y -1
LOAD 2
JOINT LOAD
3 FX 45
LOAD 3
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
LOAD 4
UBC LOAD Z 1.2
MEMBER LOAD
3 UNI GY -4.5
PERFORM ANALYSIS
```

The error here is that the UBC cases appear as the 3rd and 4th cases, when they ought to be the 1st and 2nd cases.

Correct usage

```
SET NL 10
LOAD 1
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
PERFORM ANALYSIS
CHANGE
LOAD 2
UBC LOAD Z 1.2
MEMBER LOAD
3 UNI GY -4.5
PERFORM ANALYSIS
CHANGE
LOAD 3
SELFWEIGHT Y -1
LOAD 4
JOINT LOAD
3 FX 45
PERFORM ANALYSIS
LOAD LIST ALL
```

Incorrect usage

```
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
```

The error here is that the CHANGE command is missing before load case 2.

Correct usage

```
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
CHANGE
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y –1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
CHANGE
```

4) REPEAT LOAD specification cannot be used for load cases involving UBC load generation unless each UBC case is followed by an analysis command then CHANGE. For example,

Correct usage

```
LOAD 1
UBC LOAD X 1.0
PERFORM ANALYSIS
CHANGE
LOAD 2
SELFWEIGHT Y -1
PDELTA ANALYSIS
CHANGE
LOAD 3
REPEAT LOAD
1 1.4 2 1.2
PDELTA ANALYSIS
```

5) If UBC load generation is performed for the X and the Z (or Y if Z up) directions, the command for the X direction must precede the command for the Z (or Y if Z up) direction.

Incorrect usage

```
LOAD 1
UBC LOAD Z 1.2
SELFWEIGHT Y- 1
LOAD 2
UBC LOAD X 1.2
SELFWEIGHT Y -1
PDELTA ANALYSIS
```

Incorrect because UBC for Z direction precedes UBC for X direction

Correct usage

```
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y -1
PDELTA ANALYSIS
```

Generation of IS:1893 Seismic Load

See Sections 1.17.2 and 5.31.2.5

The following general format should be used to generate the IS 1893 load in a particular direction.

General Format:

LOAD i

$$\textbf{1893 LOAD} \quad \left\{ \begin{array}{c} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{array} \right\} \quad \textbf{(f)}$$

where i = load case number
f = factor to be used to multiply the 1893 Load (default = 1.0)

Use only horizontal directions.

Example

```
DEFINE 1893 LOAD
ZONE 0.05 RF 1.0  I 1.5 SS 1.0
SELFWEIGHT
JOINT WEIGHT
7 TO 12 WEIGHT 17.5
13 TO 30 WEIGHT 18.0
MEMBER WEIGHT
1 TO 20 UNI 2.0
LOAD 1 1893 LOAD IN X-DIRECTION
1893 LOAD X
JOINT LOAD
5 25 30 FY -17.5
LOAD 2 1893 LOAD IN Z-DIRECTION
1893 LOAD Z
LOAD 3 DEAD LOAD
SELFWEIGHT
LOAD COMBINATION 4
1  0.75  2  0.75  3  1.0
```

In the above example, the first two load cases are the 1893 load cases. They are specified before any other load case.

Generation of Wind Load

The built-in wind load generation facility can be used to calculate the wind loads based on the parameters defined in Section 5.31.3. The following general format should be used to perform the wind load generation. See section 1.17.3 for the two types of structures on which the load can be generated. For closed type structures, the panel areas bounded by beam members only (and ground), and exposed to the wind, are used to define loaded areas (plates and solids are ignored). The loads generated are applied only at the joints at vertices of the bounded areas. For open type structures also, generation is done considering only the members in the model.

General Format:

```
        LOAD  i
                     ⎧ X  ⎫                   ⎧ XR  f1, f2        ⎫
WIND  LOAD           ⎨ Y  ⎬ (f) TYPE j (OPEN) ⎨ YR  f1, f2        ⎬
                     ⎩ Z  ⎭                   ⎪ ZR  f1, f2        ⎪
                                              ⎪ LIST memb-list    ⎪
                                              ⎩ ALL               ⎭
```

Where

i Load case number

See Sections 1.17.3 and 5.31.3

X, -X, Z or -Z, Y or -Y Direction of wind in global axis system. Use horizontal directions only.

j Type number of previously defined systems

f The factor to be used to multiply the wind loads. Negative signs may be used to indicate opposite direction of resulting load (default=1.0)

f_1 and f_2 global coordinate values to specify X or Y or Z range for member selection.

OPEN optional word to be used if loading is to be generated on open-type of structures. If this not specified, load will be generated assuming the panels are "closed".

X, -X, Z or –Z [A minus sign indicates that suction occurs on the other side of the selected structure.] and the f factor. If all of the members are selected and X (or Z) is used and the factor is positive, then the exposed surfaces facing in the –x (or –z) direction will be loaded in the positive x (or z) direction (normal wind in positive direction). See diagrams that follow. If X and a negative factor is used, then the exposed surfaces facing in the +x direction will be loaded in the negative x direction (normal wind in negative direction). [If –X is entered and a negative factor, then the exposed surfaces facing in the -x direction will be loaded in the negative x direction (suction). If –X is entered and a positive factor, then the exposed surfaces facing in the +x direction will be loaded in the positive x direction (suction).]

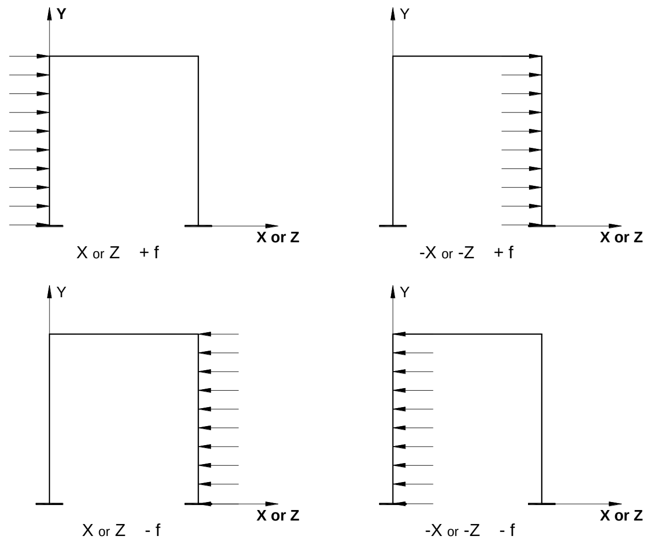


Figure 5.38

A member list or a range of coordinate values (in global system) may be used. All members which have both end coordinates within the range are assumed to be candidates (for closed type structures) for defining a surface which may be loaded if the surface is exposed to the wind. The loading will be in the form of joint loads (not member loads). 1, 2 or 3 ranges can be entered to form a “layer”, “tube” or “box” for selecting members in the combined ranges. Use ranges to speed up the calculations on larger models.

Example

```
DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1  0.12 HEIGHT 100  200
EXP  0.6  JOI  1  TO  25  BY  7  29  TO  37  BY  4  22  23
TYPE  2
INT  0.1  0.12 HEIGHT 100  900
EXP 0.3 YR 0 500
LOAD 1
SELF Y -1.0
LOAD 2
```

```
WIND LOAD Z 1.2 TYPE 2 ZR 10 11
LOAD 3
WIND LOAD X TYPE 1 XR 7 8 ZR 14 16
LOAD 4 SUCTION ON LEEWARD SIDE
WIND LOAD -X 1.2 LIST 21 22 42
```

Example for open structures

```
LOAD 1 WIND LOAD IN Z DIRECTION
WIND LOAD 2 -1.2 TYPE 1 OPEN
```

Notes

1. For closed type structures, panels or closed surfaces are generated by the program based on the members in the ranges specified and their end joints. The area within each closed surface is determined and the share of this area (influence area) for each node in the list is then calculated. The individual bounded areas must be planar surfaces, to a close tolerance, or they will not be loaded.

   Hence, one should make sure that the members/joints that are exposed to the wind make up a closed surface (ground may form an edge of the closed surface). Without a proper closed surface, the area calculated for the region may be indeterminate and the joint force values may be erroneous. Consequently, the number of exposed joints should be at least three.

2. Plates and solids are not considered for wind load generation. On such entities, wind must be applied using pressure loading facilities for plates and solids.

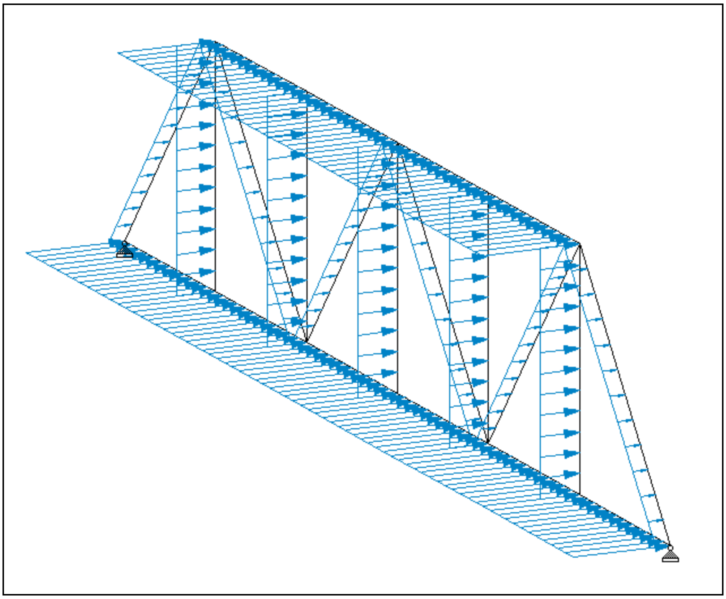


Figure 5.39 - Load diagram for wind on open structures

5.32.13 Generation of Snow Loads

Purpose

This command is used to generate Snow Loads using previously specified Snow load definitions. This input should be a part of a load case.

General format:

See Sections 1.17.4 and 5.31.5

SNOW LOAD

_flr_grp TYPE j CS f_1 { BALA | UNBA } { OBST | UNOB } { MONO | HIP | GABLE }

where,

_flr_grp = The members that form the roof and that are to be loaded by snow load must be listed in a floor group. See section 5.16.

j = type no. of previously defined snow load system.

f_1 = Roof Slope factor (CS). Default = 0.0. For sloped roofs, the roof slope factor is described in section 7.4 of the SEI/ASCE-7-02. A value of 0 indicates that the roof is horizontal.

BALA or UNBA = Balanced or unbalanced snow load. Default is balanced. These terms are described in section 7.6, and figures 7.3 and 7.5 of ASCE 7-02.

OBST or UNOB = Obstructed or unobstructed. Default is unobstructed.

MONO or HIP or GABLE= Roof type. Default is MONO.

- Mono (see mono-sloped roof shown in figure 6-6 of the ASCE 7-02 code)
- Hipped (see figures 6-3 and 6-6 of the ASCE 7-02 code)
- Gable (see figures 6-3 and 6-6 of the ASCE 7-02 code)

Use as many floor groups and types as necessary in each load case.

Example

```
LOAD 11 LOADTYPE Snow TITLE LOAD CASE 11
SNOWLOAD
_ROOFSNOW BALANCED TYPE 1 GABLE UNOBSTRUCTED CS 1
```

5.33 Reference Load Cases – Application

Purpose

Reference Load types are described in section 5.31.6 of this manual. This section shows how one can call the data specified under those types in actual load cases.

General Format

The format of a reference to a Reference Load in a primary load (j)case is thus:-

LOAD (j) LOADTYPE (type) any_title_you_choose
REFERENCE LOAD
R(i) 1.0
…

Example

```
LOAD 1 LOADTYPE None  TITLE D+L
REFERENCE LOAD
R1 1.0 R2 1.0
LOAD 2 LOADTYPE None  TITLE DEAD+SNOW
REFERENCE LOAD
R1 1.0 R3 1.0
LOAD 3 LOADTYPE None  TITLE D+H
REFERENCE LOAD
R1 1.0 R4 1.0
ELEMENT LOAD
1212 1267 TRAP GY JT -0.54 -0.44 -0.44 -0.54.
```

5.34 Frequency Calculation

Purpose

There are two methods available in STAAD for calculating the frequencies of a structure – 1) an approximate method called the Rayleigh method and 2) a more exact method which involves the solution of an eigenvalue problem.

Both methods are explained in the following sections.

Rayleigh Method – Section 5.34.1
Modal Calculation – Section 5.34.2

5.34.1 Rayleigh Frequency Calculation

Purpose

This command may be used to calculate the Rayleigh method approximate frequency of the structure for vibration corresponding to the general direction of deflection generated by the load case that precedes this command. Thus, this command typically follows a load case.

General format:

CALCULATE RAYLEIGH (FREQUENCY)

Description

See Section 1.18.3

This command is specified after all other load specifications of any primary load case for which the Rayleigh frequency is calculated. This Rayleigh frequency calculation is based on the Rayleigh iteration method using 1 iteration. If a more accurate, full-scale eigensolution is required, the MODAL CALCULATION command (see next section) may be used. A full eigensolution is automatically performed if a RESPONSE SPECTRUM, TIME HISTORY or STEADY STATE is specified in any load case.

Example

```
LOADING 1 DEAD AND LIVE LOAD
AREA LOAD
1 TO 23 ALOAD -200.0
CALCULATE RAYLEIGH FREQ
LOADING 2 WIND LOAD
```

In this example, the Rayleigh frequency for load case 1 will be calculated. The output will produce the value of the frequency in cycles per second (cps), the maximum deflection along with the global direction and the joint number where it occurs.

Since the AREA LOAD is in the global Y direction, the displaced shape to be used as the mode will be in the Y direction. If you want a mode or frequency to be for lateral motion (X or Z), then enter the loads in the X or Z directions. Entering loads in the vertical direction when the first mode is in the lateral direction is a common mistake.

Notes

This command is based on the Rayleigh method of iteration using 1 iteration. The frequency calculated estimates the frequency as if the structure were constrained to vibrate in the static deflected shape generated by the loads in the load case.

In many instances, the forces should be in one global direction to get the mode and frequency associated with that direction.

5.34.2 Modal Calculation Command

Purpose

This command may be used to obtain a full scale eigensolution to calculate relevant frequencies and mode shapes. It should not be entered if this case or any other case is a TIME LOAD or RESPONSE SPECTRUM case. For Steady State/Harmonic analysis this command must be included in the load case that defines the weights and weight moment of inertias for eigensolutions (see sections 1.18.3 and 5.32).

General Format:

MODAL (CALCULATION REQUESTED)

This command is typically used in a load case after all loads are specified. The loads will be treated as weights and weight moment of inertias for eigensolutions (see sections 1.18.3 and 5.32). The user is advised to specify the loads keeping this in mind.

This case will be independently solved statically and dynamically. Static results using the loads will include joint displacements, member forces, support reactions, and other outputs computed from a normal static analysis without any dynamic effects included.

In addition, the Dynamic results (using loads as masses) will include Mode Shapes and Frequencies.

See Sections 5.31, 5.32+ 1.18.3

Notes

The MODAL CALCULATION command can be included in any of the primary load cases, but only in one of them.

5.35 Load Combination Specification

Purpose

This command may be used to combine the results of the analysis. The combination may be algebraic, SRSS, a combination of both, or ABSolute.

General format:

LOAD COMBINATION $\left\{ \begin{array}{l} \textbf{SRSS} \\ \textbf{ABS} \end{array} \right\}$ **i** $\mathbf{a_1}$

$\mathbf{i_1, f_1, i_2, f_2 \ldots \quad (f_{srss})}$

i = Load combination number (any integer smaller than 100000 that is not the same as any previously defined primary load case number.)

a_1 = Any title for the load combination.

i_1, i_2 ... represents the load case numbers which are to be combined.

f_1, f_2 ... represents corresponding factors to be applied to loadings. A value of 0.0 is also allowed.

f_{SRSS} = optional factor to be applied as a multiplying factor on the combined result of the SRSS load combination (see examples below).

If the last character on a line is a hyphen, then the command is continued on the next line. A limit of 550 prior cases may be factored in one command.

Notes

1) In the LOAD COMBINATION SRSS option, if the minus sign precedes any load case no., then that load case will be combined algebraically with the SRSS combination of the rest.

2) The total number of primary and combination load cases combined cannot exceed the limit described in section 5.2 of this manual.

3) A value of zero (0) as a load factor is permitted. See Notes item (3) later in this section for more details..

Description

LOAD COMBINATION
Results from analyses will be combined algebraically.

```
LOAD COMBINATION 6 DL+LL+WL
1 0.75 2 0.75 3 1.33
```

LOAD COMBINATION ABS
Absolute value of results from the analyses will be combined.

```
LOAD COMBINATION ABS 7 DL+LL+WL
1 0.85 2 0.65 3 2.12
```

LOAD COMBINATION SRSS
Results from analyses may be combined both algebraically and using the SRSS (Square Root of Summation of Squares) method. The combination scheme may be mixed if required. For example, in the same load combination case, results from load cases may be combined in the SRSS manner and then combined algebraically with other load cases. Note that the case factor is not squared.

If some of the factors are negative, then the sum of the factored squares may become negative. If negative, the SQRT will be of the absolute value and the result of the SQRT will be set negative.

Refer to the following examples for illustration -

Example

Several combination examples are provided to illustrate the possible combination schemes -

Simple SRSS Combination

```
LOAD COMBINATION SRSS 8 DL+SEISMIC
1  1.0  2  -0.4  3  0.4
```

This (LOAD COMBINATION SRSS 8) illustrates a pure SRSS load combination with a default SRSS factor of 1. The following combination scheme will be used -

$$v= 1.0\sqrt{1 \times L1^2 - 0.4 \times L2^2 + 0.4 \times L3^2}$$

where v = the combined value and L1 - L3 = values from load cases 1,2 and 3. Since an SRSS factor is not provided, the default value of 1.0 is being used.

Algebraic & SRSS Combination in the Same Load Combination Case

Example 1

```
LOAD COMBINATION SRSS 9
-1 0.75 2 1.3 3 2.42 0.75
```

The combination formula will be as follows -

$$v = 0.75 \times L1 + 0.75\sqrt{1.3 \times L2^2 + 2.42 \times L3^2}$$

where v = combined value
L2 & L3 = values from load cases 2 & 3.

In the above specification, a minus sign precedes load case 1. Thus, Load 1 is combined algebraically with the result obtained from combining load cases 2 and 3 in the SRSS manner. The SRSS factor of 0.75 is applied on the SRSS combination of 2 and 3.

Example 2

```
LOAD COMBINATION SRSS 10
-1 0.75 -2 0.572 3 1.2 4 1.7 0.63
```

Here, both load cases 1 and 2 are combined algebraically with the SRSS combination of load cases 3 and 4. Note the SRSS factor of 0.63. The combination formula will be as follows.

$$v = 0.75 \times L1 + 0.572 \times L2 + 0.63 \sqrt{1.2 \times L3^2 + 1.7 \times L4^2}$$

Notes

1) This option combines the results of the analysis in the specified manner. It does not analyze the structure for the combined loading.
2) If the secondary effects of combined load cases are to be obtained through a PDELTA , Member/Spring Tension/Compression, Multi-linear Spring, or Nonlinear Analysis, then the LOAD COMBINATION command is inappropriate for the purpose. See the REPEAT LOAD command (section 5.32.11) for details.
3) In a load combination specification, a value of 0 (zero) as a load factor is permitted. In other words, a specification such as

```
LOAD COMB 7
1 1.35 2 0.0 3 1.2 4 0.0 5 1.7
```

is permitted. So, the above is the same as

```
LOAD COMB 7
1 1.35 3 1.2 5 1.7
```

4) All combination load cases must be provided immediately after the last primary load case.
5) The maximum number of load cases that can be combined using a LOAD COMBINATION command is 550.

5.36 Calculation of Problem Statistics

Removed. Please contact the Technical Support division for more details.

5.37 Analysis Specification

Purpose

STAAD analysis options include linear static analysis, P-Delta (or second order analysis), and several types of Dynamic analysis.

This command is used to specify the analysis request. In addition, this command may be used to request that various analysis related data, like load info, statics check info, etc. be printed.

5.37.1 Linear Elastic Analysis

General format:

```
                                                 ⎧ LOAD DATA     ⎫
⎧                      ⎫                         ⎪ STATICS CHECK ⎪
⎨ PERFORM              ⎬ ANALYSIS (PRINT         ⎨ STATICS LOAD  ⎬ )
⎩ PERform IMPerfection ⎭                         ⎪ BOTH          ⎪
                                                 ⎩ ALL           ⎭
```

Without one of these analysis commands, no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

If the PRINT LOAD DATA command is specified, the program will print an interpretation of all the load data.

PRINT STATICS CHECK will provide a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.

PRINT STATICS LOAD prints everything that PRINT STATICS CHECK does, plus it prints a summation of all internal and external forces at each joint (generates voluminous output).
Use the PRINT MODE SHAPES command separately after this command if mode shapes are desired.

PRINT BOTH is equivalent to PRINT LOAD DATA plus PRINT STATICS CHECK.

PRINT ALL is equivalent to PRINT LOAD DATA plus PRINT STATICS LOAD.

Notes for PERFORM ANALYSIS

See Section 1.18

This command directs the program to perform the analysis that includes:

a) Checking whether all information is provided for the analysis;
b) Forming the joint stiffness matrix;
c) Checking the stability of the structure;
d) Solving simultaneous equations, and
e) Computing the member forces and displacements.
f) If a RESPONSE SPECTRUM or TIME LOAD is specified within a load case or the MODAL CALCULATION command is used, a dynamic analysis is performed.

Notes for Imperfection Analysis

This performs a modified linear elastic analysis using Member Imperfection Specifications (see section 5.26.6) defined on beam and column members.

An Imperfection analysis will reflect the secondary effects only if the camber and/or drift is specified in a DEFINE IMPERFECTIONS specification (Section 5.26.6). For combination of load cases, with imperfection, use the Repeat Load specification rather than the Load Combination.

General Comments

STAAD allows multiple analyses in the same run. Multiple analyses may be used for the following purposes:

1) Successive analysis and design cycles in the same run result in optimized design. STAAD automatically updates changes in member cross-sectional sizes. Thus the entire process is automated.

   Refer to Example 1 in the Getting Started & Examples manual for detailed illustration.

2) Multiple analyses may be used for load-dependent structures. For example, structures with bracing members are analyzed in

several steps. The bracing members are assumed to take Tension load only. Thus, they need to be activated and inactivated based on the direction of lateral loading.

The entire process can be modeled in one STAAD run using multiple PERFORM ANALYSIS commands. STAAD is capable of performing a design based on the load combinations provided.

Refer to Example 4 in the Getting Started & Examples manual for detailed illustration.

3) The user may also use Multiple Analyses to model change in other characteristics like SUPPORTS, RELEASES, SECTION PROPERTIES etc.

4) Multiple Analyses may require use of additional commands like the SET NL command and the CHANGE command.

5) Analysis and CHANGE are required after UBC cases if the case is subsequently referred to in a Repeat Load command or if the UBC case will be re-solved after a Select command or after a Multiple analysis.

5.37.2 PDELTA Analysis options

General format:

Option 1

P-Delta analysis with Large Delta effects only

PDELTA (n) ANALYSIS (CONVERGE (m)) (print options)

where n = no. of iterations desired (default value of n = 1).

Option 2

P-Delta analysis including small delta effect (effective from STAAD.Pro 2007 Build 01).

PDELTA (n) ANALYSIS SMALLDELTA (print options)

Option 3

P-Delta analysis including stress stiffening effect of the KG matrix (effective from STAAD.Pro 2007 Build 01).

PDELTA KG ANALYSIS (print options)

Print options: PRINT LOAD DATA, PRINT STATICS CHECK, PRINT STATICS LOAD, PRINT BOTH, PRINT ALL.

If the PRINT LOAD DATA command is specified, the program will print an interpretation of all the load data.

PRINT STATICS CHECK will provide a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.

PRINT STATICS LOAD prints everything that PRINT STATICS CHECK does, plus it prints a summation of all internal and external forces at each joint (generates voluminous output). Use the PRINT MODE SHAPES command separately after this command if mode shapes are desired.

PRINT BOTH is equivalent to PRINT LOAD DATA plus PRINT STATICS CHECK.

PRINT ALL is equivalent to PRINT LOAD DATA plus PRINT STATICS LOAD.

Without one of these analysis commands, no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

A PDELTA ANALYSIS will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification (Section 5.32.11). Secondary effects will not be evaluated correctly for LOAD COMBINATIONS.

PDELTA effects are computed for frame members and plate elements only. They are not calculated for solid elements.

Notes for option 1

This command directs the program to perform the analysis that includes:

a) Checking whether all information is provided for the analysis;
b) Forming the joint stiffness matrix
c) Checking the stability of the structure;
d) Solving simultaneous equations, and
e) Computing the member forces and displacements.
f) For P-Delta analysis, forces and displacements are recalculated, taking into consideration the P-Delta effect.
g) If a RESPONSE SPECTRUM or TIME LOAD is specified within a load case or the MODAL CALCULATION command is used, a dynamic analysis is performed.

h) In each of the "n" iterations of the PDELTA analysis, the load vector will be modified to include the secondary effect generated by the displacements caused by the previous analysis.

The default procedure is based on “P-large Delta” effects only. Enter the SmallDelta parameter to include the “P-small Delta” effects as well (see option 2 above)

This PDELTA ANALYSIS command should specify 5 to 25 iterations to properly incorporate the PDELTA effect. With this many iterations, the PDELTA (n) Analysis command results are as good as the PDELTA KG command results (see option 3). The advantage of this PDELTA (n) Analysis command comes from not having to re-form and then triangular factorize the stiffness matrix for every iteration within every case. Also this command allows tension/compression.

Be aware that global buckling can occur in PDELTA analysis, resulting in large or infinite or NaN values for displacement. Do not use the results of such a case. Sometimes the loads from Repeat Load combination cases are too large; sometimes partial moment releases rather than the full release is needed, sometimes connectivity needs to be corrected. Always check the maximum displacements for PDELTA analyses.

When the CONVERGE command is not specified, the member end forces are evaluated by iterating "n" times. The default value of "n" is 1 (one).

When the CONVERGE command is included, the member end forces are evaluated by performing a convergence check on the joint displacements. In each step, the displacements are compared with those of the previous iteration in order to check whether convergence is attained. In case "m" is specified, the analysis will stop after that iteration even if convergence has not been achieved. If convergence is achieved in less than "m" iterations, the analysis is terminated. (DO NOT ENTER "n" when CONVERGE is provided.)

To set convergence the displacement tolerance, enter SET DISPLACEMENT f command. Default is maximum span of the structure divided by 120.

Example

Following are some examples on use of the command for P-Delta analysis as described in option 1.

```
PDELTA ANALYSIS
PDELTA 5 ANALYSIS
PDELTA ANALYSIS CONVERGE
PDELTA ANALYSIS CONVERGE 5
```

STAAD allows multiple p-delta analyses in the same run (see the General Comments section of 5.37.1 for details).

Notes for option 2 (PDELTA SMALL-DELTA)

A regular STAAD P-Delta Analysis can now account for the small P-Delta effect whilst performing a P-Delta analysis.
Without the Small Delta option, i.e. a regular STAAD P-Delta analysis, STAAD performs a first order linear analysis and obtains a set of joint forces, from members/plates based on the large P-Delta effect, which are then added to the original load vector. A second analysis is then performed on this updated load vector.

With the Small Delta option selected, both the large & small P-Delta effects are included in calculating the end forces, (5 to 10 iterations will usually be sufficient).

Example

```
PDELTA 20 ANALYSIS SMALLDELTA PRINT STATICS CHECK
```

Notes for option 3 (PDELTA KG)

The P-Delta analysis capability has been enhanced with the option of including the stress stiffening effect of the Kg matrix into the member / plate stiffness.

A regular STAAD P-Delta Analysis (option 1) performs a first order linear analysis and obtains a set of joint forces from member/plates based on the large P-Delta effect. These forces are added to the original load vector. A second analysis is then performed on this updated load vector (5 to 10 iterations will usually be sufficient).

In the new P-Delta KG Analysis, that is, with the Kg option selected, the effect of the axial stress after the first analysis is used to modify the stiffness of the member/plates. A second analysis is then performed using the original load vector. Large & small P-Delta effects are always included.

See Section 1.18

This command directs the program to perform the analysis that includes:

a) Solving the static case.
b) Re-forming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces.
c) Solving simultaneous equations for displacements;
d) If a RESPONSE SPECTRUM or TIME LOAD is specified within a load case or the MODAL CALCULATION command is used, a dynamic analysis is performed. The static cases solved by this PDELTA KG analysis command will be solved first then the dynamic analysis cases. <u>The stiffness matrix used in the dynamic analysis will be the K+Kg matrix used in the last iteration for the last static case. This is a stress stiffened dynamic analysis, sometimes known as a PDELTA Dynamic analysis.</u>

A PDELTA KG ANALYSIS will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification (Section 5.32.11). Secondary effects will not be evaluated correctly for LOAD COMBINATIONS.

PDELTA KG effects are computed for frame members and plate elements only. They are not calculated for solid elements. The results are based on “P-large & small Delta” effects.

The other PDELTA command [PDELTA (n) Analysis] with 10 or more iterations (option 1 or 2) is preferred unless there are only 1 or 2 load cases.

Tension/compression only are not allowed with this PDELTA KG command; use the PDELTA (n) Analysis command instead.

Be aware that global buckling can occur in PDELTA KG analysis. This condition is usually detected by STAAD. A message is issued and the results for that case are set to zero. STAAD will continue with the next load case.

Global buckling may not be detected which could result in a solution with large or infinite or NaN values for displacement or stability errors. Do not use the results of such cases. Sometimes the loads from Repeat Load combination cases are too large; sometimes partial moment releases rather than the full release is needed, sometimes connectivity needs to be corrected. Always check the maximum displacements for PDELTA analyses.

Example

```
PDELTA KG ANALYSIS PRINT BOTH
```

5.37.3 CABLE ANALYSIS (NON LINEAR)

```
{ PERFORM CABLE }   ANALYSIS   ( { STEPS          f1      } ) (print options)
                                 { EQITERATIONS   f2      }
                                 { EQTOLERANCE    f3      }
                                 { SAGMINIMUM     f4      }
                                 { STABILITY      f5  f6  }
                                 { KSMALL         f7      }
```

See Section 1.11.2 and 1.18.2.7

Print options: PRINT LOAD DATA, PRINT STATICS CHECK, PRINT STATICS LOAD, PRINT BOTH, PRINT ALL. See section 5.37.2 for details.

This command may be continued to the next line by ending with a hyphen.

Steps = Number of load steps. The applied loads will be applied gradually in this many steps. Each step will be iterated to convergence. Default is 145. The f1 value, if entered, should be in the range 5 to 145.

Eq-iterations = Maximum number of iterations permitted in each load step. Default is 300. Should be in the range of 10 to 500.

Eq-tolerance = The convergence tolerance for the above iterations. Default is 0.0001.

Sag minimum= Cables (not trusses) may sag when tension is low. This is accounted for by reducing the E value. Sag minimum may be between 1.0 (no sag E reduction) and 0.0 (full sag E reduction). Default is 0.0. As soon as SAGMIN becomes less than 0.95 the possibility exists that a converged solution will not be achieved without increasing the steps to 145 or the pretension loads. The Eq iterations may need to be 300 or more. The Eq tolerance may need to be greater or smaller.

Stability stiffness = A stiffness matrix value, f5, that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for the first f6 load steps. The amount added linearly decreases with each of the f6 load steps (f6 is 1 if omitted). If f5 entered, use 0.0 to 1000.0. Default is 1.0. This parameter alters the stiffness of the structure.

K small stiffness = A stiffness matrix value, f7, that is added to the global matrix at each translational direction for joints connected to cables and nonlinear trusses for every load step. If entered, use 0.0 to 1.0. Default is 0.0 . This parameter alters the stiffness of the structure.

Notes

1. STAAD allows multiple analyses in the same run (see the General Comments section of 5.37.1).
2. Multiple Analyses may require use of additional commands like the SET NL command and the CHANGE command.
3. Analysis and CHANGE are required between primary cases for PERFORM CABLE ANALYSIS.

5.37.4 BUCKLING ANALYSIS (Available effective STAAD.Pro 2007 Build 01)

General format:

PERFORM BUCKLING ANALYSIS (MAXSTEPS f1) (print options)

See Section 1.18.2.2

Without this command (or any of the analysis commands described earlier), no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

Print options: same as in the previous sections.

There are two procedures available. One comes with the **basic solver** and the other with the **advanced solver**.

BASIC SOLVER: f1 = maximum no. of iterations desired (default value of n = 10). 15 is recommended.

This command directs the program to perform an analysis that includes:

a) Solving the static case.
b) Re-forming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces & inplane plate stresses..
c) Solving simultaneous equations for displacements;
d) Repeat b) and c) for the number of required additional iterations; either until convergence or until MAXSTEPS is reached.

If the loads must be in the opposite direction, STAAD will stop solving that case at 1 iteration. The results for the case will be outputted; then STAAD will continue with the next case.

Convergence occurs when 2 consecutive buckling factors in the iteration are within 0.1% of each other.

Results are based on the highest successful Buckling Factor estimate that was calculated; as if the original applied loads times the buckling factor had been entered.

ADVANCED SOLVER: MAXSTEPS input is ignored.

This command directs the program to perform the analysis that includes:

a) Solving the static case.
b) Re-forming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces & inplane plate stresses..
c) Solving an eigenvalue problem for up to 4 buckling factors and buckling shapes

A Buckling Analysis will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification (Section 5.32.11). Buckling will not be performed for LOAD COMBINATIONS cases.

Buckling Kg matrices are computed for frame members and plate elements only. They are not calculated for solid elements. The results are based on “P-large & small Delta” effects.

Buckling Analysis solves for Buckling Factors. These are the amounts by which the load case must be factored for the buckling shape to occur. Only one load case may precede this command. Only one buckling analysis of this type may be entered per run.

If the loads must be in the opposite direction, STAAD will compute negative buckling factors.

Results are for the normalized buckling shape not as if the original applied loads times the buckling factor had been entered.

Example with ADVANCED SOLVER

```
PERFORM BUCKLING ANALYSIS
```

See Section 1.18

Example with BASIC SOLVER

```
PERFORM BUCKLING ANALYSIS MAXSTEPS 15 –
PRINT LOAD DATA
```

5.37.5 DIRECT ANALYSIS (Available effective STAAD.Pro 2 007 Build 03)

General format:

See Section 1.18

PERFORM DIRECT ANALYSIS ({LRFD or ASD} TAUtol f1 DISPtol f2 ITERdirect i3) (print options)

Print options are same as in earlier sections.

This command directs the program to perform the analysis that includes:

a) Reduce Axial & Flexure stiffness to 80% for selected members.
b) Solving the static case which has notional loads included.
c) Re-forming the global joint stiffness matrix to include the Kg matrix terms which are based on the computed tensile/compressive axial member forces
d) Solving simultaneous equations for displacements;
e) Compute Tau-b of AISC 05 Direct Analysis Appendix 7 based on required strength versus yield strength.
f) Re-forming the global joint stiffness matrix.
g) Solving simultaneous equations for displacements;
h) Repeat steps c) through g) until converged or ITERDIRECT iterations are reached.

A Direct Analysis will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification (Section 5.32.11). Direct analysis will not be performed for LOAD COMBINATIONS cases.

Notional loads must be defined using a DEFINE NOTIONAL table.

A list of members which will have their initial Tau-b value set and/or have their Axial stiffness reduced and/or their flexural stiffness reduced must be entered using a DEFINE DIRECT table.

Convergence occurs when 2 consecutive iterations have all member tau-b values the same within a tolerance, TAUTOL, and displacements & rotations the same within a tolerance, DISPTOL.

LRFD is the default (all generated loads are factored by 1.0). If ASD entered then loads are factored by 1.6 for the Pdelta and Tau-b calculations. Final results are based on the final displacements divided by 1.6
TAUTOL is normally 0.001 to 1.0. Default is 0.01.
DISPTOL should not be too tight. The value is in current length units. Default is 0.01 inch displacement or 0.01 radians.
ITERDIRECT limits the number of iterations, 1 to 10 is usually enough. Default is 1.

Example

```
PERFORM DIRECT ANALYSIS  LRFD  TAUTOL 0.01 -
DISPTOL 0.01  ITERDIRECT 2 -
PRINT LOAD DATA
```

5.37.6 Steady State & Harmonic Analysis

The options available under steady state analysis in STAAD are described in the next few sections.

5.37.6.1 Purpose

This analysis type is used to model steady, harmonically varying load on a structure to solve for the steady harmonic response after the initial transient response has damped out to zero. STAAD Steady State analysis options include results for one forcing frequency or for a set of frequencies. You may specify ground motion or a distributed joint loading in one load case. Damping is required either in this input or from the Modal Damp input or from the Composite Damping input.

This command is used to specify the analysis request, specify that the load case with the MODAL CALC command (which must be prior to this analysis command) be used as the definition of the mass distribution, and to begin a block of data input describing the steady state forcing functions, the output frequencies, and the printing of the joint responses.

All of the input and output frequencies are in Hertz (Hz or CPS).

Related topics can be found in the following sections:

Misc. Settings for Dynamics, Cut off values, and mode selection	-	5.30
Frequency and Mode Extraction	-	5.34
Modal & Composite Damping	-	5.26.4,5
Analysis Specification	-	5.37
Dynamic Analysis overview	-	1.18.3

The Modal Calculation command is required in the weight/mass definition load case. See section 5.34.

General format:

<u>PER</u>FORM <u>STE</u>ADY <u>STA</u>TE <u>ANA</u>LYSIS

This command directs the program to perform the analysis that includes:

a) Checking whether all information is provided for the analysis;
b) Forming the joint stiffness matrix;
c) Solving simultaneous equations;
e) Solving for modes and frequencies;
f) Computing for the steady state joint displacements, velocities & accelerations and phase angles;
g) Computing the above quantities versus frequency and displaying the results graphically.
h) Member & element forces & stresses and support reactions are not currently computed.

The first input after the Perform Steady State Analysis command is:

```
BEGIN  { STEADY   }  { FORCE  }
       { HARMONIC }  { GROUND }
```

Steady or Harmonic

Steady = the analysis is at one forcing frequency.
Harmonic = the analysis is at several frequencies.

Force or Ground

Choose whether the loading is a distributed joint force load or a ground motion.

This command selects which of the 4 load/analysis types, that are available, will be used in this analysis. These four are described in sections 5.37.6.4 through 5.37.6.7.

This block of data should be terminated with the END STEADY command as mentioned in Section 5.37.6.9.

The steady state/harmonic analysis will calculate the maximum displacement and the associated phase angle for each of 6 joint directions, relative to the ground motion, for each frequency defined in the next section.

In PRINT JOINT DISP and in Post processor displayed results, the load case displacement for a given joint and direction will be the maximum value over all of the frequencies (without the phase angles) for a Steady State load case.

In post-processing for harmonic analysis, Log-Log graphs of any joint's relative translational displacement or velocity or acceleration versus frequency may be selected.

See section 5.37.1.8 for printing displacements with phase angles by frequency.

5.37.6.2 Define Harmonic Output Frequencies

If Harmonic is requested above, then optionally include the next input.

FREQUENCY FLO f_1 FHI f_2 NPTS f_3 (MODAL) FLIST freqs

FLO f_1 = Lowest frequency to be included in Harmonic output. Default to half the first natural frequency.

FHI f_2 = Highest frequency to be included in Harmonic output. Default to highest frequency plus largest difference between two consecutive natural frequencies.

NPTS f_3 = Number of plot frequencies to be included between natural frequencies. Defaults to 5 (7 if fewer than 10 modes) (3 if more than 50 modes). These points are added to improve the graphic display of responses versus frequency.

The natural and forcing frequencies are automatically included in the plot frequencies.

MODAL = This option causes the natural frequencies between FLO and FHI to be added to the list of forcing frequencies.

FLIST freqs = List of forcing frequencies to be included in the Harmonic analysis. Continue freqs input to additional lines by ending each line except the last with a hyphen.

Only forcing frequencies will be used to create load case results and print results.

5.37.6.3 Define Load Case Number

Currently the load case number is automatically the case with the MODAL CALC command.

5.37.6.4 Steady Ground Motion Loading

This set of commands may be used to specify steady ground motion loading on the structure, the ground motion frequency, the modal damping, and the phase relationship of ground motions in each of the global directions.

General format:

$$\underline{\text{STE}}\text{ADY}\ \ \underline{\text{GRO}}\text{UND}\ \ \underline{\text{FRE}}\text{Q}\ \ f_1 \left\{ \begin{array}{l} \underline{\text{DAM}}\text{P}\ \ f_2 \\ \underline{\text{CDA}}\text{MP} \\ \underline{\text{MDA}}\text{MP} \end{array} \right\} \quad \left\{ \begin{array}{l} \underline{\text{ABS}} \\ \\ \underline{\text{REL}} \end{array} \right\}$$

This command specifies the ground motion frequency and damping.

FREQ. The f_1 value is the steady state frequency (Hz) at which the ground will oscillate.

DAMP , MDAMP and CDAMP. Select source of damping input.

f_2 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP indicates to use the f_2 value for all modes.
MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
CDAMP indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the CONSTANT specification.

ABSolute or RELative. Ground motion results in output file will be relative to the ground unless ABSolute is specified. Graphical results are relative. This option has no effect for the force loading cases.

General format:

$$\underline{\textbf{GRO}}\textbf{UND}\ \ \underline{\textbf{MOT}}\textbf{ION}\quad \begin{Bmatrix} \underline{\textbf{X}} \\ \underline{\textbf{Y}} \\ \underline{\textbf{Z}} \end{Bmatrix} \quad \begin{Bmatrix} \underline{\textbf{ACC}}\textbf{EL} \\ \underline{\textbf{DIS}}\textbf{P} \end{Bmatrix}\ f_3\ \ \underline{\textbf{PHA}}\textbf{SE}\ \ f_4$$

Enter the direction of the ground motion, the acceleration magnitude, and the phase angle by which the motion in this direction lags (in degrees). One Ground Motion command can be entered for each global direction.

f_3 Ground acceleration in g’s or displacement in length units.
f_4 Phase angle in degrees

5.37.6.5 Steady Force Loading

This set of commands may be used to specify JOINT loads on the structure, the forcing frequency, the modal damping, and the phase relationship of loads in each of the global directions.

General format:

$$\textbf{\underline{STE}ADY} \;\; \textbf{\underline{FOR}CE} \;\; \textbf{\underline{FRE}Q} \;\; f_1 \;\; \left\{ \begin{array}{l} \textbf{\underline{DAM}P} \;\; f_2 \\ \textbf{\underline{CDA}MP} \\ \textbf{\underline{MDA}MP} \end{array} \right\}$$

This command specifies the forcing frequency and damping for a case of steady forces.

FREQ. The f_1 value is the steady state frequency at which the joint loads below will oscillate.

DAMP , MDAMP and CDAMP. Select source of damping input.

f_2 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP indicates to use the f_2 value for all modes.
MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
CDAMP indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the CONSTANT specification.

General format:

$$\textbf{\underline{JOI}NT} \;\; \textbf{\underline{LOA}D} \;\; (\, [\, \textbf{\underline{PHA}SE}^{*} \left\{ \begin{array}{l} \underline{\textbf{X}} \\ \underline{\textbf{Y}} \\ \underline{\textbf{Z}} \end{array} \right\} f_7 \,] \,)$$

Bracketed data may be entered for each global direction on the same line. All moments specified below will be applied with a phase angle of 0.0. All forces specified below will be applied with the phase angle specified above, if any. Default is 0.0.

f_7 Phase angle in degrees. One phase angle per global direction.

Next are the joint forces, if any. Repeat this command as many times as needed.

$$\textbf{joint-list} \quad * \left\{ \begin{array}{ll} \underline{\textbf{FX}} & \textbf{f}_1 \\ \underline{\textbf{FY}} & \textbf{f}_2 \\ \underline{\textbf{FZ}} & \textbf{f}_3 \\ \underline{\textbf{MX}} & \textbf{f}_4 \\ \underline{\textbf{MY}} & \textbf{f}_5 \\ \underline{\textbf{MZ}} & \textbf{f}_6 \end{array} \right\}$$

FX, FY and FZ specify a force in the corresponding global direction.
MX, MY and MZ specify a moment in the corresponding global direction.
f_1, f_2 … f_6 are the values of the loads.

Notes

- Joint numbers may be repeated where loads are meant to be additive in the joint.
- UNIT command may be on lines in between joint-list lines.

The next command, Copy Load, may optionally be placed here to use the equivalent joint loads from prior cases. This feature enables using the more complex loading commands like selfweight, floor load, wind load, etc. that are not directly available here.

General format:

<u>COPY</u> <u>LOA</u>D

$\mathbf{i_1, f_1, i_2, f_2 \ldots i_n, f_n}$

where,

$i_1, i_2 \ldots i_n$ = prior primary load case numbers that are in this analysis set.

$f_1, f_2 \ldots f_n$ = corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. These cases must have been between the Perform Steady State Analysis command and the prior Analysis command (if any).

5.37.6.6 Harmonic Ground Motion Loading

This set of commands may be used to specify harmonic ground motion loading on the structure, the modal damping, and the phase relationship of ground motions in each of the global directions. Response at all of the frequencies defined in section 5.37.1.2 will be calculated.

General format:

```
HARMONIC GROUND   { DAMP f2 }   { ABS }
                  { CDAMP   }   {     }
                  { MDAMP   }   { REL }
```

This command specifies the damping. The steady state response will be calculated for each specified output frequency entered or generated, see section 5.37.1.2.

DAMP , MDAMP and CDAMP. Select source of damping input.

f_2 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP indicates to use the f_2 value for all modes.
MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
CDAMP indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the CONSTANT specification.

ABSolute or RELative. Ground motion results in output file will be relative to the ground unless ABSolute is specified. Graphical results are relative. This option has no effect for the force loading cases.

General format:

GROUND MOTION $\left\{ \begin{array}{l} \textbf{X} \\ \textbf{Y} \\ \textbf{Z} \end{array} \right\}$ $\left\{ \begin{array}{l} \textbf{ACCEL} \\ \textbf{DISP} \end{array} \right\}$ **f_3 PHASE f_4**

Enter the direction of the ground motion, the acceleration and the phase angle by which the motion in this direction lags (in degrees). One Ground Motion command can be entered for each global direction.

f_3 Ground acceleration in g's or displacement in length units.
f_4 Phase angle in degrees

Next is an optional amplitude versus frequency specification to be used when the ground motion acceleration is a function of frequency. For any forcing frequency an amplitude can be determined, from the data below, which will multiply the acceleration f_3 entered above. If no amplitude data is entered for a direction then the acceleration is f_3 for that direction.

AMPLITUDE (A a B b C c)

Amplitude = $a*\omega^2 + b*\omega + c$
ω = forcing frequency in rad/sec.
a, b, c = constants entered above. a & b default to zero and c defaults to 1.0

--------------- or -----------------

AMPLITUDE
(F1 A1 F2 A2 F3 A3.....)

Frequency - Amplitude pairs are entered to describe the variation of acceleration with frequency. Continue this data onto as many lines as needed by ending each line except the last with a hyphen (-). These pairs must be in ascending order of frequency. Use up to 199 pairs. Linear interpolation is used.

One Ground Motion and Amplitude command set can be entered for each global direction.

5.37.6.7 Harmonic Force Loading

This set of commands may be used to specify JOINT loads on the structure, the modal damping, and the phase relationship of loads in each of the global directions. Response at all of the frequencies defined in section 5.37.1.2 will be calculated.

General format:

HARMONIC FORCE $\left\{ \begin{array}{l} \textbf{DAMP} \ f_2 \\ \textbf{CDAMP} \\ \textbf{MDAMP} \end{array} \right\}$

This command specifies the damping for a case of harmonic forces.

DAMP , MDAMP and CDAMP. Select source of damping input.

f_2 = Damping ratio for all modes. Default value is 0.05 (5% damping if 0 or blank entered).

DAMP indicates to use the f_2 value for all modes.
MDAMP indicates to use the damping entered or computed with the DEFINE DAMP command if entered, otherwise default value of 0.05 will be used.
CDAMP indicates to use the composite damping of the structure calculated for each mode. One must specify damping for different materials under the CONSTANT specification.

General format:

JOINT LOAD ([PHASE* $\left\{ \begin{array}{l} \textbf{X} \\ \textbf{Y} \\ \textbf{Z} \end{array} \right\}$ f_7 **])**

Bracketed data may be entered for each global direction on the same line. All moments specified below will be applied with a phase angle of 0.0. All forces specified below will be applied with the phase angle specified above, if any. Default is 0.0.

f_7 Phase angle in degrees. One phase angle per direction.

Next are the joint forces, if any. Repeat this command as many times as needed.

$$\textbf{joint-list} \; * \left\{ \begin{array}{ll} \underline{\textbf{FX}} & \mathbf{f_1} \\ \underline{\textbf{FY}} & \mathbf{f_2} \\ \underline{\textbf{FZ}} & \mathbf{f_3} \\ \underline{\textbf{MX}} & \mathbf{f_4} \\ \underline{\textbf{MY}} & \mathbf{f_5} \\ \underline{\textbf{MZ}} & \mathbf{f_6} \end{array} \right\}$$

FX, FY and FZ specify a force in the corresponding global direction.
MX, MY and MZ specify a moment in the corresponding global direction.
f_1, f_2 ... f_6 are the values of the loads.

Notes

- Joint numbers may be repeated where loads are meant to be additive in the joint.
- UNIT command may be on lines in between joint-list lines but not within the amplitude data.

The next command, Copy Load, may optionally be placed here to use the equivalent joint loads from prior cases. This feature enables using the more complex loading commands like selfweight, floor load, wind load, etc. that are not directly available here.

General format:

COPY LOAD

$\mathbf{i_1, f_1, i_2, f_2 \ldots i_n, f_n}$

where,
i_1, i_2 ... i_n = prior primary load case numbers that are in this analysis set.

f_1, f_2 … f_n = corresponding factors

This command can be continued to additional lines by ending all but last with a hyphen. These cases must have been between the Perform Steady State Analysis command and the prior Analysis command (if any).

Next is an optional force multiplier (amplitude) versus frequency specification to be used when the force loading is a function of frequency. For any forcing frequency an amplitude can be determined, from the data below, which will multiply the forcing loads entered above. If no amplitude data is entered then the forcing loads are as entered above.

AMPLITUDE $\left\{ \begin{array}{l} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{array} \right\}$ **(A a B b C c)**

Leaving the direction field blank or inserting ALL will use the same Frequency versus Amplitude for all 6 force directions.

Amplitude = $a*\omega^2 + b*\omega + c$
ω = forcing frequency in rad/sec.
a, b, c = constants entered above. a & b default to zero and c defaults to 1.0

---------------- or ------------------

AMPLITUDE $\left\{ \begin{array}{l} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{array} \right\}$

(F1 A1 F2 A2 F3 A3.....)

Leaving the direction field blank or inserting ALL will use the same Frequency versus Amplitude for all 6 force directions.

Frequency - Amplitude pairs are entered to describe the variation of the force multiplier (amplitude) with frequency. Continue this data onto as many lines as needed by ending each line except the last with a hyphen (-). These pairs must be in ascending order of frequency. Use up to 199 pairs. Linear interpolation is used.

Enter amplitudes for up to 3 directions. For directions without amplitude input, including moment directions, the amplitude will be set to 1.0.

5.37.6.8 Print Steady State/Harmonic Results

<u>PRI</u>NT <u>HAR</u>MONIC <u>DIS</u>PLACEMENTS List- spec

List-spec = { **(<u>ALL</u>)** / **<u>LIST</u> list of items-joints** }

This command must be after all steady state/harmonic loadings and before the END STEADY command. For each harmonic frequency of section 5.37.1.2 the following will be printed:

1. Modal responses.
2. Phase angles with 1 line per selected joint containing the phase angle for each of the 6 directions of motion.
3. Displacements table with 1 line per selected joint containing the maximum displacements for each of the 6 directions of motion.
4. Velocities.
5. Accelerations.

STEADY STATE EXAMPLES:

```
BEGIN  STEADY  GROUND
STEADY  GROUND  FREQ  22.4  DAMP .033  ABS
GROUND  MOTION  X  ACC  .11  PHASE  0.0
GROUND  MOTION  Y  ACC  .21  PHASE  10.0
GROUND  MOTION  Z  ACC  .15  PHASE  20.0
PRINT  HARMONIC  DISP  ALL
END
```

```
BEGIN  HARMONIC  GROUND
FREQ  FLO 3.5  FHI  33  NPTS 5  MODAL  FLIST  4 5 10 -
17 21 30
HARMONIC  GROUND  DAMP  .033  REL
GROUND  MOTION  X  ACC  .11  PHASE  0.0
AMPLIT    A  0.10 B .21 C  0.03
GROUND  MOTION  Y  DIS  .21  PHASE  10.0
AMPLITUDE
```

```
3 5  5 4  10 6 -
35 3
GROUND  MOTION  Z  ACC  .15  PHASE  20.0
AMPLIT    A  0.10 B .21 C  0.03
PRINT  HARMONIC  DISP  ALL
END
```

```
BEGIN  STEADY  FORCE
STEADY  FORCE  FREQ 11.2  DAMP .033
JOINT  LOAD  PHASE  X  0.0   PHASE Y  10.0   PHASE Z  15.0
UNIT KIP
10 5 TO 7 BY 2 88  FX  10.0  FY  5.0
UNIT POUND
10 5 TO 7 BY 2 -
88  FX  10.0  FY  5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
PRINT  HARMONIC  DISP  ALL
END
```

```
BEGIN  HARMONIC  FORCE
FREQ  FLO 3.5  FHI  33  NPTS 5  MODAL  FLIST  4 5 10 -
17 21 30
HARMONIC  FORCE  DAMP  .033
JOINT  LOAD  PHASE  X  0.0   PHASE Y  10.0   PHASE Z  15.0
UNIT KIP
10 5 TO 7 BY 2 88  FX  10.0  FY  5.0
UNIT POUND
10 5 TO 7 BY 2 -
88  FX  10.0  FY  5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT  X  A  0.10 B .21
AMPLITUDE  Y
3 5  5 4  10 6 -
35 3
AMPLIT  Z  A  0.10  C  0.03
PRINT  HARMONIC  DISP  ALL
END
```

```
BEGIN  HARMONIC  FORCE
FREQ  FLO 3.5  FHI  33  NPTS 5  MODAL  FLIST  4 5 10 -
17 21 30
HARMONIC  FORCE  DAMP  .033
JOINT  LOAD  PHASE  X  0.0   PHASE Y  10.0   PHASE Z  15.0
UNIT KIP
10 5 TO 7 BY 2 88  FX  10.0  FY  5.0
UNIT POUND
10 5 TO 7 BY 2 -
88  FX  10.0  FY  5.0
COPY LOAD
1 1.5 2 0.8 -
3 1.0
AMPLIT  ALL  A  0.10 B .21
PRINT  HARMONIC  DISP  ALL
END
```

STEADY STATE NOTES:

For members, the final results at the joints will be complete but the section results will be as if the member loads were applied as statically equivalent loads at the member ends.

5.37.6.9 Last Line of this Steady State/Harmonic Analysis

<u>END</u> <u>STE</u>ADY

5.37.7 Pushover Analysis

See Section 1.18.3.7

Please contact the technical support group for a separate document containing the details of the implementation of pushover analysis.

Users will require a license for the advanced analysis module to access this feature.

5.38 Change Specification

Purpose

This command is used to reset the stiffness matrix. Typically, this command is used when multiple analyses are required in the same run.

General format:

CHANGE

This command indicates that input, which will change the stiffness matrix, will follow. This command should only be used when an analysis has already been performed. The CHANGE command does or requires the following:

a) sets the stiffness matrix to zero,
b) makes members active if they had been made inactive by a previous INACTIVE command, and
c) allows the re-specification of the supports with another SUPPORT command that causes the old supports to be ignored. The SUPPORT specification must be such that the number of joint directions that are free to move (DOF or "releases") before the CHANGE must be greater than or equal to the number of "releases" after the CHANGE.
d) Also, the supports must be specified in the same order before and after the CHANGE command. To accomplish this when some cases have more supports than others do, you can enter unrestrained joints into the SUPPORT command list using FIXED BUT FX FY FZ MX MY MZ. It is best to put every joint that will be supported in any case into every SUPPORT list.
e) CHANGE, if used, should be after PERFORM ANALYSIS and before the next set of SUPPORT, LOADS.
f) Only active cases are processed after the CHANGE command.

g) Analysis and CHANGE are required between primary cases for PERFORM CABLE ANALYSIS.

h) Analysis and CHANGE are required after each UBC case if the case is subsequently referred to in a Repeat Load command or if the UBC case will be re-solved after a Select command or after a Multiple analysis.

Example

Before CHANGE

```
1 PINNED
2 FIXED BUT FX MY MZ
3 FIXED BUT FX MX MY MZ
```

After CHANGE

```
1 PINNED
2 FIXED
3 FIXED BUT FX MZ
```

See Section 5.18

The CHANGE command is not necessary when only member properties are revised to perform a new analysis. This is typically the case in which the user has asked for a member selection and then uses the PERFORM ANALYSIS command to reanalyze the structure based on the new member properties.

Notes

1) If new load cases are specified after the CHANGE command such as in a structure where the INACTIVE MEMBER command is used, the user needs to define the total number of primary load cases using the SET NL option (see Section 5.5 and Example 4).

2) Multiple Analyses using the CHANGE command should not be performed if the input file contains load cases involving dynamic analysis or Moving Load Generation.

3) Section forces and moments, stress and other results for postprocessing will use the last entered data for supports and member properties regardless of what was used to compute the displacements, end forces and reactions. So beware of changing member properties and releases after a CHANGE command.

5.39 Load List Specification

Purpose

This command allows specification of a set of active load cases. All load cases made active by this command remain active until a new load list is specified.

General format:

$$\textbf{\underline{LOA}D \underline{LIS}T} \quad \left\{ \begin{array}{l} \textbf{load-list} \\ \textbf{\underline{ALL}} \end{array} \right\}$$

Description

This command is used to activate the load cases listed in this command and, in a sense, deactivate all other load cases not listed in this command. In other words, the loads listed are used for printing output and in design for performing the specified calculations. When the PERFORM ANALYSIS command is used, the program internally uses all load cases, regardless of LOAD LIST command, except after a CHANGE command. In these two cases, the LOAD LIST command allows the program to perform analysis only on those loads in the list. If the LOAD LIST command is never used, the program will assume all load cases to be active.

Example

```
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 3
PRINT SUPPORT REACTIONS
CHECK CODE ALL
```

In this example, member forces will be printed for all load cases, whereas loading 1 and 3 will be used for printing support reactions and code-checking of all members.

Notes

The LOAD LIST command may be used for multiple analyses situations when a re-analysis needs to be performed with a selected set of load cases only. All load cases are automatically active before the first CHANGE command is used.

After a CHANGE command has been used anywhere in the data, it is good practice to specify the LOAD LIST command after an ANALYSIS command and before the next command; otherwise only the last set of cases analyzed will be used in the design.

Do not enter this command within the loads data (from the first Load command in an analysis set down to the associated Analysis command).

5.40 Load Envelope

Purpose

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

It is an alternative to the LOAD LIST command described in Section 5.39 of this manual.

Description

The envelope can be tagged with optional key words to specify the qualitative nature of the load or load combination cases included in the envelop definition. In the example below, the keyword SERVICEABILITY is associated with envelop 2. Keywords can be any single word (with no blank spaces) of the user's choice.

Example

```
DEFINE ENVELOP
1 TO 8 ENVELOP 1 TYPE CONNECTION
9 TO 15 ENVELOP 2 TYPE SERVICEABILITY
16 TO 28 ENVELOP 4 TYPE STRESS
END DEFINE ENVELOP
```

The first line within the DEFINE ENVELOPE command means that load cases numbered 1 to 8 make up the CONNECTION type load envelope 1. Similarly load cases 9 to 15 define the SERVICEABILITY type load envelope 2.

To print out the support reactions corresponding to load envelope 1, the following commands should be provided in the input file

LOAD LIST ENV 1
PRINT SUPPORT REACTIONS

5.41 Section Specification

Purpose

This command is used to specify intermediate sections along the length of frame member for which forces and moments are required.

General format:

<u>SEC</u>TION f_1, f_2 ... f_3 $\left\{ \begin{array}{l} \textbf{\underline{MEM}BER memb-list} \\ \textbf{(\underline{ALL})} \end{array} \right\}$

Description

This command specifies the sections, in terms of fractional member lengths, at which the forces and moments are considered for further processing.

See Sections 1.19.2,and 1.19.4

f_1, f_2 ... f_3 = Section (in terms of the fraction of the member length) provided for the members. Maximum number of sections is 3, and only values between 0.0 and 1.0 will be considered. In other words, no more than three intermediate sections are permissible per SECTION command.

Example

```
SECTION 0.17 0.48 0.72 MEMB 1 2
SECTION 0.25 0.75 MEMB 3 TO 7
SECTION 0.6 MEMB 8
```

In the above example, first, section locations of 0.17, 0.48, and 0.72 are set for members 1 and 2. In the next SECTION command, sections 0.25 and 0.75 are set for members 3 to 7. In the third SECTION command, member 8 has its section specified at 0.6. The remainder of the members will have no sections provided for them. As mentioned earlier, no more than three intermediate sections are allowed per SECTION command. However, if more than three intermediate sections are desired, they can be examined by repeating the SECTION command after completing the required calculations. The following example will clarify.

Example

```
SECTION 0.2 0.4 0.5 ALL
PRINT SECTION FORCES
SECTION 0.6 0.75 0.9 ALL
PRINT SECTION FORCES
```

In this example, forces at 3 intermediate sections (namely 0.2, 0.4 and 0.5) are printed. Then forces at an additional 3 sections (namely 0.6, 0.75 and 0.9) are printed. This gives the user the ability to obtain section forces at more than three intermediate sections.

Notes

1) The SECTION command just specifies the sections. Use the PRINT SECTION FORCES command after this command to print out the forces and moments at the specified sections.
2) This is a secondary analysis command. The analysis must be performed before this command may be used.
3) To obtain values at member ends (START and END), use the PRINT MEMBER FORCES command.
4) If this command is specified before a steel design operation, and if the BEAM parameter is set to zero, the section locations specified by this command will also be designed for, in addition to the BEAM ends.

5.42 Print Specifications

Purpose

This command is used to direct the program to print various modeling information and analysis results. STAAD offers a number of versatile print commands that can be used to customize the output.

General format for data related print commands:

```
         ┌ JOINT  COORDINATES                 ┐
         │ MEMBER  INFORMATION                │
         │ ELEMENT INFORMATION (SOLID)        │  ┌ (ALL)                    ┐
PRINT   <  MEMBER  PROPERTIES                  > <                           >
         │ MATERIAL  PROPERTIES               │  │ LIST  list of items      │
         │ SUPPORT  INFORMATION               │  └       i.e. joints,       ┘
         │ or                                 │          members
         └ ALL                                ┘
```

General format to print location of cg:

PRINT CG (_group_name)

General format to print analysis results:

```
         ┌ (JOINT)  DISPLACEMENTS                         ┐
         │ (MEMBER)  FORCES                               │
PRINT   <  ANALYSIS  RESULTS                               >  List-
         │ (MEMBER)  SECTION FORCES                       │   spec
         │ (MEMBER)  STRESSES                             │
         │ ELEMENT  (JOINT)  STRESSES (AT f1 f2)          │
         │ ELEMENT  FORCES                                │
         │ ELEMENT  (JOINT)  STRESSES SOLID               │
         └ MODE  SHAPES                                   ┘

              ┌ (ALL) ,                          ┐
List-spec =  <  LIST   list of items-joints,      >
              └        members or elements       ┘
```

General format to print support reactions:

<u>PRI</u>NT <u>SUP</u>PORT <u>REA</u>CTIONS

General format to print story drifts:

<u>PRI</u>NT <u>STO</u>RY DRIFT

Description

The list of items is not applicable for PRINT ANALYSIS RESULTS and PRINT MODE SHAPES commands.

The PRINT JOINT COORDINATES command prints all interpreted coordinates of joints.

The PRINT MEMBER INFORMATION command prints all member information, including member length, member incidences, beta angles, whether or not a member is a truss member and the member release conditions at start and end of the member (1=released, 0 = not released).

The PRINT ELEMENT INFORMATION command prints all incident joints, element thicknesses, and Poisson ratios for Plate/Shell elements. The PRINT ELEMENT INFORMATION SOLID command prints similar information for Solid elements.

The PRINT MEMBER PROPERTIES command prints all member properties including cross sectional area, moments of inertia, and section moduli in both axes. Units for the properties are always INCH or CM (depending on English Inch/Kip/Second or METRIC) regardless of the unit specified in UNIT command.

The following designation is used for member property names:

AX - Cross section area
AY - Area used to adjust shear/bending stiffness in local Y axis to account for pure shear in addition to the classical bending stiffness.
AZ - Area used to adjust shear/bending stiffness in local Z axis to account for pure shear in addition to the classical bending stiffness.
IZ - Moment of Inertia about the local Z-axis
IY - Moment of Inertia about the local Y-axis
IX - Torsional constant
SY - Smallest section modulus about the local Y-axis
SZ - Smallest section modulus about the local Z-axis

The PRINT MATERIAL PROPERTIES command prints all material properties for the members, including E (modulus of elasticity), G (shear modulus), weight density and coefficient of thermal expansion (alpha) for frame members. This command is available for members only. G may be listed as zero if command is before load cases and Poisson ratio was entered but G was not entered.

The PRINT SUPPORT INFORMATION command prints all support information regarding their fixity, releases and spring constant values, if any. The LIST option is not available for this command.

The PRINT ALL command is equivalent to last five print commands combined. This command prints joint coordinates, member information, member properties, material properties and support information, in that order.

The PRINT CG command prints out the coordinates of the center of gravity and the total weight of the structure or of a single group of member/elements. If the CG of a portion of the structure is desired, the members and elements of that portion must be assigned using a group name (see section 5.16 for details on using group-names). Only the selfweight of the structure is used to calculate the C.G. User defined joint loads, member loads etc. are not considered in the calculation of C.G.

The PRINT (JOINT) DISPLACEMENTS command prints joint displacements in a tabulated form. The displacements for all six global directions will be printed for all specified load cases. The length unit for the displacements is always INCH or CM (depending on English Inch/Kip/Second or METRIC unit) regardless of the unit specified in UNIT command.

The PRINT (MEMBER) FORCES command prints member forces (i.e. Axial force (AXIAL), Shear force in local Y and Z axes (SHEAR-Y and SHEAR-Z), Torsional Moment (TORSION), Moments about local Y and Z axes (MOM-Y and MOM-Z)) in a tabulated form for the listed members, for all specified load cases.

The PRINT SUPPORT REACTIONS command prints global support reactions in a tabulated form, by support, for all specified load cases. Use LIST option for selected joints.

PRINT ANALYSIS RESULTS command is equivalent to the above three commands combined. With this command, the joint displacements, support reactions and member forces, in that order, are printed.

The PRINT (MEMBER) SECTION FORCES command prints shears & bending moments at the intermediate sections specified with a previously input SECTION command. The printing is done in a tabulated form for all specified cases for the first requested member, then for the next member, etc.

The PRINT (MEMBER) STRESSES command tabulates member stresses at the start joint, end joint and all specified intermediate sections. These stresses include axial (i.e. axial force over the area), bending-y (i.e. moment-y over section modulus in local y-axis), bending-z (i.e. moment-z over section modulus in local z-axis), shear stresses in both local y and z directions and combined (absolute combination of axial, bending-y and bending-z) stresses.

For PRISMATIC sections, if AY and/or AZ is not provided, the full cross-sectional area (AX) will be used.

For TAPERED sections, the values of AY and AZ are those for the location where the stress is printed. Hence at the location 0.0, the AY and AZ are based on the dimensions of the member at the start node.

The PRINT ELEMENT STRESSES command must be used to print plate stresses (SX, SY, SXY, SQX, SQY), moments per unit width (MX, MY, MXY) and principal stresses (SMAX, SMIN, TMAX) for plate/shell elements. Typically, the stresses and moments per unit width at the centroid will be printed. The Von Mises stresses at the top and bottom surfaces (VONT, VONB) as well as the angle (ANGLE) defining the orientation of the principal planes are also printed.

The variables that appear in the output are the following. See Figure 1.13 in Section 1 of this Reference Manual for more information regarding these variables.

SQX	= Shear stress on the local X face in the Z direction
SQY	= Shear stress on the local Y face in the Z direction
MX	= Moment per unit width about the local X face
MY	= Moment per unit width about the local Y face
MXY	= Torsional Moment per unit width in the local X-Y plane
SX	= Axial stress in the local X direction
SY	= Axial stress in the local Y direction
SXY	= Shear stress in the local XY plane
VONT	= Von Mises stress on the top surface of the element
VONB	= Von Mises stress on the bottom surface of the element
TrescaT	= Tresca stress on the top surface of the element
TrescaB	= Tresca stress on the bottom surface of the element
SMAX	= Maximum in-plane Principal stress
SMIN	= Minimum in-plane Principal stress
TMAX	= Maximum in-plane Shear stress

ANGLE = Angle which determines direction of maximum principal stress with respect to local X axis

If the JOINT option is used, forces and moments at the nodal points are also printed out in addition to the centroid of the element.

The AT option may be used to print element forces at any specified point within the element. The AT option must be accompanied by f_1 and f_2. f_1 and f_2 are local X and Y coordinates (in current units) of the point where the stresses and moments are required. For detailed description of the local coordinate system of the elements, refer to Section 1.6 of this manual.

The PRINT ELEMENT FORCES command enables printing of plate "corner forces" [$F_p = K_p \cdot D_p$] in global axis directions.

The PRINT ELEMENT (JOINT) STRESS SOLID command enables printing of stresses at the center of the SOLID elements. The variables that appear in the output are the following.

Normal Stresses	: SXX, SYY and SZZ
Shear Stresses	: SXY, SYZ and SZX
Principal Stresses	: S1, S2 and S3.
Von Mises Stresses	: SE
Direction cosines	: 6 direction cosines are printed following the expression DC, corresponding to the first two principal stress directions.

The JOINT option will print out the stresses at the nodes of the solid elements.

The PRINT MODE SHAPES command prints the relative joint motions of each of the modes that were calculated. The maximum motion is arbitrary and has no significance. Dynamic analysis will scale and combine the mode shapes to achieve the final dynamic results.

Example

```
PERFORM ANALYSIS
PRINT ELEMENT JOINT STRESS
PRINT ELEMENT STRESS AT 0.5 0.5 LIST 1 TO 10
PRINT SUPPORT REACTIONS
PRINT JOINT DISPLACEMENTS LIST 1 TO 50
PRINT MEMBER FORCES LIST 101 TO 124
```

Notes

1) The output generated by these commands is based on the current unit system. The user may wish to verify the current unit system and change it if necessary.

2) Results may be printed for all joints/members/elements or based on a specified list.

The PRINT STORY DRIFT command may be used to obtain a print-out of the average lateral displacement of all joints at each horizontal level along the height of the structure.

The procedure used in STAAD for calculating story drift is independent of any code. For example, the story drift determination as explained in section 12.8.6 of the ASCE 7-05 code is not implemented in STAAD.

The method implemented in STAAD involves:

a. Find all the distinct Y coordinates in the model. Those are what STAAD calls as storys (storeys).

b. For each of those distinct storys, find all the nodes at that story elevation.

c. For each story, find the average displacement along the horizontal directions (X and Z) by adding up corresponding displacement for all the nodes at that storey, and dividing by the number of nodes for that story. Thus, even if there is only a single node representing a story, a drift is calculated for that story too.

Example for printing the CG

```
PRINT CG
PRINT CG _RAFTERBEAMS
PRINT CG _RIDGEBEAMS
```

5.43 Stress/Force output printing for Surface Entities

Default locations for stress/force output, design, and design output for surface elements are set as follows:

SURFACE DIVISION X xd

SURFACE DIVISION Y yd

where:

xd - number of divisions along X axis,

yd - number of divisions along Y axis.

xd and yd represent default numbers of divisions for each edge of the surface where output is requested. The output is provided for sections located between division segments. For example, if the number of divisions = 2, then the output will be produced for only one section (at the center of the edge).

Values of internal forces may be printed out for any user-defined section of a wall modelled using the Surface element.

General Format:

PRINT SURFACE FORCE (ALONG ξ) (AT a) (BETWEEN d1, d2) LIST s1, ...,si

where:

ξ - local axis of the surface element (X or Y),

a - distance along the ξ axis from start of the member to the full cross-section of the wall, d1, d2 - coordinates in the direction orthogonal to ξ, delineating a fragment of the full cross-section for which the output is desired.

s1, ...,si - list of surfaces for output generation

Note: If the keyword ALONG is omitted, direction Y (default) is assumed.

If command AT is omitted, output is provided for all sections along the specified (or default) edge. Number of sections will be determined from the SURFACE DIVISION X or SURFACE DIVISION Y input values.

If the BETWEEN keyword is omitted, the output is generated based on full cross-section width.

The attributes associated with surfaces, and the sections of this manual where the information may be obtained, are listed below:

Attributes	**Related Sections**
Surfaces incidences -	5.13.3
Openings in surfaces -	5.13.3
Local coordinate system for surfaces -	1.6.3
Specifying sections for stress/force output -	5.13.3
Property for surfaces -	5.21.2
Material constants -	5.26.3
Surface loading -	5.32.3.4
Stress/Force output printing -	5.42
Shear Wall Design -	3.8.2, 5.54

5.44 Printing Section Displacements for Members

Purpose

This command is used to calculate and print displacements at sections (intermediate points) of frame members. This provides the user with deflection data between the joints.

General format:

PRINT SECTION (MAX) DISPL (NSECT i) (SAVE a) { **NOPRINT** / **ALL** / **LIST memb-list** }

Description

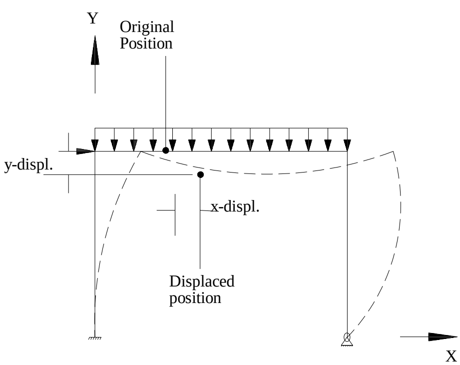


Figure 5.40

This command prints displacements at intermediate points between two joints of a member. These displacements are in global coordinate directions (see figure). If the MAX command is used, the program prints only the maximum local displacements among all load cases.

i = number of sections to be taken. Defaults to 12 if NSECT is not used and also if SAVE is used (max=24, min=2).

a = File name where displacement values can be stored and used by the STAADPL graphics program. If the NOPRINT command is used in conjunction with the SAVE command, the program writes the data to file only and does not print them in the output. <u>This option is not necessary in STAAD.Pro.</u>

Example

```
PRINT SECTION DISPL SAVE
PRINT SECTION MAX DISP
```

See Section 1.19.3

SECTION DISPLACEMENTS are measured in GLOBAL COORDINATES. The values are measured from the original (undeflected) position to the deflected position. See figure above.

The maximum local displacement is also printed. First, the location is determined and then the value is measured from this location to the line joining start and end joints of the deflected member.

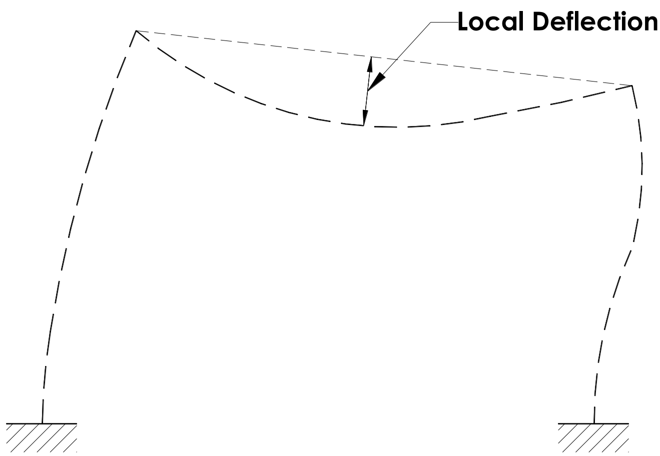


Figure 5.41

Notes

1) The section displacement values are available in Global Coordinates. The undeflected position is used as the datum for calculating the deflections.
2) This is a secondary analysis command. An analysis must be performed before this command may be used.

5.45 Printing the Force Envelope

Purpose

This command is used to calculate and print force/moment envelopes for frame members. This command is not available for finite elements.

General format:

```
        { FORCE    }
PRINT   {          }   ENVELOPE  (NSECTION i)   list-sp.
        { MAXFORCE }

                 { LIST  memb-list }
   list-spec =   {                 }
                 { (ALL)           }
```

Description

See Section 1.19.5

Where i is the number of equally spaced sections to be considered in printing maximum and minimum force envelopes (where i is in the range 2 through 96). If the NSECTION i command is omitted, i will default to 12. MAXFORCE command produces maximum/minimum force values only of all sections, whereas the FORCE command prints maximum/minimum force values at every section as well as the max/min force values of all sections. The force components include FY, MZ, FZ, and MY. The SECTION command (as described in section 5.40) does not define the number of sections for force envelopes. For the sign convention of force values, refer to Section 2.19.

Example

```
PRINT FORCE ENV
PRINT MAXF ENV NS 15
PRINT FORCE ENV NS 4 LIST 3 TO 15
```

Notes

This is a secondary analysis command and should be used after analysis specification.

5.46 Post Analysis Printer Plot Specifications

Purpose

This command has been discontinued in STAAD.Pro. Please use the facilities of the Graphical User Interface (GUI) for screen and hard copy graphics.

5.47 Size Specification

Purpose

This command provides an estimate for required section properties for a frame member based on certain analysis results and user requirements.

General Format:

$$\underline{\text{SIZ}}\text{E} \quad {}^{*}\left\{ \begin{array}{ll} \underline{\text{WID}}\text{TH} & f_1 \\ \underline{\text{DEF}}\text{LECTION} & f_2 \\ \underline{\text{LEN}}\text{GTH} & f_3 \\ \underline{\text{BST}}\text{RESS} & f_4 \\ \underline{\text{SST}}\text{RESS} & f_5 \end{array} \right\} \left\{ \begin{array}{l} \underline{\text{MEM}}\text{BER member-list} \\ \underline{\text{ALL}} \end{array} \right\}$$

where,

f_1 = Maxm. allowable width

f_2 = Maxm. allowable (Length/Maxm. local deflection) ratio

f_3 = Length for calculating the above ratio.
Default = actual member length.

f_4 = Maxm. allowable bending stress.

f_5 = Maxm. allowable shear stress.

The values must be provided in the current unit system.

Description

This command may be used to calculate required section properties for a member based on analysis results and user specified criteria. The user specified criteria may include Member width, Allowable (Length/Maxm. Deflection) Ratio, Maxm. allowable bending stress and Maximum allowable shear stress. Any number of these criteria may be used simultaneously. The output includes required Section Modulus (about major axis), required Shear Area (for shear parallel to minor axis), Maxm. moment capacity (about major axis), Maxm. shear capacity (for shear parallel to minor axis) and Maxm. (Length/local maxm. deflection) ratio.

Example

```
SIZE WID 12 DEFL 300 LEN 240 BSTR 36 ALL
SIZE DEFL 450 BSTR 42 MEMB 16 TO 25
```

Note:
It may be noted that sizing will be based on only the criteria specified by the user in the relevant SIZE command.

In the first example above, sizing will be based on user specified member width of 12, Length/Deflection ratio of 300 (where Length= 240) and max. allowable bending stress of 36.

In the second example, sizing will be based on Length/Deflection ratio of 450 (where Length= actual member length) and max. allowable bending stress of 42.

Notes

This is a post-analysis facility and must be used after the analysis specifications.

5.48 Steel and Aluminum Design Specifications

This section describes the specifications necessary for structural steel & aluminum design.

The specific details of the implementation of these codes may be found in the following places:

American AISC ASD & AISC LRFD -	Section 2 of this manual
AASHTO -	Section 2 of this manual
AISI (American Cold formed steel code) -	Section 2 of this manual
All other countries -	International Codes Manual
American Aluminum specifications -	International Codes Manual
ASCE72 Transmission Tower code -	International Codes Manual
API code -	International Codes Manual

Section 5.48.1 discusses specification of the parameters that may be used to control the design. Sections 5.48.2 and 5.48.3 describe the CODE CHECKING and MEMBER SELECTION options respectively. Member Selection by optimization is discussed in 5.48.4.

STAAD also provides facilities for Weld Design per the American welding codes. Details may be found in sections 2.12 and 5.48.5 of this manual.

5.48.1 Parameter Specifications

Purpose

This set of commands may be used to specify the parameters required for steel and aluminum design.

General format:

PARAMETER

```
        { AASHTO     }
        { AISC       }
        { AISI       }
        { ALUMINUM   }
        { S136       }
        { AUSTRALIAN }
        { BRITISH    }
        { CANADIAN   }
CODE    { FRENCH     }
        { GERMAN     }
        { INDIA      }
        { JAPAN      }
        { LRFD       }
        { NORWAY     }
```

Other code names are: BS5400, RUSSIA, LRFD2, API, ALUMINUM, TIMBER, SPANISH, CHINA, EUROPE, ASCE, DUTCH, NPD,DANISH, BSK94, FINNISH, IS801, IS802, MEXICAN, BS5950 1990

$$\left\{\begin{array}{l}\text{parameter-name } f_1 \\ \text{PROFILE } a_1, (a_2, a_3)\end{array}\right\} \quad \left\{\begin{array}{l}\text{MEMBER memb-list} \\ \text{ALL} \\ \text{membergroupname} \\ \text{deckname}\end{array}\right\}$$

Please see section 5.16 for the definition of member group names. Deck names are explained in section 5.20.7.

Description

Parameter-name refers to the "PARAMETER NAME" (s) listed in the parameter table contained in the Steel and Aluminum Design section.

f_1 = Value of the parameter.

The details of the parameters available for specific codes may be found in the following places:

American AISC ASD & AISC LRFD -	Section 2 of this manual
AASHTO -	Section 2 of this manual
AISI (American Cold formed steel code) -	Section 2 of this manual
All other countries -	International Codes Manual
American Aluminum specifications -	International Codes Manual
ASCE72 Transmission Tower code -	International Codes Manual
API code -	International Codes Manual

The user can control the design through specification of proper parameters.

The PROFILE parameter is available for only a limited number of codes like the AISC ASD and AISC LRFD. The user can specify up to three profiles (a_1, a_2 and a_3). Profile is the first three letters of a section name from its steel table, like, W8X, W12, C10, L20 etc. The PROFILE parameter-name is used only for member selection where members are selected from each of those profile names. The PROFILE for a T-section is the corresponding W-shape. Also, the shape specified under PROFILE has to be the same as that specified initially under MEMBER PROPERTIES.

See Section 2.3 and Table 2.1

CODE parameter lets you choose the type of steel or aluminum code to be checked for design. The default steel code depends on the country of distribution.

Example

```
PARAMETERS
CODE AISC
KY 1.5 MEMB 3 7 TO 11
NSF 0.75 ALL
PROFILE W12 W14 MEMB 1 2 23
RATIO 0.9 ALL
```

Notes

1) All unit sensitive values should be in the current unit system.
2) For default values of the parameters, refer to the appropriate parameter table.

5.48.2 Code Checking Specification

Purpose

This command may be used to perform the CODE CHECKING operation for steel and aluminum members.

General format:

$$\text{\underline{CHE}CK \underline{CODE}} \begin{Bmatrix} \text{\underline{MEM}BER memb-list} \\ \text{\underline{ALL}} \\ \text{membergroupname} \\ \text{deckname} \end{Bmatrix}$$

Description

This command checks the specified members against the specification of the desired code. Refer to Section 2 of this manual for detailed information.

See Section 2.5

Notes

The output of this command may be controlled using the TRACK parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in 5.48.1 for more information on the TRACK parameter.

Member group names and deck names are explained in section 5.16 and 5.20.7 respectively.

5.48.3 Member Selection Specification

Purpose

This command may be used to perform the MEMBER SELECTION operation.

General format:

```
SELECT  { MEMBER memb-list }
        { ALL              }
        { membergroupname  }
        { deckname         }
```

Description

This command instructs STAAD to select specified members based on the parameter value restrictions and specified code. The selection is done using the results from the most recent analysis and iterating on sections until a least weight size is obtained. Refer to Section 2 for more details.

It is important that the keywords MEMBER or ALL be provided. Thus, the keyword SELECT by itself is not sufficient.

Examples

1) SELECT MEMB 22 TO 35
2) SELECT ALL

Notes

1) The output of this command may be controlled using the TRACK parameter. Various codes support various levels of details. Refer to the appropriate section of the documentation, as explained in the table in 5.48.1 for more information on the TRACK parameter.

2) Member selection can be done only after an analysis has been performed. Consequently, the command to perform the analysis has to be specified before the SELECT MEMBER command can be specified.
3) This command does not cause the program to re-analyze for results based on the selected member sizes. However, to maintain compatibility of analysis results with the final member sizes, you should enter a subsequent PERFORM ANALYSIS command. Otherwise the post processor will display the prior results with the revised member sizes.
4) Member group names and deck names are explained in section 5.16 and 5.20.7 respectively.

5.48.4 Member Selection by Optimization

Purpose

This command performs member selection using an optimization technique based on multiple analysis/design iterations.

General format:

<u>SEL</u>ECT <u>OPT</u>IMIZED

Description

The program selects all members based on an optimization technique. This method performs 2 analyses as well as iteration of sizes to reduce the overall structure weight. This command should be used with caution since it will require longer processing time.

Notes

1) The output of this command may be controlled using the TRACK parameter. Three levels of details are available. Refer to the appropriate Steel Design section for more information on the TRACK parameter.
2) This command will perform 1 additional analysis/design cycles and therefore may be time consuming. Steps taken are: CHECK CODE ALL; then modify ratios; then SELECT ALL; then PERFORM ANALYSIS; then SELECT ALL. See section 5.48 Group Specification for other options used with this command. You may want to repeat this command for further optimization.
3) See section 5.48.3 note 3).

5.48.5 Weld Selection Specification

Purpose

This command performs selection of weld sizes for specified members.

General format:

```
SELECT WELD (TRUSS) { MEMBER memb-list }
                    { ALL              }
```

Description

By this command, the program selects the weld sizes of the specified members at start and end. The selections are tabulated with all the necessary information. If the TRUSS command is used, the program will design welds for angle and double angle members attached to gusset plates with the weld along the length of members.

Notes

Currently, this feature is available only with the American welding specifications. Regardless of the code name specified prior to this command, design will be performed according to US code specifications. Some information on the implementation of these specifications is available in Section 2.12 of this manual. The parameters available for controlling weld sizes are listed in Table 2.1 in Section 2 of this manual.

It is important that the keywords MEMBER or ALL be provided. Thus, the keyword SELECT by itself is not sufficient.

Examples

1) SELECT WELD TRUSS MEMB 22 TO 35
2) SELECT WELD ALL

5.49 Group Specification

Purpose

This command may be used to group members together for analysis and steel design.

General format:

(<u>FIX</u>ED GROUP)
<u>GRO</u>UP prop-spec <u>MEM</u>B memb-list (<u>SAM</u>E <u>AS</u> i_1)

$$\textbf{prop-spec} = \left\{\begin{array}{l}\textbf{AX}\\ \textbf{SY}\\ \textbf{SZ}\end{array}\right\}\quad\begin{array}{l}\textbf{= Cross-section area}\\ \textbf{= Section modulus in local y-axis}\\ \textbf{= Section modulus in local z-axis}\end{array}$$

Description

This command enables the program to group specified members together for analysis based on their largest property specification. However, if member number i_1 is provided in the SAME AS command, the program will group the members based on the properties of i_1.

When FIXED GROUP is omitted, the GROUP command is usually entered after the member selection command, and the selected members will be grouped immediately and the new member properties will be used in any further operations. After the GROUPing is completed, the GROUP commands are discarded and will not be used again. Further grouping will be done only if a new GROUP command is encountered later.

If the FIXED GROUP option precedes the group data, the specified grouping will be retained in memory by the program and will be used in subsequent SELECT commands. No "GROUPing" will occur unless a "SELECT (MEMBER or ALL or OPTIMIZED)" command is performed. However, grouping will be performed with every subsequent SELECT command.

Example 1

```
SELECT ALL
GROUP SZ MEMB 1 3 7 TO 12 15
GROUP MEMB 17 TO 23 27 SAME AS 30
```

In this example, the members 1, 3, 7 to 12, and 15 are assigned the same properties based on which of these members has the largest section modulus. Members 17 to 23 and 27 are assigned the same properties as member 30, regardless of whether member 30 has a smaller or larger cross-sectional area. AX is the default property upon which grouping is based.

Example 2

```
FIXED GROUP
GROUP MEMB 1 TO 5
SELECT OPTIMIZED
```

In the above example, the usage of the FIXED GROUP command is illustrated. In this example, the SELECT OPTIMIZED command involves the 6 stage process of

1. CHECK CODE ALL followed by modification of RATIO
2. SELECT ALL
3. GROUPING MEMBERS 1 TO 5
4. PERFORM ANALYSIS
5. SELECT ALL
6. GROUPING MEMBERS 1 TO 5

The FIXED GROUP command (and the GROUP commands that follow it) is required for execution of steps 3 and 6 in the cycle. You may want to repeat this data for further optimization.

Notes

The FIXED GROUP + GROUP commands are typically entered before the member selection for further analysis and design. This facility may be effectively utilized to develop a practically oriented design where several members need to be of the same size.

All the members in a list for a specific GROUP command should have the same cross section type. Thus, if the command reads

GROUP MEMB 1 TO 10

and member 3 is a W shape and member 7 is a Channel, grouping will not be done. The 10 members must be either all W shapes or all channels.

Also see section 5.48.3 note 3.

5.50 Steel and Aluminum Take Off Specification

Purpose

These commands may be used to obtain a summary of all steel sections and Aluminum sections being used along with their lengths and weights (quantity estimates).

General format:

$$\underline{\text{STE}}\text{EL}\ \ (\underline{\text{MEM}}\text{BER})\ \ \underline{\text{TAK}}\text{E}\ \ (\ \underline{\text{OFF}}\ )\ \left(\begin{Bmatrix} \underline{\text{LIS}}\text{T memb-list} \\ \text{LIST membergroupname} \\ \underline{\text{ALL}} \end{Bmatrix}\right)$$

$$\underline{\text{ALUMINUM}}\ \ (\underline{\text{MEM}}\text{BER})\ \ \underline{\text{TAK}}\text{E}\ \ (\ \underline{\text{OFF}}\ )\ \left(\begin{Bmatrix} \underline{\text{LIS}}\text{T memb-list} \\ \text{LIST membergroupname} \\ \underline{\text{ALL}} \end{Bmatrix}\right)$$

Description

These commands provide a listing of the different steel and aluminum table sections used in the members selected. For each section name, the total length and total weight of all members which have been assigned that section will be listed in a tabular form. This can be helpful in estimating steel and aluminum quantities.

If the MEMBER option is specified, the length and weight of each member and the section name it is assigned will be reported.

Example

```
STEEL TAKE OFF LIST 71 TO 85
ALUMINUM TAKE OFF LIST_PLGNO3
```

5.51 Timber Design Specifications

This section describes the specifications required for timber design. Detailed description of the timber design procedures is available in Section 4.

Section 5.51.1 describes specification of parameters for timber design. Sections 5.51.2 and 5.51.3 discusses the code checking and member selection facilities respectively.

5.51.1 Timber Design Parameter Specifications

Purpose

This set of commands may be used for specification of parameters for timber design.

General Format:

PARAMETER

CODE TIMBER

$$\textbf{parameter-name } f_1 \left\{ \begin{array}{l} \textbf{MEMBER member-list} \\ \textbf{ALL} \end{array} \right\}$$

Description

f_1 = the value of the parameter.

The parameter-name refers to the parameters described in Section 4.

Notes

1) All values must be provided in the current unit system.

2) For default values of parameters, refer to Section 4.

5.51.2 Code Checking Specification

Purpose

This command performs code checking operation on specified members based on the American Institute of Timber Construction (AITC) codes.

General Format:

$$\underline{\textbf{CHE}}\textbf{CK}\quad \underline{\textbf{COD}}\textbf{E}\quad \left\{\begin{array}{l}\underline{\textbf{MEM}}\textbf{BER member-list}\\ \underline{\textbf{ALL}}\end{array}\right\}$$

Description

This command checks the specified members against the requirements of the American Institute of Timber Construction (AITC) codes. The results of the code checking are summarized in a tabular format. Examples and detailed explanations of the tabular format are available in Section 4.

Notes

The output of this command may be controlled by the TRACK parameter. Two levels of details are available. Refer to Section 4 for detailed information on the TRACK parameter.

5.51.3 Member Selection Specification

Purpose

This command performs member selection operation on specified members based on the American Institute of Timber Construction (AITC) codes.

General Format:

SELECT { **MEMBER member-list** / **ALL** }

Description

This command may be used to perform member selection according to the AITC codes. The selection is based on the results of the latest analysis and iterations are performed until the least weight member satisfying all the applicable code requirements is obtained. Parameters may be used to control the design and the results are available in a tabular format. Detailed explanations of the selection process and the output are available in Section 4.

Notes

The output of this command may be controlled by the TRACK parameter. Two levels of details are available. Refer to Section 4 for detailed information on the TRACK parameter.

5.52 Concrete Design Specifications for beams, columns and plate elements

This section describes the specifications for concrete design for beams, columns and individual plate elements. The concrete design procedure implemented in STAAD consists of the following steps:

1) Initiating the design.
2) Specifying parameters.
3) Specifying design requirements.
4) Requesting quantity take-off.
5) Terminating the design.

Section 5.52.1 describes the design initiation command. Section 5.52.2 discusses the specification of parameters. Design requirement specifications are described 5.52.3. The CONCRETE TAKE OFF command is described in 5.52.4. Finally, the design termination command is described in 5.52.5.

5.52.1 Design Initiation

Purpose

This command is used to initiate concrete design for beams, columns and individual plate elements.

General format:

START CONCRETE DESIGN

Description

This command initiates the concrete design specification. With this, the design parameters are automatically set to the default values (as shown on Table 3.1). Without this command, none of the following concrete design commands will be recognized.

Notes

This command must be present before any concrete design command is used.

Related topics can be found in the following sections:

Design of beams, columns and elements – theoretical background per ACI 318 -	3.1 to 3.8.1
Design of footings per ACI 318 -	5.52
Design of shear walls per ACI 318 -	3.8.2, 5.53
Design of slabs per ACI 318 using the Slab Designer -	3.8.3

5.52.2 Concrete Design-Parameter Specification

Purpose

This set of commands may be used to specify parameters to control concrete design for beams, columns and individual plate elements.

General format:

CODE
ACI
AUSTRALIA
BRITISH
CANADIAN
CHINA
EUROPE
FRENCH
GERMAN
INDIA
JAPAN
MEXICO
NORWAY

parameter-name f_1
MEMBER memb/elem list
(ALL)

Description

Parameter-name refers to the concrete parameters described in Table 3.1.

f_1 = is the value of the parameter. Wherever applicable, this value is input in the current units. The UNIT command is also accepted during any phase of concrete design.

Notes

1) All parameter values are provided in the current unit system.

2) For default values of parameters, refer to Section 3 for the ACI code. For other codes, please see the International Codes manual.

5.52.3 Concrete Design Command

Purpose

This command may be used to specify the type of design required. Members may be designed as BEAM, COLUMN or ELEMENT.

General format:

```
              { BEAM    }   { memb-list }
DESIGN        { COLUMN  }   { ( ALL )   }
              { ELEMENT }
```

Description

Members to be designed must be specified as BEAM, COLUMN or ELEMENT. Members, once designed as a beam, cannot be redesigned as a column again, or vice versa.

Notes

Only plate elements may be designed as ELEMENT.

Enter this command after the parameters needed for this command have been entered.

The DESIGN ELEMENT command designs individual plate elements using the procedure explained in section 3.8.1 of the ACI code. For theoretical information on designing individual plate elements per other design codes, please see the International Codes manual. For designing shear walls and slabs, see sections 3.8.2, 3.8.3 and 5.54 of this manual.

5.52.4 Concrete Take Off Command

Purpose

This command may be used to obtain an estimate of the total volume of the concrete, reinforcement bars used and their respective weights.

General Format:

CONCRETE TAKE OFF

Description

This command can be issued to print the total volume of concrete and the bar numbers and their respective weight for the members designed.

SAMPLE OUTPUT:

```
************** CONCRETE TAKE OFF **************
(FOR BEAMS AND COLUMNS DESIGNED ABOVE)

TOTAL VOLUME OF CONCRETE       =    87.50

BAR SIZE              WEIGHT
NUMBER                (     )
--------              --------
   4                   805.03
   6                    91.60
   8                  1137.60
   9                   653.84
  11                   818.67
------------
*** TOTAL     =       3506.74
```

Notes

This command may be used effectively for quick quantity estimates.

5.52.5 Concrete Design Terminator

Purpose

This command must be used to terminate the concrete design.

General format:

END CONCRETE DESIGN

Description

This command terminates the concrete design, after which normal STAAD commands resume.

Example

```
START CONCRETE DESIGN
CODE ACI
FYMAIN 40.0 ALL
FC 3.0 ALL
DESIGN BEAM 1 TO 4 7
DESIGN COLUMN 9 12 TO 16
DESIGN ELEMENT 20 TO 30
END
```

Notes

Without this command, further STAAD commands will not be recognized.

5.53 Footing Design Specifications

Removed. Please contact the Technical Support department for more information.

5.54 Shear Wall Design

Purpose

STAAD performs design of reinforced concrete shear walls per two codes currently: ACI 318-02 and BS 8110. In order to design a shear wall, it must first be modelled using the Surface element.

The attributes associated with surfaces, and the sections of this manual where the information may be obtained, are listed below:

Attributes	Related Sections
Surfaces incidences -	5.13.3
Openings in surfaces -	5.13.3
Local coordinate system for surfaces -	1.6.3
Specifying sections for stress/force output -	5.13.3
Property for surfaces -	5.21.2
Material constants -	5.26.3
Surface loading -	5.32.3.4
Stress/Force output printing -	5.42
Shear Wall Design -	3.8.2, 5.54

Scope of the design is dependent on whether panel definition, as explained in section 5.53.1, precedes the shear wall design input block which is explained in section 5.53.2.

a. No panel definition.

   Design is performed for the specified horizontal full cross-section, located at a distance c from the origin of the local coordinates system.

b. Panels have been defined.

Design is performed for all panels, for the cross-section located at a distance c from the start of the panel.

Shear Wall design is currently not available for dynamic load cases.

5.54.1 Definition of Wall Panels for Shear Wall Design

Due to the presence of openings, three types of structural elements may be defined within the boundaries of a shear wall: wall, column, and beam. For each of those entities, a different set of design and detailing rules applies. Users assign those types to panels - functionally different parts of the shear wall. The assignment is based on panel geometry, its position, and overall wall configuration. Wall, column and beam panels are then designed in accordance with relevant provisions of the code.

General format:

START PANEL DEFINITION
SURFACE i PANEL j ptype x1 y1 z1 x2 y2 z2 x3 y3 z3 x4 y4 z4
END PANEL DEFINITION

where:

I - ordinal surface number,
j - ordinal panel number,
ptype - panel type, one of: WALL, COLUMN, BEAM
x1 y1 z1 (...) - coordinates of the corners of the panel

5.54.2 Shear Wall Design Initiation

General format:

START SHEARWALL DESIGN
CODE a
parameters
DESIGN SHEARWALL (AT c) LIST s
CREINF cr
TRACK tr
END SHEARWALL DESIGN

where:

a - code name – ACI (for ACI 318), BRITISH (for BS 8110)

parameters - these are listed in a tabular form in section 3.8.2 of this manual for the ACI code, and in the International Codes manual for BS 8110.

cr - column reinforcing parameter,

tr - output parameter.

(see below for description)

Reinforcement distribution within COLUMN panels is controlled by parameter CREINF, that may have one of three possible values:

0 - equal number of bars on all four faces of the column (default),
1 - equal number of bars on two faces perpendicular to the plane of the wall,
2 - equal number of bars on two faces parallel to the plane of the wall.

Parameter TRACK specifies how detailed the design output should be:

0 - indicates a basic set of results data (default),
1 - full design output will be generated.

Note: If the command AT is omitted, the design proceeds for all cross sections of the wall or panels, as applicable, defined by the SURFACE DIVISION X or SURFACE DIVISION Y input values.

5.55 End Run Specification

Purpose

This command must be used to terminate the STAAD run.

General format:

FINISH

Description

This command should be provided as the last input command. This terminates a STAAD run.

N
o
t
e
s

N
o
t
e
s

Index by Section Numbers

A

AASHTO,

ASD, 2.13.1

allowable code stress, 2.13.1

axial stress, 2.13.1

bending-axial stress interaction, 2.13.1

bending stress, 2.13.1

general comments, 2.13.1

MOVING LOAD, 1.17.1, 5.31.1, 5.32.12

shear stress, 2.13.2

LRFD, 2.13.2

axial Strength, 2.13.2

bending strength, 2.13.2

bending-axial interaction, 2.13.2

design parameters, 2.13.2

general comment, 2.13.2

shear strength, 2.13.2

Accidental seismic load,(see IBC 2006, IS 1893, etc…)

ACI CODE (see Concrete Design),

ASCE 7-02, seismic,5.31.2.6

ASCE 7-02, snow load,5.32.13, 1.17.4,5.31.5

ASCE 7-02, wind (see wind load generation)

Advanced Solver,

description, 1.18.1, 1.18.2.2,

AIJ (Seismic)-5.31.2.4

AISC CODE

allowables, 2.3

bending stress, 2.3.4

CODE CHECK, 2.5

combined compression and bending, 2.3.5

compressive stress, 2.3.3

design parameters, 2.4

member selection, 2.6

plate girders, 2.10,

shear stress, 2.3.2

steel table, 1.7.3, 5.20.1

tension stress, 2.3.1

torsion per Pub.T114, 2.3.7

truss members, 2.7

unsymmetric members, 2.8

AISC UNIFIED

ASD,2.17.1

Axial compression, 2.17.4

Axial tension, 2.17..3

Code checking, 2.17.9

Combined forces and Torsion, 2.17.7

Design parameters, 2.17.8, Table-2.9

Flexural design strength, 2.17.5

General comments, 2.17.1

LRFD, 2.17.1

Member selection, 2.17.9

Section classification, 2.17.2

Shear, 2.17.6

Specification, 2.17
Tabulated results, 2.17.10
AISI
Code checking, 2.15
Design parameter, Table-2.7
Design procedure,2.15
General, 2.15
Member selection, 2.15
Properties, 2.15
Steel section library, 2.15
AITC
Code checking,4.4
Combined bending and
Axial stress, 4.2
Design commands, 4.1
Design parameters, Table. 4.2
Design operation, 4.2
Implementation, 4.1
Input specification, 4.3
Member selection, 4.6
Naming, 4.1
Orientation, 4.5
Output, 4.6
Properties, 4.1
Algerian seismic,(see RPA)
ALPHA, 1.13, 5.26.1, 5.32.6
Alphanumerics, 5.1.1
Aluminum TAKE OFF,5.50
ANALYSIS
assumption of, 1.18.1
buckling ,1.18.2.2,5.37.4
dynamic analysis, 1.18.3
non-linear, 1.18.2.3,5.37.3
P-DELTA, 1.18.2.1, 5.37.2
PERFORM, 1.18, 5.37
pushover, 1.18.3.7
printing (see PRINT)
RESPONSE SPECTRUM, 1.18.3.4
static analysis, 1.18
Time-History, 1.18.3.5, 5.31.4 5.32.10.2
ANALYSIS FACILITIES, 1.18
ANGLE,
from User Table, 5.19
ASSIGN of, 1.7.5, 5.20.5
BETA, 1.5.3
specification of, 2.2.1
user table, 1.7.3, 5.19, 5.20.4
AREA LOAD
command, 1.16.3, 5.32.4
definition of, 1.16.3
ARRIVAL TIME, 5.31.4
ASME(see curved member)
ASSIGN command, 1.7.5, 5.20.5
ASSIGN PROFILE, 5.20.5
ASSUMPTIONS OF THE
ANALYSIS, 1.18.1
Axes
global coordinate, 1.5.1
local coordinate, 1.5.2
local and global relationship, 1.5.3

B

Bandwidth Reduction, 1.18.1, 5.11
BASIC EQUATION, 1.18.1
Basic Solver,
Description, 1.18.1,1.18.2.2.1
Specification,5.37.4
BEAM
ASSIGN, 1.7.5, 5.20.5

concrete DESIGN of, 3.6, 5.51.3
BEAM parameter, Table 2.1, 2.2,3.2
BENDING MOMENT/SHEAR FORCE, 1.19, 5.41
Bending stress (see AISC, AASHTO, & LRFD design)
BETA angle, 1.5.3, 5.26.2,5.42
command, 5.26.2
description, 1.5.3
BFP(see fireproofing)
BLOCK FIRE PROOFING,(see fireproofing)
BUCKLING, 1.18.2.2
advanced solver, 1.18.2.2.2
basic solver, 1.18.2.2.1
format, 5.37.4
Built-In Steel Section Library 1.7.2

C

CABLE
command of, 1.11, 5.23.2,
description, 1.11
nonlinear, 1.18.2.3,5.37.3
CALCULATE RAYLEIGH
(FREQUENCY), 5.34
Camber,
Format, 5.26.6
Purpose, 5.26.6
CANADIAN SEISMIC,
NRC (1995), 5.31.2.10
NRC (2005), 5.31.2.11
Cartesian Coordinate System, 1.5.1
CASTELLATED BEAMS, 2.2.1, 2.16
CB parameter, Table 2.1 & 2.2
Center of Gravity (CG)(See PRINT)
CFP (see fire proofing)
CFE seismic load,5.31.2.7
CHANGE
command of, 5.38
(see also Multiple Analysis, INACTIVE command)
CHANNEL SECTION
ASSIGN, 1.7.5, 5.20.5
double channels, 2.2.1
Specification, 2.2.1
user table property of, 1.7.3, 5.19
CHECK CODE, 5.48.2, 5.51.2
CHECK SOFT STOREY(see IS 1893)
CMY parameter, Table 2.1
CMZ parameter, Table 2.1
CODE CHECKING
specification, 5.48.2, 5.51.2
Cold Formed Steel, 2.15, 5.20
COLOMBIAN (Seismic),5.31.2.3
COLUMN
ASSIGN, 5.20.5
concrete DESIGN, 5.52.3
COMPOSITE BEAM, design per AISC-ASD, 2.9,
AISC-LRFD, 2.14.13
COMPOSITE BEAMS AND COMPOSITE DECKS, 1.7.7,
Definition, 5.20.7
Design parameters, Table-2.2
COMPOSITE DAMPING, 5.26.1
CONCRETE DESIGN- 3.0
Beam, 3.6

Column, 3.7
Design, 3.1
Dimension, 3.3
Element, 3.8.1
parameter ,3.4, 5.51.2, Table 3.1
RC designer, 3.8.3
Shear wall, 3.8.2
Slabs, 3.8.3
specifications, 5.52
terminator, 5.52..5
CONCRETE TAKE OFF, 5.52.4
CONSTANTS, 5.26
CONTOUR FIRE PROOFING
(see fire proofing)
COORDINATE
systems, 1.5
joint, 5.11
CQC, Modal Combination, 5.32.10.1
CURVED MEMBERS, 1.7.8
specification, 5.20.8
CUT-OFF FREQUENCY, 5.30.1
CUT-OFF MODE SHAPE, 5.30.1
CUT-OFF TIME, 5.30.1
CYLINDRICAL REVERSE, 5.11, 1.5.1

D

DAMPING, 5.26, 5.32.10.2
DEFINE, 5.26
DEFINE IMPERFECTION, 5.26.6
DEFINE MESH, 5.14.1
DEFINE MOVING LOAD, 5.31.1
DEFINE SNOW LOAD, 5.31.5
DEFINE TIME HISTORY, 5.31.4
DEFINE WIND LOAD, 5.31.5
DELETE MEMBERS, 5.18
DELETE JOINTS, 5.18
DESIGN - CONCRETE, 3
DESIGN BEAM, 5.51.3
DESIGN COLUMN, 5.51.3
DESIGN ELEMENT, 5.51.3
DESIGN FOOTING, 5.52.3
DESIGN initiation, 5.52.1
DESIGN OF I-SHAPED BEAMS PER ACI-318, 3.8.4
DFF parameter, Table 2.1
DJ1, DJ2 parameters, Table 2.1
DIAPHRAGM (see MASTER/SLAVE)
DIRECT ANALYSIS,
Analysis, 1.18
Format, 5.37.5
Definition, 5.31.7
DIRECTION, 5.27.3
DISPLACEMENTS
PRINT command, 5.41, 5.42
DMAX parameter, Table 2.1& 2.2
DMIN parameter, Table 2.1& 2.2
DOUBLE ANGLE, 2.2.1, 5.19
DOUBLE CHANNELS, 2.2.1
DRAW, 5.29
DRIFT(see imperfection)
DYNAMIC ANALYSIS, 1.18.3,5.30

E

ELASTIC MAT Automatic Spring Support Generator, 5.27.3
ELEMENT, 1.6.1
ELEMENT INCIDENCE, 5.13
ELEMENT LOAD, 5.32.3, 1.6.1, 1.16.8

ELEMENT LOCAL AXIS, 1.6.1

ELEMENT NUMBERING, 1.6.1

ELEMENT PLANE STRESS, 5.24

ELEMENT PROPERTY, 5.21

ELEMENT RELEASE, 5.22.2

ELEMENT WEIGHT, 5.31.2.1, 5.31.2.2

END CONCRETE DESIGN, 5.52.5

END RUN SPECIFICATION, 5.55

ENVELOPE,LOAD,5.40

EPSILON (see imperfection)

EXPOSURE FACTOR, WIND LOAD GENERATOR, 5.31.5,5.32.12

F

FINISH, 5.55

FINITE ELEMENT
- Information, 1.6
- Plate/shell, 1.6.1, 5.13, 5.21
- Solid, 1.6.2, 5.13

FIREPROOFING ON MEMBERS, 5.20.9

FIXED END MEMBER LOAD, 1.16.4, 5.32.7

FLEXIBILITY FACTOR (see Curved Member)

FLOOR HEIGHT(see IS 1893)

FLOOR LOAD, 5.32.4

FLOOR WEIGHT, 5.31.2.1, 5.31.2.2

FOOTING Automatic Spring Support Generator, 5.27.3

FORCE ENVELOPES, 1.19.5, 5.45

FORCES
- PRINT command, 5.42

FREQUENCY, 5.34

FYLD parameter, Table 2.1& 2.2

G

GAMMA (see Curved Member)

GENERAL SECTION
- User Table, 5.19

GENERATE ELEMENT, 5.14.1

GENERATION OF LOADS, 5.32.12

GENERATION OF MOVING LOADS, 5.32.12, 1.17.1

GEOMETRY MODELING CONSIDERATIONS, 1.6.1

GLOBAL COORDINATE, 1.5.1, 1.5.3

GROUND MOTION, 5.32.10.2

GROUP SPECIFICATION, 5.48

H

HARMONIC TIME HISTORY LOAD, 5.31.4

HARMONIC ANALYSIS, 5.37.6

I

IBC 2000/2003 LOAD, 5.31.2.6

IGNORE LIST, 5.9

IMPERFECTION Load, 1.18.2.4

IMPERFECTION, member, 5.26.6

INACTIVE MEMBERS, 1.20, 5.18

INCLINED SUPPORT, 5.27.2

INDIAN SEISMIC (see IS1893)

INPUT GENERATION, 1.2, 6.2

INPUT NODESIGN, 5.10

INPUT WIDTH, 5.4

IS1893, 5.31.2, 5

ISECTION
- User Table, 5.19

J

JAPANESE (Seismic) – see AIJ, 5.31.2.4

JOINT COORDINATES, 5.11

JOINT LOAD, 1.16.1, 5.32.1

JOINT NUMBERING, 5.11, 5.15

JOINT WEIGHT, 5.31.2

JOIST,(see AISI)

K

KG,

P-Delta, 1.18.2.1, 1.18.2.1.2

KINGSPAN,(see AISI)

KX parameter, Table 2.1 & 2.2

KY parameter, Table 2.1& 2.2

KZ parameter, Table 2.1& 2.2

L

LOAD COMBINATION, 5.35

LOAD COMBINATION, SRSS, 5.35

LOAD ENVELOPE,5.40

LOAD GENERATION, 1.17, 5.32.12

LOAD SYSTEMS

Definition of, 5.31

LOADING, 1.16, 5.32

LOCAL COORDINATE, 1.5.2, 1.5.3

LX parameter, Table 2.1 & 2.2

LY parameter, Table 2.1 & 2.2

LYSAGHT,(see AISI)

LZ parameter, Table 2.1 & 2.2

M

MAIN parameter, Table 2.1& 2.2

MASS MODELING, 1.18.3.2

MASTER/SLAVE, 1.15,

specification, 5.28

MAT Automatic Spring Support Generator, 5.27.3

MATERIAL CONSTANTS, 1.13

MEMBER CABLE, 5.23

MEMBER COMPRESSION - only, 5.23.3

MEMBER DISPLACEMENTS, 1.19.3

MEMBER END FORCES, 1.19

MEMBER FORCES, 1.19.2

MEMBER INCIDENCES, 5.12

MEMBER PROPERTY- FROM TABLE, 5.20

MEMBER - PRISMATIC, 5.20

MEMBER - TAPERED, 5.20

MEMBER - from User Provided Table, 5.20

MEMBER - ASSIGN, 5.20

MEMBER LOAD, 1.16.2, 5.32.2

MEMBER NUMBERING, 5.12, 5.15

MEMBER OFFSETS, 1.12, 5.25

MEMBER POSTSTRESS, 5.32.5

MEMBER PRESTRESS, 5.32.5

MEMBER PROPERTIES, 1.7, 5.20

MEMBER RELEASE, 1.8, 5.22

MEMBER SELECTION SPECIFICATION, 5.47.3

MEMBER STRESSES (SPECIFIED SECTIONS), 1.19.4, 5.41

MEMBER Tension-Only 1.9, 5.23.3

MEMBER TRUSS, 1.9, 5.23.1

MEMBER WEIGHT, 5.31.2

MESH GENERATION, 5.14

MEXICAN SEISMIC(see CFE,NTC)

MODAL CALCULATION, 5.34.2

MODE SELECTION, 5.30.2

MODE SHAPES (see PRINT)

Moving Load, 1.17.1, 5.31.1, 5.32.12

MULTIPLE ANALYSES, 1.20, 5.37

MultiLinear Analysis, 1.18.2.3, 5.27.4

Multilinear Spring, 5.1.2, 5.27.4

N

NATURAL FREQUENCY, 5.34

NBCC SEISMIC LOAD,5.31.2.10, 5.31.2.11

NON-LINEAR ANALYSIS, 1.18.2.2, 5.37

NOREDUCE BAND, 5.11

NOTIONAL LOAD(see direct analysis)

NRC SEISMIC LOAD,5.31.2.10, 5.31.2.11

NSF parameter, Table 2.1 & 2.2

NTC SEISMIC LOAD,5.31.2.8

O

OUTPUT WIDTH, 5.4

P

PAGE EJECT, 5.8

PAGE LENGTH, 5.8

PAGE NEW, 5.7

PARAMETER
- specifications,.52.2,5.51.1,5.48.1
- STEEL DESIGN, Table 2.1
- CONCRETE DESIGN, Table 3.1
- TIMBER DESIGN, 5.50, 5.50.1
- PARAMETERIC MESH,5.14.3

PARTIAL MOMENT RELEASE, 5.22.1

P-DELTA ANALYSIS, 1.18.2.1

PERFORM ANALYSIS, 5.37

PERFORM CABLE, 5.37

PERFORM IMPERFECTION, 5.37

PERFORM ROTATION, 5.17

PERFORM STEADY STATE, 5.37

PERIOD, 5.34

PHYSICAL MEMBER
- Definition, 5.16.2
- Description, 5.16.2
- Entities, 5.16
- Purpose, 5.16.2

PIPE, 1.7.2, 2.2.1,5.19, 5.20.1

PLATE/SHELL ELEMENT, 1.6.1, 5.13, 5.14, 5.21

PMEMBER(see physical member)

PROPERTIES
- members, 1.7, 5.20

PRESTRESS LOAD, 1.16.5, 5.32.5

PRINT
- ALL, 5.37
- BOTH, 5.37
- CG, 5.41
- ELEMENT FORCES, 5.42
- ELEMENT INFORMATION, 5.42
- ELEMENT STRESSES, 5.42
- ENTIRE (TABLE), 5.42
- FORCE ENVELOPE, 5.43
- JOINT DISPLACEMENTS, 5.42
- LOAD DATA, 5.37
- MATERIAL PROPERTIES, 5.42
- MAXFORCE ENVELOPE, 5.43
- MEMBER INFORMATION, 5.42
- MODE SHAPES, 5.37

SECTION DISPLACEMENT, 5.42
STATICS CHECK, 5.37
STATICS LOAD, 5.37
SUPPORT INFORMATION, 5.42
SUPPORT REACTIONS, 5.42
PRISMATIC, 5.19
PRISMATIC PROPERTY, 1.7.1, 5.20.2
PROFILE, 5.47.1
PROPERTY SPEC, 5.20.2
PUSH OVER ANALYSIS, 5.37.7

R

RADIUS(see Curved Member)
RATIO parameter, Table 2.1& 2.2
RAYLEIGH FREQUENCY, 5.34
RCECO,(see AISI)
REACTIONS (SUPPORT)
PRINT command, 5.41
REDUCTION FACTOR
PROPERTIES,5.20.10
REFERENCE LOAD, 5.33
DEFINITION, 5.33
GENERATION,5.33
REFERENCE POINT, 1.5.3
RELEASE
members, 1.8, 5.22.1
elements, 5.22.2
REPEAT, 511, 5.12, 5.13
REPEAT ALL, 5.11, 5.12, 5.13
REPEAT LOAD SPECIFICATION, 5.32.11
RESPONSE SPECTRUM, 1.18.3, 5.32.10
RESPONSE SPECTRUM –
Generic Method, 5.32.10.1.1.1
RESPONSE SPECTRUM –
IS 1893, 5.32.10.1.2
Eurocode 8 1996, 5.32.10.1.3
Eurocode 8 2004, 5.32.10.1.4
IBC 2006, 5.32.10.1.5
RESPONSE TIME HISTORY, 5.31.4, 5.32.10.2
RPA SEISMIC LOAD,5.31.2.9

S

SECONDARY ANALYSIS, 1.19.1
SECTION DISPLACEMENTS, 5.42
SECTION SPECIFICATION, 5.41
SEISMIC (see UBC, IBC)
SEISMIC (See GROUND MOTION)
SEISMIC (CFE), 5.31.2.7
SEISMIC (NTC), 5.31.2.8
SEISMIC (RPA), 5.31.2.9
SELECT ALL, 5.48.5, 5.50.3
SELECT MEMBER, 5.48.5, 5.50.3
SELECT OPTIMIZED, 5.48.5
SELECT WELD TRUSS, 5.48.5
SELFWEIGHT, 5.32.9
SEPARATOR, 5.6
SET NL, 5.5
SET DATA CHECK, 5.5
SET RUN, 5.5
SET ECHO, 5.5
SET Z UP, 5.5
SHEAR WALL, 3.8.2, 5.54
SINKING SUPPORT (see SUPPORT DISPLACEMENT)
SIZE
specification, 5.47

SJI JOIST,(see AISI)

SLAB DESIGN, 3.8

SLAVE RIGID, 5.28

SMALL DELTA,1.18.2.1.1

SNOW LOAD,

Definition,5.31.5,

Generation, 5.32.13

SOFT STOREY CHECKING(see IS 1893)

SOLID ELEMENT, 1.6.2, 5.13

SPECTRUM, 5.32.10.1

SRSS Modal Combination, 5.32.10.1

SRSS Static Load Combination, 5.35

SSY parameter, Table 2.1

SSZ parameter, Table 2.1

START CONCRETE DESIGN, 5.52.1

START USER TABLE, 5.19

STEADY STATE, 1.18.3.6, 5.37.6

STEEL DESIGN SPECIFICATIONS, 2.0, 5.48

STEEL JOIST AND JOIST GIRDERS, 1.7.6

STEEL TABLE, 5.20.1

STEEL TAKE-OFF, 5.50

STIFF parameter, Table 2.1

STRUCTURE GEOMETRY, 1.5

STRUCTURES – Type, 1.3, 5.2

SUBGRADE, 5.27.3

SUBSTITUTION - JOINT, 5.15

SUBSTITUTION - MEMBER, 5.15

SUBSTITUTION - COLUMN, 5.15

SUPPORT DISPLACEMENT, 1.16.7, 5.32.8

SUPPORTS, 1.14, 5.27

SUPPORTS - Inclined, 5.27.2

SURFACE DIVISION,5.13.3

SURFACE OPENING,5.13.3

SURFACE element, 1.6.3

SURFACE INCIDENCE, 5.13.3

T

TABLE - Steel Table , 1.7.3, 5.20.1

TABLE - User Provided, 5.19

TAPERED MEMBER, 5.20.3

TAPERED SECTIONS, 2.7.2, 5.20.3

TEES, 1.7.2, 5.19

TEMPERATURE LOAD, 1.16.6, 5.32.6

TEMPERATURE/STRAIN, 1.16.6

TENSION-ONLY MEMBER, 1.10, 1.18.2.4, 5.23.3

TIMBER DESIGN PARAMETER SPECIFICATIONS, 4.0, 5.50.1

TIMBER DESIGN SPECIFICATIONS, 4.0, 5.50

TIME HISTORY ANALYSIS, 1.18.3, 5.31.4, 5.32.10.2

TIME HISTORY ANALYSIS, Harmonic Loading, 5.31.4

TORSION parameter, Table 2.1

TRACK parameter, Table 2.1& 2.2

TRAPEZOIDAL LOAD, 5.32

TRUSS MEMBERS, 1.9, 5.23.1

TUBE, 1.7.2, 5.19

TURKISH SEISMIC LOAD,5.31.2.12

U

UBC

Accidental Torsion, 5.31.2

UBC LOAD, 5.31.2

UBC SEISMIC LOAD, 1.17.2, 5.31.2.1, 5.32.12

UNB parameter, Table 2.1 & 2.2

UNIFIED(see AISC UNIFIED)

UNIT, 5.3

UNIT SPECIFICATION, 5.3

UNIT SYSTEMS, 1.4

UNT parameter, Table 2.1 & 2.2

USER PROVIDED TABLE, 1.7.3, 5.19, 5.20.4

USER STEEL TABLE, 5.19

W

WELD parameter, Table 2.1

WELDED PLATE GIRDERS, 1.7.2

WIDE FLANGE, 5.19

WIND LOAD, 1.17.3, 5.31.5, 5.32.12

WMIN parameter, Table 2.1

WSTR parameter, Table 2.1

*N
o
t
e
s*

Notes