GT STRUDL ®

Version 29.1

Release Guide

Volume 1 of 1

June 2007

Computer-Aided Structural Engineering Center
School of Civil & Environmental Engineering
Georgia Institute of Technology
Atlanta, Georgia 30332-0355
U.S.A.

Telephone: (404) 894-2260
Fax: (404) 894-8014
e-mail: casec@ce.gatech.edu

NOTICES

This GTSTRUDL$_{\circledR}$ Release Guide is applicable to Version 29.1, with a release date in the GTSTRUDL title block of June 2007.

The GTSTRUDL$_{\circledR}$ computer program is proprietary to, and a trade secret of the Georgia Tech Research Corporation, Atlanta, Georgia, U.S.A.

GTMenu and its documentation were developed as an enhancement to GTSTRUDL authored by the Computer-Aided Structural Engineering Center, Georgia Institute of Technology.

DISCLAIMER

NEITHER GEORGIA TECH RESEARCH CORPORATION NOR GEORGIA INSTITUTE OF TECHNOLOGY MAKE ANY WARRANTY EXPRESSED OR IMPLIED AS TO THE DOCUMENTATION, FUNCTION, OR PERFORMANCE OF THE PROGRAM DESCRIBED HEREIN, AND THE USER OF THE PROGRAM IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE PROGRAM IN THEIR OWN ENVIRONMENT.

Commercial Software Rights Legend

Any use, duplication, or disclosure of this software by or for the U.S. Government shall be restricted to the terms of a license agreement in accordance with the clause at DFARS 227.7202-3 (June 2005).

This material may be reproduced by or for the U.S. Government pursuant to the copyright license under the clause at DFARS 252.227-7013, September 1989.

S)))))))))))))))))Q
GTSTRUDL® is a registered service mark of the Georgia Tech Research Corporation, Atlanta, Georgia USA.
© Windows Vista, Windows XP, Windows 2000, Windows NT, Windows ME, and Windows 98 are registered trademarks of Microsoft Corporation, Redmond, Washington.
©Excel is a registered trademark of Microsoft Corporation, Redmond, Washington.

Table of Contents

CHAPTER 1

CHAPTER 2 NEW FEATURES IN VERSION 29.1

CHAPTER 3 ERROR CORRECTIONS

CHAPTER 4 KNOWN DEFICIENCIES

CHAPTER 5 PRERELEASE FEATURES

Chapter 1

Introduction

Version 29.1 covers GTSTRUDL operating on PC's under the Windows Vista, XP and Windows 2000 operating systems. Chapter 2 presents the new features and enhancements which have been added since the release of Version 29. Chapter 3 provides you with details regarding error corrections that have been made since the Version 29 release. Chapter 4 describes known problems with Version 29.1. Chapter 5 describes prerelease features -- new features which have been developed and subjected to limited testing, or features for which the user documentation have not been added to the GTSTRUDL User Reference Manual. The command formats and functionality of the prerelease features may change before they become supported features based on additional testing and feedback from users.

The Prerelease features are subdivided into Design, Analysis, and General categories. The features in these categories and their sections numbers in Chapter 5 are shown below:

5.2 Design Prerelease Features
  5.2.1 LRFD3 Steel Design Code and Parameters
  5.2.2 BS5950 Steel Design Code and Parameters
  5.2.3 Steel Design by Indian Standard Code IS800
  5.2.4 ACI Code 318-99
  5.2.5 Rectangular and Circular Concrete Cross Section Tables
  5.2.6 ASD9-E Code
  5.2.7 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

5.3 Analysis Prerelease Features
  5.3.1 Calculate Error Estimate Command
  5.3.2 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

5.4 General Prerelease Features

- 5.4.1 Rotate Load Command
- 5.4.2 Coutput Command
- 5.4.3 Reference Coordinate System Command
- 5.4.4 Hashing Algorithm to Accelerate Input Processing
- 5.4.5 GTMenu Point coordinates and Line Incidences Commands

We encourage you to experiment with these prerelease features and provide us with suggestions to improve these features as well as other GTSTRUDL capabilities.

Please note that the features and overall appearance of GTMenu have not been changed for Version 29.1 and are therefore still described in Volume 2 of the Version 29 Release Guide. The GTMenu Release Guide is available under Help in the GTSTRUDL Output Window (Help - Reference Documentation - GTMenu).

Chapter 2

New Features in Version 29.1

This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GTSTRUDL in Version 29.1. This release guide is also available online upon execution of GTSTRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 Static Analysis

1. The GTSES sparse equation solver, first implemented under the STIFFNESS ANALYSIS GTSES command in Version 29, has been implemented as a stand-alone program. When executed as such, the GTSES sparse equation solver is able to allocate all available virtual memory to its own execution process, thereby increasing the efficiency of the equation solution beyond the improvements already made by the STIFFNESS ANALYSIS GTSES command.

   The STIFFNESS ANALYSIS GTSES command also stores the results of the analysis (joint displacements, member and finite element forces, finite element stresses and strains, reactions and resultant joint loads) in files in the current working directory, further increasing the size of static analysis models that can be solved and the efficiency with which they are solved.

   An example of a large model execution which completed in Version 29.1 but ran out of memory in Version 29 is shown below:

Number of Joints	40042
Number of Members	3055
Number of Elements	41332
Number of Loadings	50
Number of Loading Combinations	48
Average Bandwidth + Standard Deviation	584

   Time to solve using GTSES for 240,252 degrees of freedom = 197 seconds
   Total STIFFNESS ANALYSIS time = 767.04 seconds

2.2 General

1. The AREA LOAD command has improved geometrical error detection and reporting. In addition, the total area and applied load are now printed as an additional verification tool.

2.3 Finite Elements

1. The output from the DESIGN SLAB command was modified to display a full listing of elements selected and used in the computation of the total moment acting on a cut section, for both the Wood & Armer and element force algorithms. The DESIGN SLAB command remains a prerelease feature in Version 29.1 and is described in Section 5.2.7 of this Release Guide.

CHAPTER 3

ERROR CORRECTIONS

This chapter describes changes that have been made to GTSTRUDL to correct errors. These errors may have produced aborts, incorrect results, or restricted use of a feature in previous versions of GTSTRUDL. The error corrections are discussed by the primary feature areas of GTSTRUDL.

3.1 Dynamics

1. The abort that may occur when the external file solver is used in conjunction with the COMPUTE RESPONSE SPECTRUM command has been corrected. (GPRF 2007.03)

2. COMPUTE RIGID BODY MODES is no longer turned on by default when using the new GTSELANCZOS eigensolver. The user must now specify COMPUTE RIGID BODY MODES under the EIGENPROBLEM PARAMETERS command with all eigensolvers. (No GPRF issued)

3.2 Elastic Buckling

1. Elastic Buckling Analysis will no longer produce an incorrect eigenvalue (buckling multiplier) and buckled mode shape under the following conditions:

    a. The model contained a mixture of space frame and space truss members and one or more of the space frame members had member releases.

    b. The space truss members were created after the space frame members with member releases.

    For the cases tested in Version 29.0, the incorrect results were obvious as a buckling multiplier of 0.0 was computed. (GPRF 2007.05)

3.3 Finite Elements

1. The LIST STRESSES command now outputs correct curved element nodal forces for loading conditions that remain active through two or more successive STIFFNESS ANALYSIS command executions during which the structural model changes. (GPRF 2007.02)

2. The following corrections were made to the DESIGN SLAB command (See Section 5.2.7 of this Release Guide) which was a prerelease feature in Version 29 and remains a prerelease feature in Version 29.1:

   a. The Wood and Armer element selection algorithm was updated to follow the same selection rules as the Element Force element selection algorithm, meaning that only elements geometrically located between normals originating at the start and end nodes of the cut are incorporated in the moment computation. Due to the requirement that all elements across a cut section have uniform thickness, this modification allows for a change in thickness at the starting and ending boundaries of the cut, as is the case if a cut is made across a drop panel. Previously, the Wood and Armer algorithm averaged the moment resultants for all elements incident to the cut.

   b. In addition, the element selection algorithm now only performs the check for a cut that traverses across an element surface if the element has two nodes incident on the cut line. (Cuts may only be given along element boundaries.) This corrects the problematic case exhibited in an element attached with only one node to the end of a cut which may possess nodes that geometrically lie on both sides of the cut.

   c. The Wood and Armer moment computation was updated to correctly process the case where one or more loading conditions are inactive. This corrects the error of references to improper analysis results when intermediary loading conditions are inactive.

   d. In the design parameter summary, the format statement for printing the yield stress of the steel reinforcement (FY) was modified so that units of lb and ft do not cause truncation of the output.

   e. A correction was made to the command to allow the word “ELEMENT” to be specified twice as shown in the example below:

      ```
      DESIGN SLAB REINFORCEMENT USING CALCULATE -
      RESULTANT ELEMENT FORCES JOINTS ‘J1’ ‘J2’ ELEMENT ‘E1’
      ```

   f. An abort will no longer occur if nonlinear springs are located on the cut.

   g. Additional modifications were made to the tolerance used for determining the nodes located on the cut line and the tolerance used for determining which elements to use during the design moment computation. As originally implemented, the user may specify a linear tolerance on the command line. This linear tolerance, while originally used only to determine the cut bounds for element selection, is now also used to determine whether a node is located on the cut line, whether a given element is located in the same plane as the user-specified element, and for determining the cut bounds for element selection. If the user does not specify a linear tolerance, a default value of 0.1% the length of the cut (the distance between the two user-specified cut definition nodes) is applied.

h. The methodology used to compute the distance between the cut line and an arbitrary node was updated. The new computation uses a more robust and accurate evaluation of the distance between joints and the cut.

3.4 General

1. The MEMBERS | ELEMENTS | CABLES | NLS ONLY option for EXISTING has been corrected to work for all commands that accept EXISTING. Previously, some commands did not recognize this option, such as the WRITE command and the CALCULATE SOIL SPRING command. (No GPRF issued)

3.5 GTMenu

(*GPRF's are not issued for GTMenu unless specifically noted below*)

1. Label Member End Forces no longer produces incorrect results when there is an inactive member or if the model contains members and finite elements.

2. Inactive members are no longer used to compute the size of the graphics window. In earlier versions, inactive members could cause the structure to appear in only a small portion of the screen since the inactive members were used in computing the size of the structure.

3. The tolerance used in GTMenu for determining the special case of the Beta angle (member parallel to Global Y) has been corrected so it is now consistent with the tolerance used in analyses.

4. The following element types may now be used when creating new elements using the Create Elements Only option: CSTG, UTLQ1, LST and PSR. Previously, a pop-up error dialog would appear stating “Element Type Not Defined” while generating the elements when these element types were selected or the elements would not be created correctly.

5. Material constants having a value of 0.0 are now translated correctly.

3.6 Static Analysis

1. The STIFFNESS ANALYSIS command will no longer abort or produce numerous statics check error messages indicating severe equilibrium problems when a structural model contains ELASTIC CONNECTION member releases and the active load list for the stiffness analysis contains load combinations. (GPRF 2007.01)

3.7 Superelement

1. The SUPERELEMENT DEFINITION command has been improved so that the list of internal superelement members and finite elements can now consist of multiple sub lists, each beginning with the words MEMBERS and ELEMENTS as illustrated by the following example:

```
SUPERELEMENT DEFINITION

'SUP1' BOUDARY NODE 15 16 21 22 ELEMENT  25 26 27 28

'SUP2' BOUNDARY  NODE  8 TO 11 14 17 20 23 26 27 28 29  ELEMENT  7 TO 9 -
    MEMBERS 12 13 16 TO 18
```

Note that the definition for superelement ‘SUP2’ contains a list of internal members and finite elements that consists of two sub lists: ELE 7 TO 9 and MEMBERS 12 13 16 TO 18. In previous versions only the members in the last sub list, in this case MEMBERS 12 13 16 TO 18, would have been stored as the internal members of superelement ‘SUP2’. Now, all finite elements 7 to 9 and members 12, 13, and 16 to 18 are stored as the internal members and elements of ‘SUP2’. (No GPRF issued)

CHAPTER 4

KNOWN DEFICIENCIES

This chapter describes known problems or deficiencies in Version 29.1. These deficiencies have been evaluated and based on our experience, they are seldom encountered or there are workarounds. The following sections describe the known problems or deficiencies by functional area.

4.1 Finite Elements

1. The ELEMENT LOAD command documentation indicates that header information such as type and load specs are allowed. If information is given in the header and an attempt is made to override the header information, a message is output indicating an invalid command or incorrect information is stored. (GPRF 90.06)
2. Incorrect results (displacements, stresses, reactions, frequencies, ... etc.) will result if a RIGIDITY MATRIX is used to specify the material properties for the IPSL, IPSQ, and TRANS3D elements. (GPRF 93.09)
3. The CALCULATE RESULTANT command may either abort or print out an erroneous error message for cuts that appear to be parallel to the Planar Y axis. (GPRF 94.21)
4. If a superelement is given the same name as a member or finite element, an abort will occur in the DEVELOP STATIC PROPERTIES command. (GPRF 95.08)
5. The curved elements, TYPE ‘SCURV’ and ‘PCURV’ will produce incorrect results for tangential member loads (FORCE X). An example of the loading command which will produce this problem is shown below:

   ```
   LOADING 1
   MEMBER LOADS
   1  FORCE  X  UNIFORM   W  -10
   ```

   where member (element) 1 is a ‘SCURV’ or ‘PCURV’ element.
   (GPRF 99.13)

4.2 General Input/Output

1. An infinite loop may occur if a GENERATE MEMBERS or GENERATE ELEMENTS command is followed by a REPEAT command with an incorrect format. An example of an incorrect REPEAT command is shown below by the underlined portion of the REPEAT Command:

   ```
   GENERATE 5 MEM ID 1 INC 1 FROM 1 INC 1 TO 2 INC 1
   REPEAT 2 TIMES ID 5 FROM 7 INC 1 TO 8 INC 1
   ```

   Only the increment may be specified on the REPEAT command. (GPRF 93.22)

2. Rigid body elements can not be deleted or inactivated as conventional finite elements. The specification of rigid body elements as conventional finite elements in the INACTIVE command or in DELETIONS mode will cause an abort in a subsequent stiffness, nonlinear, or dynamic analysis. (GPRF 97.21)

3. The path plus file name on a SAVE or RESTORE is limited to 256 characters. If the limitation is exceeded, the path plus file name will be truncated to 256 characters. This is a Windows limitation on the file name including the path. (No GPRF issued)

4. Object groups, created by the DEFINE OBJECT command, may not be used in a GROUP LIST as part of a list. If the OBJECT group is the last group in the list, processing will be correct. However, if individual components follow the OBJECT group, they will fail. Also, you can not copy members or joints from the OBJECT group into a new group.

   (GPRF 99.26)

5. Numerical precision problems will occur if joint coordinate values are specified in the JOINT COORDINATES command with more than a total of seven digits. Similar precision problems will occur for joint coordinate data specified in automatic generation commands. (GPRF 2000.16)

6. Internal member results will be incorrect under the following conditions:

   1. Dynamic analysis is performed (response spectra or time history)
   2. Pseudo Static Loadings are created
   3. Buckling Analysis is Performed
   4. Internal member results are output or used in a subsequent steel design after the Buckling Analysis.

In addition, the eigenvalues and eigenvectors from the Dynamic Analysis are overwritten by the eigenvalues and eigenvectors from the Buckling Analysis.

We consider this problem to be very rare since we had never encountered a job which contained both a Dynamic Analysis and a Buckling Analysis prior to this error report.

Workaround:

Execute the Buckling Analysis in a separate run which does not contain a dynamic analysis.

Alternatively, execute the Buckling Analysis before the Dynamic Analysis and output the Buckling results and then perform a Dynamic Analysis. The Dynamic Analysis results will then overwrite the buckling multiplier and mode shape which is acceptable since the buckling results have been output and are not used in any subsequent calculations in GTSTRUDL.

(GPRF 2004.14)

4.3 GTMenu

1. Gravity loads and Self-Weight loads are generated incorrectly for the TRANS3D element.

   Workaround: Specify the self-weight using Body Forces under Element Loads. ELEMENT LOADS command is described in Section 2.3.5.4.1 of Volume 3 of the GTSTRUDL Reference Manual.

   (GPRF 95.18)

2. The Copy Model feature under Edit in the Menu Bar will generate an incorrect model if the model contains the TRANS3D element.

   Workaround: Use the DEFINE OBJECT and COPY OBJECT commands in Command Mode as described in Section 2.1.6.7.1. and 2.1.6.7.5 of Volume 1 of the GTSTRUDL Reference Manual.

   (GPRF 95.21)

4. The Load Summations option available under CHECK MODEL will produce incorrect load summations for line, edge, and body loads on all finite elements. The Load Summations are also incorrect for projected loads on finite elements. The load

summations for line and edge loadings should be divided by the thickness of the loaded elements. The body force summations should be multiplied by the thickness of the loaded elements.

Workaround: You can check the load summation by specifying the LIST SUM REACTIONS command after STIFFNESS ANALYSIS.
(No GPRF issued)

5. Projected element loads will be displayed incorrectly when they are created or when they are displayed using Display Model 6 Loads.

   Workaround: Verify that the loads are correct in the GTSTRUDL Output Window using the PRINT LOAD DATA command or by checking the reactions using LIST SUM REACTIONS.
   (No GPRF issued)

4.4 Rigid Bodies

1. Response spectrum analysis may abort if rigid bodies and/or joint ties with slave releases are present in the model. (GPRF 99.18)

2. Static and dynamic analyses will abort if member releases are specified for rigid bodies. (GPRF 2005.02)

4.5 Scope Environment

1. OVERLAY DIAGRAM in the Plotter Environment produces diagrams that are much smaller relative to the plot size than the Scope environment does. This is because the structure plot is magnified to fill the Plotter graphics area, but the height of the diagram is not increased. As a work-around, use the PLOT FORMAT SCALE command to decrease the scale factor, which will increase the size of the diagram. The current value is printed with a Scope Environment OVERLAY DIAGRAM. The value printed with a Plotter Environment OVERLAY DIAGRAM is incorrect. For example, if a Moment Z diagram is OVERLAYed with a scale factor of 100.0 on the Scope, the command PLOT FORMAT SCALE MOMENT Z 50. would scale a reasonable OVERLAY DIAGRAM for the Plotter.
   (GPRF 96.19)

CHAPTER 5

PRERELEASE FEATURES

5.1 Introduction

This chapter describes new features that have been added to GTSTRUDL but are classified as prerelease features due to one or more of the following reasons:

1. The feature has undergone only limited testing. This limited testing produced satisfactory results. However, more extensive testing is required before the feature will be included as a released feature and documented in the GTSTRUDL User Reference Manual.
2. The command formats may change in response to user feedback
3. The functionality of the feature may be enhanced in to response to user feedback.

The Prerelease features in Version 29.1 are subdivided into Design, Analysis, and General categories. The features in these categories are shown below:

- 5.2 Design Prerelease Features
  - 5.2.1 LRFD3 Steel Design Code and Parameters
  - 5.2.2 BS5950 Steel Design Code and Parameters
  - 5.2.3 Steel Design by Indian Standard Code IS800
  - 5.2.4 ACI Code 318-99
  - 5.2.5 Rectangular and Circular Concrete Cross Section Tables
  - 5.2.6 ASD9-E Code
  - 5.2.7 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

5.3 Analysis Prerelease Features

  - 5.3.1 Calculate Error Estimate Command
  - 5.3.2 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

5.4 General Prerelease Features

  - 5.4.1 Rotate Load Command
  - 5.4.2 Coutput Command
  - 5.4.3 Reference Coordinate System Command
  - 5.4.4 Hashing Algorithm to Accelerate Input Processing
  - 5.4.5 GTMenu Point Coordinates and Line Incidences Commands

We encourage you to experiment with these prerelease features and provide us with suggestions to improve these features as well as other GTSTRUDL capabilities.

5.2 Design Prerelease Features

5.2.1 LRFD3 Steel Design Code and Parameters

LRFD3 Code
American Institute of Steel Construction
Load and Resistance Factor Design
AISC LRFD Third Edition

LRFD3.1.2 LRFD3 Steel Design Code and Parameters

The LRFD3 code of GTSTRUDL may be used to select or check any of the following shapes:

Design for bi-axial bending and axial forces:

I shapes	Round Bars
Channels	Square Bars
Single Angles	Rectangular Bars
Tees	Plate Girders
Double Angles	

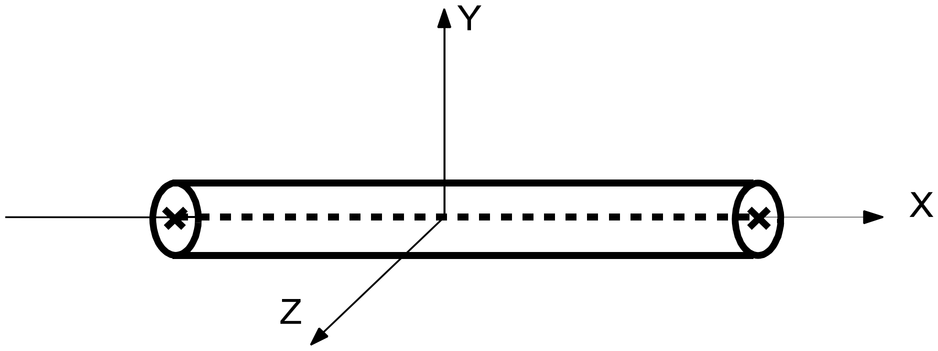
Design for bi-axial bending, axial, and torsional forces:

- Round HSS (Pipes)
- Rectangular and Square HSS (Structural Tubes)

The term I shapes is used to mean rolled or welded I and H beams and columns, universal beams and columns, joists, universal bearing piles, W, S, M, and HP profiles with doubly symmetric cross-sections.

The code is primarily based on the AISC “Load and Resistance Factor Design Specification for Structural Steel Buildings” adopted December 27, 1999 with errata incorporated as of September 4, 2001. The Specification is contained in the Third Edition of the AISC Manual of Steel Construction, Load and Resistance Factor Design (96). The LRFD3 code utilizes the Load and Resistance Factor design techniques of the AISC Specification.

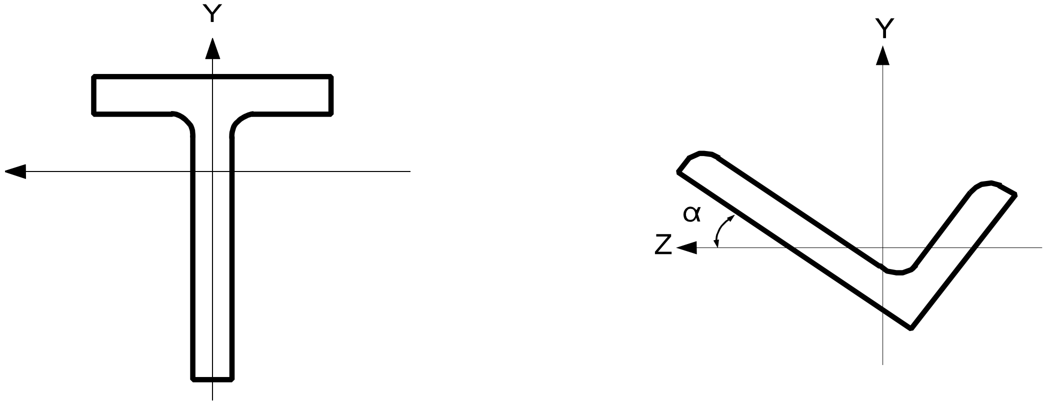
Second order elastic analysis using factored loads is required by the GTSTRUDL LRFD3 code. Second order effects may be considered by using GTSTRUDL Nonlinear Analysis (Section 2.5 or Volume 3 of the User Reference Manual). GTSTRUDL LRFD3 code check does not consider the technique discussed in Section C1.2 of AISC, *Manual of Steel Construction, Load & Resistance Factor Design, Third Edition*, for determination of M_u (B_1 and B_2 factors) in lieu of a second order analysis.



(a) Local Principle Axes Orientation

(b) I shapes and Plate Girders

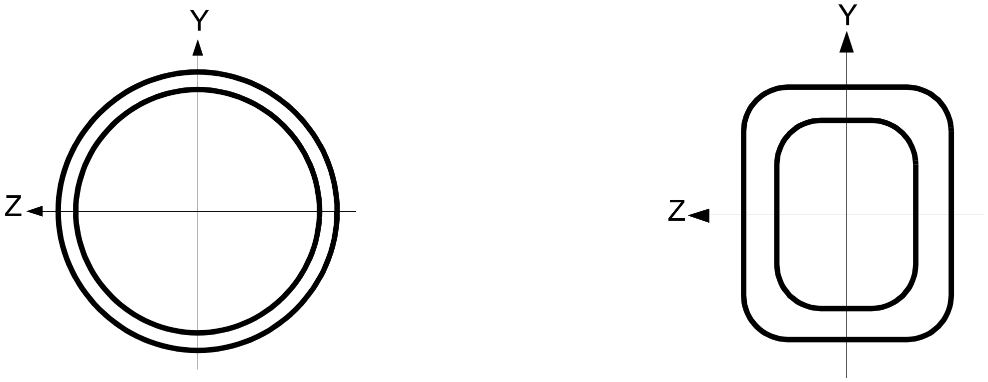
(c) Channels



(d) Tees

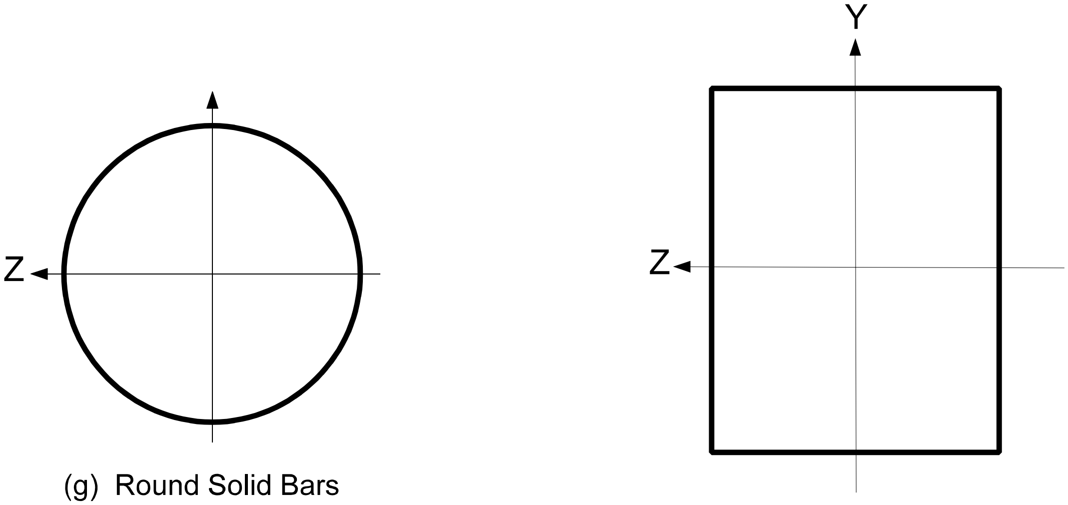
(e) Single angles

Figure LRFD3.1-1 Local Axes for Design with LRFD3



(f) Pipes

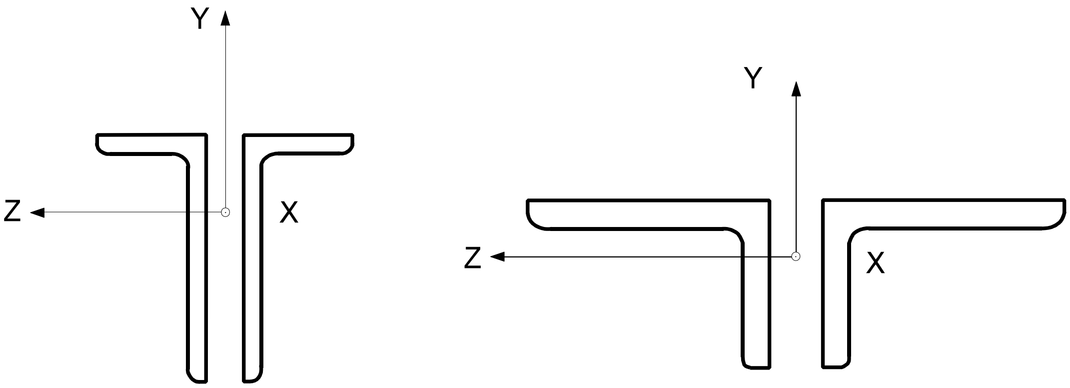
(g) Structural tubing



(g) Round Solid Bars

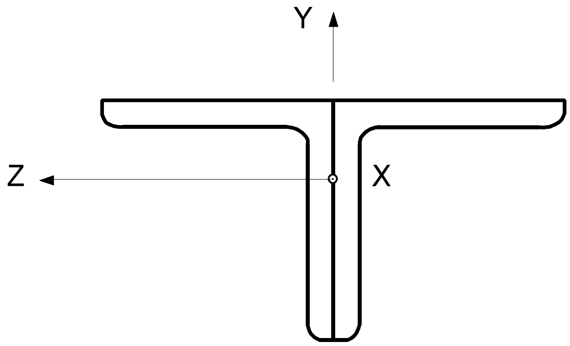
(h) Square and Rectangular Solid Bars

Figure LRFD3.1-1 Local Axes for Design with LRFD3 (continued)



(h) Long legs back-to-back
double angle with spacing

(i) Short legs back-to-back
double angle with spacing



(j) Equal legs back-to-back
double angle in contact

Figure LRFD3.1-1 Local Axes for Design with LRFD3 (Continued)

The following assumptions are made throughout the LRFD3 code.

1. Open cross-sections (I shapes, channels, single angles, double angles, tees, bars, and plate girders) are normally not used in situations wherein significant torsional moments must be resisted by the member. Torsional stresses are usually small for open cross-sections when compared to axial and bending stresses, and may be neglected. No checks are made for torsion in open cross-sections (I shapes, channels, single angles, double angles, tees, bars, and plate girders). The designer is reminded to check the torsional stresses for open cross-sections (I shape, channels, single angles, double angles, tees, bars, and plate girders) whenever they become significant.

2. Torsional stresses are checked for round HSS (pipes), rectangular and square HSS (structural tubes) based on the Section 6.1 on Page 16.2-8 of the AISC LRFD Third Edition. Combined torsion, shear, flexure, and/or axial forces are also checked for round HSS (pipes), rectangular and square HSS (structural tubes) based on the Section 7.2 on Page 16.2-10 of the AISC LRFD Third Edition. Closed cross-sections (HSS) are frequently used in situations wherein significant torsional moments must be resisted by the members. Generally the normal and shear stresses due to warping in closed cross-sections (HSS) are insignificant and the total torsional moment can be assumed to be resisted by pure torsional shear stresses (Saint-Venant's torsion).

3. Web stiffeners are considered for web shear stress, but they are not designed.

4. Modified column slenderness for double angle member is considered (Section E4 of the AISC LRFD Third Edition). Modified column slenderness of the double angle member is computed based on the user specified or designed number of the intermediate connectors.

5. Double angles contain an adequate number of intermediate connectors (stitch plates) which make the two angles act as one, Tee-like section.

The sections of the AISC LRFD Third Edition specifications (96) which are considered by the GTSTRUDL LRFD3 code are summarized below:

Section	Title
Chapter D	**Tension members**
Section B7	Limiting slenderness ratios
Section D1	Design tensile strength
Chapter E	**Columns and other compression members**
Section B7	Limiting slenderness ratios
Table B5.1	Limiting width to thickness ratio for unstiffened compression elements
Table B5.1	Limiting width to thickness ratio for stiffened compression elements
Section E2	Design compressive strength for flexural buckling
Section E3	Design compressive strength for flexural-torsional buckling
Section E4	Built-up member.
Section E4.1	Design strength. Modified column slenderness
Section E4.2	Detailing requirements
Appendix E	**Columns and other compression members**
Appendix E3	Design compressive strength for flexural-torsional buckling
Appendix B	**Design requirements**
Appendix B5.3a	Unstiffened compression elements
Appendix B5.3b	Stiffened compression elements
Appendix B5.3c	Design properties
Section B5.3d	Design strength
Chapter F	**Beam and other flexural members**
Section F1.1	Yielding
Section F1.2	Lateral-Torsional Buckling
Section F1.2a	Doubly symmetric shapes and channels with L_b # L_r
Section F1.2b	Doubly symmetric shapes and channels with $L_b > L_r$
Section F1.2c	Tees and Double angles
Appendix F	**Beams and other flexural members**
Appendix F1	Design for flexure
Table A-F1.1	Nominal strength parameters
Appendix F2	**Design for shear**
Appendix F2.2	Design shear strength
Appendix F2.3	Transverse stiffeners

Section	Title
Appendix G	**Plate Girders**
Appendix G1	Limitations
Appendix G2	Design flexural strength
Appendix G3	Design shear strength
Appendix G4	Transverse stiffeners
Appendix G5	Flexure-shear
Chapter H	**Member under combined forces**
Section H1	Symmetric members subject to bending and axial force
Section H1.1	Doubly and singly symmetric member in flexure and tension
Section H1.2	Doubly and singly symmetric member in flexure and compression

Load and Resistance Factor Design Specification for Single-Angle Members

Section	Title
Section 2	Tension
Section 3	Shear
Section 4	Compression
Section 5	Flexure
Section 5.1	Flexure Design Strength
Section 5.3	Bending About Principal Axes
Section 6	Combined Forces
Section 6.1	Members in Flexure and Axial Compression
Section 6.2	Members in Flexure and Axial Tension

Load and Resistance Factor Design Specification for Steel Hollow Structural Sections

Section	Title
Table 2.2-1	Limiting Wall Slenderness for Compression Elements
Section 3	Tension Members
Section 3.1	Design Tensile Strength
Section 4	Column and Other Compression Members
Section 4.2	Design Compressive Strength
Section 5	Beams and Other Flexural Members
Section 5.1	Design Flexural Strength

Section	Title
Section 5.2	Design Shear Strength
Section 6	Torsion Members
Section 6.1	Design Torsional Strength
Section 7	Members Under Combined Forces
Section 7.1	Design for Combined Flexure and Axial Force
Section 7.2	Design for Combined Torsion, Shear, Flexure, and/or Axial Force

Tensile or compressive axial strengths, bi-axial bending, shear strengths, and combined strengths are considered for all cross-sections. Parameters allowing for the changes which occur in structural steel at high temperatures have been included and may be invoked at the user's discretion.

The detailed explanation of the code parameters and cross-section properties are as follows.

1.	Table LRFD3.1-1	Shows the parameters used by LRFD3 code. Table LRFD3.1-1 contains the applicable parameter names, their default values, and a brief description of the parameters.
2.	Section LRFD3.2	Describes the cross-section properties used for each shape.
3.	Section LRFD3.3	Contains detailed discussion of the parameters used by the LRFD3 code and they are presented in the alphabetic order in this section.

Table LRFD3.1-1

LRFD3 Code Parameters

Parameter Name	Default Value	Meaning
CODE	Required	Identifies the code to be used for member checking or member selection. Specify LRFD3 for code name. Second order elastic analysis using factored loads is required by the GTSTRUDL LRFD3 code. Second order effect may be considered by using GTSTRUDL Nonlinear Analysis (Section 2.5 of Volume 3 of the User Reference Manual). See Sections LRFD3.2 and LRFD3.3 for a more detailed description of parameters and cross-section properties.
TBLNAM	WSHAPES9	Identifies the table of profiles to be used during selection (SELECT command). See Table LRFD3.1-3 for a list of available table names.
CODETOL	0.0	Percent variance from 1.0 for compliance with the provisions of a code. The ratio of Actual/Allowable must be less than or equal to [1.0 + CODETOL/100].
PF	1.0	Area reduction factor for holesout in members subject to axial tension.
a	10000.0 (inches)	The clear distance between transverse stiffeners. This parameter is used to compute a/h ratio which is used in the computation of the limiting shear strength. The default value indicates that the shear check does not consider transverse stiffeners.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
SECTYPE	Computed	Indicates that the cross-section is rolled or welded shape. This parameter is used to compute the value of F_r. F_r is the compressive residual stress in flange. ROLLED = rolled shape. Compressive residual stress is equal to 10 ksi. WELDED = welded shape. Compressive residual stress is equal to 16.5 ksi.
Material Properties		
STEELGRD	A36	Identifies the grade of steel from which a member is made. See Tables LRFD3.1-4 and LRFD3.1-5 for steel grades and their properties.
Fy	Computed	Yield stress of member. Computed from parameter ‘STEELGRD’ if not given.
Fu	Computed	Minimum tensile strength of member. Computed from parameter ‘STEELGRD’ if not given.
Fyf	Fy	Minimum yield stress of the flange. If not specified, it is assumed equal to the parameter ‘Fy’. This parameter is used to define a hybrid cross-section. See parameter ‘Fyw’ also.
Fyw	Fy	Minimum yield stress of the web. If not specified, it is assumed equal to the parameter ‘Fy’. This parameter is used to define a hybrid cross-section. See parameter ‘Fyf’ also.
RedFy	1.0	Reduction factor for parameter ‘Fy’. This factor times parameter ‘Fy’ gives the F_y value used by the code. Used to account for property changes at high temperatures.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	Default Value	Meaning
Material Properties (continued)		
RedFu	1.0	Reduction factor for parameter ‘Fu’. Similar to parameter ‘RedFy’.
REDE	1.0	Reduction factor for E, the modulus of elasticity. Similar to parameter RedFy.
Slenderness Ratio		
SLENCOMP	200	Maximum permissible slenderness ratio (KL/r) for a member subjected to axial compression. When no value is specified for this parameter, the value of 200 is used for the maximum slenderness ratio.
SLENTEN	300	Maximum permissible slenderness ratio (L/r) for a member subjected to axial tension. When no value is specified for this parameter, the value of 300 is used for the maximum slenderness ratio.
K-Factors		
COMPK*	NO	Parameter to request the computation of the effective length factors KY and KZ (Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual). YES = compute KY and KZ factors. KY = compute KY only. KZ = compute KZ only. NO = use default or specified values for KY and KZ.

*K-factor Leaning Columns Concept has not been implemented for the automatic K-factor Computation.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	Default Value	Meaning
K-Factors (continued)		
KY	1.0	Effective length factor for buckling about the local Y axis of the profile. See Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual for GTSTRUDL computation of effective length factor, KY.
KZ	1.0	Effective length factor for buckling about the local Z axis of the profile. See Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual for GTSTRUDL computation of effective length factor, KZ.
Print-K	YES	Parameter to print the computed K-factor values after the default code check or select command output (TRACE 4 output). The default value of 'YES' for this parameter indicates that the computed K-factor values should be printed after the code check or select command output. The column names attached to the start and end of the code checked member is also printed. This printed information allows the user to inspect the automatic detection of the columns attached to the start and end of the designed member. A value of 'NO' indicates that K-factor values and the names of the attached columns to the start and end of the designed member should not be printed.
SDSWAYY	YES	Indicates the presence or absence of sidesway about the local Y axis. YES = sidesway permitted. NO = sidesway prevented.

K-factor Leaning Columns Concept has not been implemented for the automatic K-factor Computation.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

K-Factors (continued)

Parameter Name	Default Value	Meaning
SDSWAYZ	YES	Indicates the presence or absence of sidesway about the local Z axis. YES = sidesway permitted. NO = sidesway prevented.
CantiMem	NO	Parameter to indicate that a member or a physical member which is part of a cantilever truss should be considered as a cantilever in the K-factor computation. True cantilever members or physical members are detected automatically. NO = member or physical member is not cantilever, YES = member or physical member is cantilever.
GAY	Computed	G-factor at the start joint of the member. GAY is used in the calculation of effective length factor KY (see parameters COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual).
GAZ	Computed	G-factor at the start joint of the member. GAZ is used in the calculation of effective length factor KZ (see parameters COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual).
GBY	Computed	G-factor at the end joint of the member. GBY is used in the calculation of effective length factor KY (see parameters COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual).
GBZ	Computed	G-factor at the end joint of the member. GBZ is used in the calculation of effective length factor KZ (see parameters COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual).

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Buckling Length		
LY	Computed	Unbraced length for buckling about the local Y axis of the profile. The default is computed as the length of the member.
LZ	Computed	Unbraced length for buckling about the local Z axis of the profile. The default is computed as the length of the member.
FRLY	1.0	Fractional form of the parameter LY, allows unbraced length to be specified as fractions of the total length. Used only when LY is computed.
FRLZ	1.0	Fractional form of the parameter LZ, similar to FRLY. Used only when LZ is computed.
Flexural-Torsional Buckling		
KX	1.0	Effective length factor for torsional buckling about the local X axis of the profile. This parameter is used in flexural-torsional buckling stress, F_e computations.
LX	Computed	Unbraced length for torsional buckling about the local X axis of the profile. The default is computed as the length of the member. This parameter is used in flexural-torsional buckling stress, F_e computations.
FRLX	1.0	Fractional form of the parameter LX, allows unbraced length to be specified as fractions of the total length. Used only when LX is computed.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Bending Strength		
CB	Computed	Coefficient used in computing allowable compressive bending strength (AISC LRFD Third Edition, Section F1.2a, Equation F1-3).
UNLCF	Computed	Unbraced length of the compression flange. The default is computed as the length of the member. In this parameter no distinction is made between the unbraced length for the top or bottom flange. See UNLCFTF or UNLCFBF.
FRUNLCF	1.0	Fractional form of the parameter UNLCF, allows unbraced length to be specified as fractions of the total length. Used only when UNLCF is computed.
UNLCFTF	Computed	Unbraced length of the compression flange for the top flange. When no value is specified, UNLCF and FRUNLCF are used for this parameter.
UNLCFBF	Computed	Unbraced length of the compression flange for the bottom flange. When no value is specified, UNLCF and FRUNLCF are used for this parameter.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Channel Parameter		
Tipping	YES	This is the parameter indicating that the tipping effect should be considered. When the load is applied to the top flange of the channel and the flange is not braced, there is a tipping effect that reduces the critical moment. A value of YES for this parameter indicates that the flange is unbraced and the flange is loaded as such that causes tipping effect. In this case the reduced critical moment may be conservatively approximated by setting the warping buckling factor X_2 equal to zero. A value of NO indicates that the tipping effect does not happen and the warping buckling factor is computed based on the Equation F1-9 of the AISC LRFD Third Edition.
Single Angle Parameter		
Cby	Computed	Coefficient used in computing elastic lateral-torsional buckling moment, M_{ob}, (AISC LRFD Third Edition, Section 5.3 on the page 16.3-6) for major axis bending (bending about the principal Y axis).
Tee Parameter		
SFYBend	1.0	Parameter to specify safety factor for the computation of the limit state of Y axis (minor axis) bending of the tee section.
Double Angle Parameters		
nConnect	0	Number of connectors between individual angles. The user specified value is used during the code check. When the SELECT MEMBER (design) is requested, the user specified value is used unless more connectors are required. If the

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Double Angle Parameters (continued)

Parameter Name	Default Value	Meaning
		designed number of connectors are larger than the user specified value, the computed number of connectors are used and printed after the SELECT MEMBER result. The default value of zero indicates that the angles are connected at the ends only. Following are additional options that you can specify for this parameter: 0 = angles are connected at the ends of the member. –1 = requesting the number of connectors to be computed during code check. –2 = bypass modified column slenderness equations. This will bypass the check for the Section E4.1 of the AISC LRFD Third Edition.
ConnType	WELDED	Type of the intermediate connectors that are used for double angle. Choices are: SNUG and WELDED. SNUG = intermediate connectors that are snug-tight bolted. WELDED = intermediate connectors that are welded or fully tensioned bolted. This is the default.
L	Computed	Actual member length is used to design a number of connectors and to check connector spacing (Section E4.2 of the AISC LRFD), and also used in the computation of the modified column slenderness, $(KL/r)_m$ (Section E4.1 of the AISC LRFD). This parameter is used to compute the distance between connectors $a = L(n+1)$, where ‘a’ is the distance between connectors, ‘L’ is the physical member length, and ‘n’ is the number of connectors. The default is computed as the length of the member.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Double Angle Parameters (continued)		
K	1.0	Effective length factor for an individual component (single angle). This parameter is used to design a number of connectors and to check the connector spacing (Section E4.2 of the AISC LRFD).
SFYBend	1.0	Parameter to specify safety factor for the computation of the limit state of Y axis (minor axis) bending of the double angle section.
Round HSS (Pipes) Shear Check Parameters		
avy	Computed	The length of essentially constant shear in the Y axis direction of a member. This parameter is used to check the Y direction shear of a round HSS (pipe) cross-section (96). This parameter is similar to the variable ‘a’ in the Equation 5.2-2 of the AISC LRFD HSS specification in the Section 16.2 of the LRFD Third Edition. The default is computed as the length of the member.
avz	Computed	The length of essentially constant shear in the Z axis direction of a member. This parameter is used to check the Z direction shear of a round HSS (pipe) cross-section (96). This parameter is similar to the variable ‘a’ in the Equation 5.2-2 of the AISC LRFD HSS specification in the Section 16.2 of the LRFD Third Edition. The default is computed as the length of the member.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	Default Value	Meaning
Round HSS (Pipes) Torsion Check Parameter		
LX	Computed	This parameter is to specify the distance between torsional restraints. LX is used in the equation 6.1-2 on Page 16.2-8 of AISC LRFD Third Edition (96). This parameter is similar to the variable ‘a’ in the Equation 5.2-2 of the AISC LRFD HSS specification in the Section 16.2 of the LRFD Third Edition. The default is computed as the length of the member.
Rectangular Hollow Structural Section (HSS) Parameters		
Cby	Computed	Coefficient used in computing limiting compressive bending strength (AISC LRFD Third Edition, Section F1.2a, Equation F1-3) for minor axis bending (bending about the Y-axis).
UNLCW	Computed	Unbraced length of the compression web about the local Y axis of the profile. The default is taken as length of member.
FRUNLCW	Computed	Fractional form of the parameter UNLCW allows unbraced length to be specified as a fraction of the total length. Used only when UNLCW is computed.
Plate Girder Parameters		
Fyst	Fy	Minimum yield stress of the transverse stiffeners material. If not specified, it is assumed equal to the parameter Fy.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Plate Girder Parameters (continued)

Parameter Name	**Default Value**	**Meaning**
Ast	0.0	Parameter to specify the transverse stiffeners area. This parameter is used to check Appendix G4 of AISC LRFD 3rd Edition. The specified transverse stiffeners area is checked to see if it is smaller than the computed value from Equation A-G4-1 of Appendix G4 of AISC LRFD 3rd Edition. The default value of 0.0 indicates that the transverse stiffeners area of Appendix G4 is not checked.
Ist	0.0	Parameter to specify the transverse stiffeners moment of inertia. This parameter is used to check Appendix F2.3 of AISC LRFD 3rd Edition for the required transverse stiffeners moment of inertia. The default value of 0.0 indicates that the transverse stiffeners moment of inertia according to Appendix F2.3 is not checked.
Dstiff	2.4	Parameter to specify the factor D that is used in the Equation A-G4-1 of Appendix G4 of AISC LRFD 3rd Edition. A default value of 2.4 for single plate stiffeners is assumed. The value of factor D (parameter 'Dstiff') in the Equation A-G4-1 is dependent on the type of transverse stiffeners used in a plate girder. Alternate values are as follows: 1.0 = for stiffeners in pairs. This is the default value when the specified value for the parameter 'NumBars' is greater than 1. 1.8 = for single angle stiffeners. 2.4 = for single plate stiffeners. This is the default value when the specified value for the parameter 'NumBars' is equal to 1.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Plate Girder Parameters (continued)		
NumBars	1.0	Parameter to specify a number of single plate stiffeners. The default value for this parameter indicates 1 single plate stiffener.
Stiff-H	0.0	Parameter to specify the single plate stiffeners cross-section's height. Parameters 'Stiff-H', 'Stiff-W', and 'NumBars' are used for the automatic computation of the parameters 'Ast' and 'Ist'. The automatic computation of the parameters 'Ast' and 'Ist' is based on the rectangular bar stiffeners geometry. If transverse stiffeners are not rectangular bar, parameters 'Ast' and 'Ist' should be specified.
Stiff-W	0.0	Parameter to specify the single plate stiffeners cross-section's width. See parameter 'Stiff-H' for more information.
Force Limitation		
FXMIN	0.5 (lb)	Minimum axial force to be considered by the code; anything less in magnitude is taken as zero.
FYMIN	0.5 (lb)	Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero.
FZMIN	0.5 (lb)	Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Force Limitation (continued)		
MXMIN	20.0 (in-lb)	Minimum torsional moment to be considered by the code; anything less in magnitude is taken as zero.
MYMIN	20.0 (in-lb)	Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero.
MZMIN	20.0 (in-lb)	Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero.
Output Processing and System Parameters		
SUMMARY	NO	Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; see Sections 2.9 and 7.2 of Volume 2A of the User Reference Manual for an explanation.
PrintLim	NO	Parameter to request to print the section limiting values for limit state and load and resistance factor codes. The default output from CHECK or SELECT command prints the section force values. A value of ‘YES’ for this parameter indicates that the section limiting values should be printed instead of default section forces.

Table LRFD3.1-1 (continued)

LRFD3 Code Parameters

Parameter Name	Default Value	Meaning
Output Processing and System Parameters (continued)		
TRACE	4.0	Flag indicating when checks of code provisions should be output during design or code checking. See Section 7.2 of Volume 2A of the User Reference Manual for an explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces.
VALUES	1.0	Flag indicating if parameter or property values are to be output when retrieved. See Section 7.2 of Volume 2A of the User Reference Manual for the explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties.

This page intentionally left blank.

Table LRFD3.1-2

GTSTRUDL AISC Codes*

Code Name	Parameter Table	Application
LRFD3	LRFD3.1-1 Volume 2 - LRFD3	Checks compliance of I shapes, channels, single angles, tees, double angles, round HSS (pipes), rectangular and square HSS (structural tubes), solid round, square and rectangular bars, and plate girder profiles to the 1999 AISC LRFD, Third Edition, Specification (96).
ASD9-E	ASD9-E.1-1 Volume 2 - ASD9-E	Checks compliance of I shape profiles to the 1989 AISC ASD Ninth Edition specification (72) with equations that have been modified to include the modulus of elasticity (constant E).
LRFD2	LRFD2 Volume 2A	Checks compliance of I shapes, pipes, structural tubing, plate girders (subjected to bi-axial bending and axial force), single and double angles (subjected to axial forces only) shape profiles to the 1993 AISC LRFD, Second Edition, Specification (81).
ASD9	ASD9 Volume 2A	Checks compliance of I shapes, single angles, channels, tees, double angles, solid round bars, pipes, solid squares and rectangular bars, and structural tubing shape profiles to the 1989 AISC ASD, Ninth Edition, Specification (72).
78AISC	2.2.3.1 Volume 2B	Checks compliance of I shapes, single angles, channels, tees, solid round bars, pipes, solid squares and rectangular bars, and structural tubing (use code name DBLANG for double angle) shape profiles to the 1978 AISC Specification (33), Eighth Edition, including 1980 updates.

---

* For latest (up to date) version of this table, see Table 2.1-1a of Volume 2A.

Table LRFD3.1-2 (continued)

GTSTRUDL AISC Codes*

Code Name	**Parameter Table**	**Application**
69AISC	2.2.3.1 Volume 2B	Checks compliance of I shapes, single angles, channels, tees, solid round bars, pipes, solid squares and rectangular bars, and structural tubing (use code name DBLANG for double angle) shape profiles to the 1969 AISC Specification (16), Seventh Edition, including supplements 1, 2, and 3.
W78AISC	2.2.3.1 Volume 2B	Similar to 78AISC code, except limited to checking I shape profiles. This code is identical to the 78AISC code which was available in older versions of GTSTRUDL (i.e., version V1M7 and older).
DBLANG	2.2.3.1 Volume 2B	Checks compliance of double angle profiles to the 1969 AISC Specification (16), Seventh Edition, including supplements 1, 2, and 3.
W69AISC	2.2.3.1 Volume 2B	Similar to 69AISC code, except limited to checking I shape profiles. This code is identical to the 69AISC code which was available in older versions of GTSTRUDL (i.e., version V1M7 and older).

---

* For latest (up to date) version of this table, see Table 2.1-1a of Volume 2A.

Table LRFD3.1-3

GTSTRUDL Profile Tables for the Design based on the LRFD3 Code

Profile Shapes	**Reference**
I shapes	See Appendix C of Volume 2A for list of applicable table names for I shapes, W, S, M, HP shapes, wide flange shapes, universal beam shapes, universal column shapes, etc.
Channels	See Appendix C of Volume 2A for a list of channel table names applicable to LRFD3 codes.
Single Angles	See Appendix C of Volume 2A for list of single angle table names applicable to LRFD3 code.
Tees	See Appendix C of Volume 2A for a list of tee table names applicable to LRFD3 codes.
Double Angles	See Appendix C of Volume 2A for list of double angle table names applicable to LRFD3 code.
Round HSS	See Appendix C of Volume 2A for list of round HSS (pipe, circular hollow section) table names applicable to LRFD3 code.
Rectangular HSS	See Appendix C of Volume 2A for list of rectangular and square HSS (structural tube, rectangular and square hollow section) table names applicable to LRFD3 code.
Solid Round Bars	See Appendix C of Volume 2A for a list of solid round bar table names applicable to LRFD3 codes.
Solid Square Bars	See Appendix C of Volume 2A for a list of solid square bar table names applicable to LRFD3 codes.
Solid Rectangular Bars	See Appendix C of Volume 2A for a list of solid rectangular bar table names applicable to LRFD3 codes.
Plate Girders	See Appendix C of Volume 2A for a list of plate girder table names applicable to LRFD3 codes.

Table LRFD3.1-4

ASTM Steel Grades and Associated Values of F_y and F_u Based on the 1999 AISC LRFD Third Edition Specifications
Applicable Shapes: W, M, S, HP, L, 2L, C, MC, WT, MT, and ST shapes from AISC Tables

Steel Grade ASTM Designation	Group Number Per ASTM A6 F_y, Minimum Yield Stress (ksi) F_u, Minimum Tensile Strength (ksi)				
	Group 1	Group 2	Group 3	Group 4	Group 5
A36	36 58	36 58	36 58	36 58	36 58
A529-G50	50 65	50 65	NA	NA	NA
A529-G55	55 70	55 70	NA	NA	NA
A572-G42	42 60	42 60	42 60	42 60	42 60
A572-G50	50 65	50 65	50 65	50 65	50 65
A572-G55	55 70	55 70	55 70	55 70	55 70
A572-G60	60 75	60 75	60 75	NA	NA
A572-G65	65 80	65 80	65 80	NA	NA
A913-G50	50 60	50 60	50 60	50 60	50 60
A913-G60	60 75	60 75	60 75	60 75	60 75
A913-G65	65 80	65 80	65 80	65 80	65 80
A913-G70	70 90	70 90	70 90	70 90	70 90

Table LRFD3.1-4 (continued)

ASTM Steel Grades and Associated Values of F_y and F_u Based on the 1999 AISC LRFD Third Edition Specifications
Applicable Shapes: W, M, S, HP, L, 2L, C, MC, WT, MT, and ST shapes from AISC Tables

Steel Grade ASTM Designation	Group Number Per ASTM A6 F_y, Minimum Yield Stress (ksi) F_u, Minimum Tensile Strength (ksi)				
	Group 1	Group 2	Group 3	Group 4	Group 5
A992[a]	50 65	50 65	50 65	50 65	50 65
A242	50 70	50 70	46[b] 67[b]	42[a] 63[a]	42[a] 63[a]
A588	50 70	50 70	50 70	50 70	50 70

a Applicable to W shapes only.

b Applicable to W and HP shapes only.

NA Indicates that shapes in the corresponding group are not produced for that grade of steel. GTSTRUDL assumes F_y and F_u to be zero in such cases and will not select profiles for these combinations of group number and steel grade. Minimum yield stresses (F_y) and minimum tensile strengths (F_u) were obtained from the summary of ASTM specifications included in the 1999 AISC LRFD Third Edition specification.

Table LRFD3.1-5

ASTM Steel Grades and Associated Values of F_y and F_u Based on the 1999 AISC LRFD Third Edition Specifications
Applicable Shapes: Round HSS, Steel Pipe, and Rectangular HSS

Steel Grade ASTM Designation	Applicable Shape Series F_y, Minimum Yield Stress (ksi) F_u, Minimum Tensile Strength (ksi)		
	Round HSS	Steel Pipe	Rectangular HSS
A53-GB	NA	35 60	NA
A500-GB	42 58	NA	46 58
A500-GC	46 62	NA	50 62
A501	36 58	NA	36 58
A618-GI A618-GII Thickness # 3/4	50 70	NA	50 70
A618-GI A618-GII Thickness > 3/4	46 67	NA	46 67
A618GIII	50 65	NA	50 65
A242-G46	NA	NA	46 67
A242-G50	NA	NA	50 70
A588	NA	NA	50 70
A847	50 70	NA	50 70

NA Not applicable. See Table LRFD3.1-4 for more explanation.

5.2.2 BS5950 Design Code and Parameters

00BS5950 Code
British Standard
BS 5950-1:2000

00BS5950.1.2 00BS5950 Code

The 00BS5950 code of GTSTRUDL may be used to select or check any of the following shapes:

I shapes	Subjected to bending and axial force
Single Angles	Subjected to axial force only
Circular Hollow Sections (Pipes)	Subjected to bending and axial force

The term I shapes is used to mean rolled or welded I and H beams and columns, universal beams and columns, joists, universal bearing piles, W, S, M, and HP profiles with doubly symmetric cross-sections.

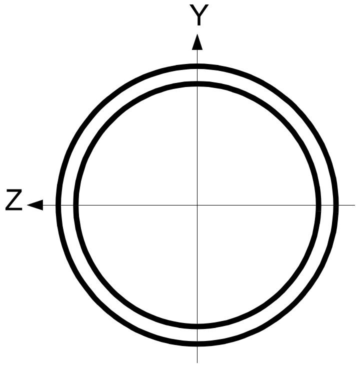
The code is based on the BS 5950-1:2000 "British Standard Structural use of steelwork in building, Part 1: Code of practice for design rolled and welded sections" amendment number 13199, issued May 2001. The 00BS5950 code utilizes the limit state design techniques of the BSI (British Standard Institution) specification.

The following assumptions are made throughout the 00BS5950 code.

1. Torsional stresses are usually small when compared to axial and bending stresses, and may be neglected. No checks are made for torsion. The designer is reminded to check the torsional stresses whenever they become significant.
2. Web stiffeners are considered for web shear stress, but they are not designed.

(a) I shapes

(b) Single angles



(c) Circular Hollow Sections
(Pipes)

Figure 00BS5950.1-1 Local Axes for Design with 00BS5950

The sections of the BS 5950-1:2000 specifications (95) that are considered by the GTSTRUDL 00BS5950 code are summarized below:

Section	Title
3.	**Properties of materials and section properties**
3.5	**Classification of cross-sections**
3.5.1	General
3.5.2	Classification
3.5.3	Flanges of compound I- or H-sections
	Table 11. Limiting width-to-thickness ratios for sections other than CHS and RHS
3.5.5	Stress ratios for classification
3.5.6.2	I- or H-sections with equal flanges
3.5.6.4	Circular hollow sections
3.6.2.2	Effective area
3.6.2.3	Effective modulus when web is fully effective
3.6.4	Equal-leg angle sections
3.6.5	Alternative method
3.6.6	Circular hollow sections
4.	**Design of structural members**
4.2.3	Shear capacity
4.2.5	**Moment capacity**
4.2.5.2	Low shear
4.2.5.3	High shear
4.3	**Lateral-torsional buckling**
4.3.4	Destabilizing load.
4.3.5	Effective length for lateral-torsional buckling
	Table 13. Effective length L_E for beams without intermediate restraint
4.3.6.2	I-, H-, channel and box sections with equal flanges
4.3.6.4	Buckling resistance moment
4.3.6.5	Bending strength p_b
4.3.6.6	Equivalent uniform moment factor m_{LT}
	Table 18. Equivalent uniform moment factor m_{LT} for lateral-torsional buckling

Section	Title
4.3.6.9	Ratio $\$_{w}$
4.4.5	**Shear buckling resistance**
4.4.5.2	Simplified method
4.4.5.3	More exact method
4.6	**Tension members**
4.6.1	Tension capacity
4.7.2	Slenderness
4.7	**Compression members**
4.7.2	Slenderness
4.7.4	Compression resistance
4.7.5	Compressive strength
	Table 23. Allocation of strut curve
4.8	**Members with combined moment and axial force**
4.8.2	**Tension members with moments**
4.8.2.2	Simplified method
4.8.2.3	More exact method
4.8.3	**Compression members with moments**
4.8.3.2	Cross-section capacity
4.8.3.3	**Member buckling resistance**
4.8.3.3.1	Simplified method
	Table 26. Equivalent uniform moment factor *m* for flexural buckling
4.8.3.3.2	More exact method for I- or H-sections with equal flanges
	Table 26. Equivalent uniform moment factor *m* for flexural buckling
4.8.3.3.3	More exact method for CHS, RHS or box sections with equal flanges
	Table 26. Equivalent uniform moment factor *m* for flexural buckling
4.9	Members with biaxial moments

Section	Title
Annex B	**(normative)** **Lateral-torsional buckling of members subject to bending**
B.2	**Buckling resistance**
B.2.1	Bending strength.
B.2.2	Perry factor and Robertson constant
B.2.3	Uniform I, H and channel sections with equal flanges
Annex C	**(normative)** **Compressive strength**
C.1	Strut formula
C.2	Perry factor and Robertson constant
Annex H	**(normative)** **Web buckling resistance**
H.1	Shear buckling strength
Annex I	**(normative)** **Combined axial compression and bending**
I.2	Reduced plastic moment capacity
I.2.1	I- or H-section with equal flanges

Tensile or compressive axial stresses, bi-axial bending, shear stresses, and combined stresses are considered for all cross-sections except single angles (tension or compression axial stresses only). Provisions for columns in simple construction are included. Parameters allowing for the changes which occur in structural steel at high temperatures have been included and may be invoked at the user's discretion.

The detailed explanation of the code parameters, cross-section properties, general nomenclature, and code equations are as follows.

1.	Table 00BS5950.1-1	Shows the parameters used by 00BS5950 code. Table 00BS5950.1-1 contains the applicable parameter names, their default values, and a brief description of the parameters.
2.	Section 00BS5950.2	Describes the cross-section properties used for each shape.

3.	Section 00BS5950.3	Contains detail discussion of the parameters used by the 00BS5950 code and they are presented in the alphabetic order in this section.
4.	Sections 00BS5950.4	Describes the subsections in the Section 00BS5950.4.
5.	Section 00BS5950.4.1	Defines the symbols used in the 00BS5950 code provisions.
6.	Section 00BS5950.4.2	Contains detailed discussion of the code provisions and the equations applicable to the I shape cross-sections subjected to bending and axial forces.
7.	Section 00BS5950.4.3	Contains detailed discussion of the code provisions and the equations applicable to the single angle cross-sections subjected to axial force only.
8.	Section 00BS5950.4.4	Contains detailed discussion of the code provisions and the equations applicable to the circular hollow sections (CHS, pipes) subjected to bending and axial forces.

Table 00BS5950.1-1

00BS5950 Code Parameters

Parameter Name	Default Value	Meaning
CODE	Required	Identifies the code to be used for member checking or member selection. Specify 00BS5950 for code name. See Sections 00BS5950.2, 00BS5950.3, and 00BS5950.4 for a more detailed description.
TBLNAM	UNIBEAMS	Identifies the table of profiles to be used during selection (SELECT command). See Table 00BS5950.1-2 for choices.
METHOD	EXACT	Identifies the design method. This parameter indicates the type of method that should be used for the shear or combined capacity checks. BOTH = Use simplified and the more exact methods. See Sections 4.4.5, 4.8.2 and 4.8.3 of BS 5950-1:2000 (95). EXACT = Use the more exact method. See Sections 4.4.5.3, 4.8.2.3, 4.8.3.3.2, and 4.8.3.3.3 of BS 5950-1:2000 (95). SIMPLIFY = Use simplified method. See Sections 4.4.5.2, 4.8.2.2 and 4.8.3.2 of BS 5950-1:2000 (95).
SECTYPE	ROLLED	Indicates that the cross-section is rolled or welded shape. This parameter is used to determine the equations that are applicable to the rolled or welded shape. ROLLED = Member is hot rolled. WELDED = Member is welded/coldformed.

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	**Default Value**	**Meaning**
SHRAREAF	Computed	SHeaR AREA Factor is used for the computation of the shear area. When an alternate value other than COMPUTE or TABLE is specified, shear area is computed as the SHRAREAF times the cross sectional area (A_V = AY = SHRAREAF × AX). COMPUTE = Compute the shear area based on the Section 4.2.3 of BS 5950-1:2000 (95) except for single and double angles. Shear area for single and double angles are extracted from GTSTRUDL or USER table. TABLE = Shear area from GTSTRUDL or USER table is used.
a	254000.0(mm)	Distance between web stiffeners. This parameter is used to compute a/d ratio. a/d is the ratio of the distance between stiffeners to web depth. An arbitrary high value of 254000.0 (mm) has been assumed as a default to indicate that the web stiffeners are absent. A value is necessary to account for web stiffeners in the shear capacity calculation (Provisions '4.4.5.2' and '4.4.5.3').
SimpSupp	NO	Indicates that if a member is simply supported or not. This parameter is used to determine the equations that are applicable to the simply supported members (Provisions ‘4.2.5.2Z’, ‘4.2.5.3Z’, ‘4.2.5.2Y’, and ‘4.2.5.3Y’. NO = Member is not simply supported. YES = Member is simply supported.
CODETOL	0.0	Percent variance from 1.0 for compliance with the provisions of a code. The ratio of actual/limiting must be less than or equal to [1.0 + CODETOL/100].

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	**Default Value**	**Meaning**
PF	1.0	Area reduction factor for holesout in members subject to axial tension.
Material Properties		
STEELGRD	S235JRG2	Identifies the grade of steel from which a member is made. See Table 00BS5950.1-3 for STEEL GRaDes and their properties.
Py	Computed	Design strength p_y (yield stress) of member. Computed from parameter STEELGRD if not given.
REDPy	1.0	Reduction factor for parameter Py. This factor times parameter Py gives the design strength (p_y) value used by the code. Used to account for property changes at high temperatures.
Pyf	Py	Design strength of the flange. If not specified, it is assumed equal to the parameter Py. This parameter is used to define a hybrid cross-section, see parameter Pyw also.
Pyw	Py	Design strength of the web. If not specified, it is assumed equal to the parameter Py. This parameter is used to define a hybrid cross-section, see parameter Pyf also.
REDE	1.0	Reduction factor for E, the modulus of elasticity. Similar to REDPy.

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Slenderness Ratio		
SLENCOMP	Computed	Maximum permissible slenderness ratio (L_E/r, KL/r) for a member subjected to axial compression. The default value for maximum compression slenderness ratio is equal to 180.
SLENTEN	Computed	Maximum permissible slenderness ratio (L/r) for a member subjected to axial tension. Only a user-specified value will initiate the slenderness ratio check for a tension member.
Effective Length for a Compression Member		
EFLEY	1.0	Effective factor value used for the computation of nominal effective length, L_{Ey} = EFLEY × LY for a compression member. Nominal effective length, L_{EY}, is used in the computation of maximum slenderness ratio about the local Y axis of the profile. See Table 00BS5950.1-4 or Sections 4.7.2, 4.7.3, and Table 22 of BS 5950-1:2000 (95) for the EFLEY values.
LY	Computed	Unbraced length for buckling about the local Y axis of the cross-section. This parameter is used to compute nominal effective length L_{Ey} for a compression member (L_{Ey} = EFLEY × LY). The default value is computed as a length of the member.
FRLY	1.0	Fractional form of the parameter LY, allows unbraced length to be specified as fractions of the total length. Used only when default value of ‘Computed’ is used for parameter LY (LY = FRLY × Member Length).

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Effective Length for a Compression Member (continued)		
EFLEZ	1.0	Effective factor value used for the computation of nominal effective length, L_{Ez} = EFLEZ × LZ for a compression member. Nominal effective length, L_{EZ}, is used in the computation of maximum slenderness ratio about the local Z axis of the profile. See Table 00BS5950.1-4 or Sections 4.7.2, 4.7.3, and Table 22 of BS 5950-1:2000 (95) for the EFLEZ values.
LZ	Computed	Unbraced length for buckling about the local Z axis of the cross-section. This parameter is used to compute nominal effective length L_{Ez} for a compression member (L_{Ez} = EFLEZ × LZ). The default value is computed as a length of the member.
FRLZ	1.0	Fractional form of the parameter LZ, allows unbraced length to be specified as fractions of the total length. Used only when default value of 'Computed' is used for parameter LZ (LZ = FRLZ × Member Length).
Effective Length for Lateral-Torsional Buckling		
LE	LLT	Effective length of a member for lateral torsional buckling of a beam with restraints at the ends. Default value is the effective length between restraints against lateral-torsional buckling of a member under bending, see parameter LLT (LE = EFLE × LLT). See Table 00BS5950.1-5 for alternative values and also see Table 13 and 14 of the BS5950-1:2000 (95).

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	Default Value	Meaning
Effective Length for Lateral-Torsional Buckling (continued)		
EFLE	1.0	Effective factor value used for the computation of the effective length, LE of a member under bending. Used only when default value of LLT is used for parameter LE (LE = EFLE × LLT, see Table 00BS5950.1-5 and parameter LE).
LLT	Computed	Segment length between restraints against lateral-torsional buckling (unbraced length). This parameter generally used to specify the segment length of the compression flange restraint against lateral-torsional buckling (unbraced length of the compression flange). Computed as length of member.
FRLLT	1.0	Fractional value used for the computation of the unbraced lateral-torsional buckling length of a member, LLT. Used only when default value of 'Computed' is used for parameter LLT (LLT = FRLLT × Member Length).
Equivalent Uniform Moment Factors		
mLT	Computed	Equivalent uniform moment factor for lateral-torsional buckling (m_{LT}), which is used in the member buckling resistance equations. This parameter modifies Z axis bending buckling capacity in combined axial and bending capacity equations. See Section 00BS5950.3 for more explanation.
my	Computed	Equivalent uniform moment factor for flexural buckling (m_y), which is used in the member buckling resistance equations. This parameter modifies Y axis bending capacity in combined axial and bending capacity equations. See Section 00BS5950.3 for more explanation.

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Equivalent Uniform Moment Factors (continued)		
mz	Computed	Equivalent uniform moment factor for flexural buckling (m_z), which is used in the member buckling resistance equations. This parameter modifies Z axis bending capacity in combined axial and bending capacity equations. See Section 00BS5950.3 for more explanation.
myz	Computed	Equivalent uniform moment factor for lateral flexural buckling (m_{yz}), which is used in the member out-of-plane buckling resistance equations. This parameter modifies Y axis bending capacity in combined axial and bending capacity equations. See Section 00BS5950.3 for more explanation.
SDSWAYY	YES	Indicates the presence or absence of SiDeSWAY about the local Y axis. YES = Sidesway permitted. NO = Sidesway prevented.
SDSWAYZ	YES	Indicates the presence or absence of SiDeSWAY about the local Z axis. YES = Sidesway permitted. NO = Sidesway prevented.

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	Default Value	Meaning
Equivalent Uniform Moment Factors (continued)		
DESTLDY	YES	Indicates the presence or absence of a DESTabilizing LoaD which causes movement in the member local Y axis direction (and possibly rotation about the member local Y axis). Destabilizing load conditions exist when a load is applied in the local Z axis direction of a member and both the load and the member are free to deflect laterally (and possibly rotationally also) relative to the centroid of the member. This parameter is only applicable to LOADS list or ALL LOADS of the PARAMETERS command. YES = Destabilizing load. NO = Normal load.
DESTLDZ	YES	Indicates the presence or absence of a DESTabilizing LoaD which causes movement in the member local Z axis direction (and possibly rotation about the member local Z axis). Destabilizing load conditions exist when a load is applied to the top flange (local Y axis load) of a member and both the load and the flange are free to deflect laterally (and possibly rotationally also) relative to the centroid of the member. This parameter is only applicable to LOADS list or ALL LOADS of the PARAMETERS command. YES = Destabilizing load. NO = Normal load.
Force Limitation		
FXMIN	2.224 (N)	Minimum axial force to be considered by the code; anything less in magnitude is taken as zero. Units are in newtons (N).
FYMIN	2.224 (N)	Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero.

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Force Limitation (continued)		
FZMIN	2.224 (N)	Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero.
MYMIN	2260.0 (mm-N)	Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero.
MZMIN	2260.0 (mm-N)	Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero.
Output Processing		
MXTRIALS	500.0	Maximum number of profiles to be tried when designing a member. Default is larger than the number of profiles in most tables.
SUMMARY	NO	Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; see Sections 2.9 and 7.2 of Volume 2A of the User Reference Manual for an explanation
PrintLim	NO	Parameter to request to print the section limiting values for limit state and load and resistance factor codes. This parameter is applicable to the steel design CHECK and SELECT commands. The default output from CHECK or SELECT command prints the section force values. A value of 'YES' for this parameter indicates that the section limiting values should be printed instead of default section forces.

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Output Processing (continued)		
TRACE	4.0	Flag indication when checks of code provisions should be output during design or code checking. See Section 7.2 of Volume 2A of the User Reference Manual for the explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces.
VALUES	1.0	Flag indication if parameter or property values are to be output when retrieved. See Section 7.2 of Volume 2A of the User Reference Manual for the explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties.

Table 00BS5950.1-2

GTSTRUDL Profile Tables for the Design based on the 00BS5950 Code

Profile Shapes	**Reference**
I shapes	See Appendix C of Volume 2A for list of Applicable Table names for universal beams, universal columns, joists, universal bearing piles, I shapes, W, S, M, HP shapes, wide flange shapes, etc.
Single Angles	See Appendix C of Volume 2A for list of single angle table names applicable to 00BS5950 code.
Circular Hollow Sections	See Appendix C of Volume 2A for list of circular hollow section (pipe, round HSS) table names applicable to 00BS5950 code.

Table 00BS5950.1-3

Steel Grades Based on the BS 5950-1:2000 (00BS5950) and 1993 Eurocode (EC3) Specification

Steel Grade	Nominal Yield Strength, f_y (N/mm^2)								Ultimate Tensile Strength, f_u		
	t # 16	16< t #40	40< t #63	63< t #80	80< t #100	100< t #150	150< t #200	200< t #250	t # 100	100< t #150	150< t #250
S185	185	175	!	!	!	!	!	!	290	!	!
S235JR	235	225	!	!	!	!	!	!	340	!	!
S235JRG1	235	225	!	!	!	!	!	!	340	!	!
S235JRG2	235	225	215	215	215	195	185	175	340	340	320
S235J0	235	225	215	215	215	195	185	175	340	340	320
S235J2G3	235	225	215	215	215	195	185	175	340	340	320
S235J2G4	235	225	215	215	215	195	185	175	340	340	320
S275JR	275	265	255	245	235	225	215	205	410	400	380
S275J0	275	265	255	245	235	225	215	205	410	400	380
S275J2G3	275	265	255	245	235	225	215	205	410	400	380
S275J2G4	275	265	255	245	235	225	215	205	410	400	380
S275N	275	265	255	245	235	225	!	!	370	350	!
S275NL	275	265	255	245	235	225	!	!	370	350	!

Table 00BS5950.1-3 (continued)

Steel Grades Based on the BS 5950-1:2000 (00BS5950) and 1993 Eurocode (EC3) Specification

Steel Grade	Nominal Yield Strength, f_y (N/mm^2)								Ultimate Tensile Strength, f_u		
	t # 16	16< t #40	40< t #63	63< t #80	80< t #100	100< t #150	150< t #200	200< t #250	t # 100	100< t #150	150< t #250
S355JR	355	345	335	325	315	295	285	275	490	470	450
S355J0	355	345	335	325	315	295	285	275	490	470	450
S355J2G3	355	345	335	325	315	295	285	275	490	470	450
S355J2G4	355	345	335	325	315	295	285	275	490	470	450
S355K2G3	355	345	335	325	315	295	285	275	490	470	450
S355K2G4	355	345	335	325	315	295	285	275	490	470	450
S355N	355	345	335	325	315	295	!	!	470	450	!
S355NL	355	345	335	325	315	295	!	!	470	450	!
S420N	420	400	390	370	360	340	!	!	520	500	!
S420NL	420	400	390	370	360	340	!	!	520	500	!
S460N	460	440	430	410	400	!	!	!	550	!	!
S460NL	460	440	430	410	400	!	!	!	550	!	!

Table 00BS5950.1-4

Effective Factor Values EFLEY and EFLEZ for Nominal Effective Length L_{Ey} and L_{Ez} computation British Standard BS 5950-1:2000 Specification

a) non-sway mode[*]

Restraint (in the plane under consideration) by other parts of structure		EFLEY and EFLEZ
Effectively held in position at both ends	Effectively restrained in direction at both ends	0.7
	Partially restrained in direction at both ends	0.85
	Restrained in direction at one end	0.85
	Not restrained in direction at either end	1.0

b) sway mode[*]

One end	Other end		EFLEY and EFLEZ
Effectively held in position and restrained in direction	Not held in position	Effectively restrained in direction	1.2
		Partially restrained in direction	1.5
		Not restrained in direction	2.0

[*] Excluding angle, channel or T-section struts designed in accordance with Section 4.7.10 of the BS 5950-1:2000 (95)

Example:

PARAMETERS
'EFLEY' 1.5 MEMBER 1 $ L_{Ey} = 1.5LY for member 1
'EFLEZ' 1.2 MEMBER 25 $ L_{Ez} = 1.2LZ for member 25

LY and LZ are the unbraced length for buckling about the local Y and Z axis of the cross-section (see parameter LY and LZ).

Table 00BS5950.1-5

Effective Length L_E
British Standard BS 5950-1:2000 Specification

Conditions of restraint at supports	Alternate values for Parameter LE	Loading conditions	
		Normal DESTLDZ = NO	Destabilizing DESTLDZ = YES
Default value for parameter LE	LLT	EFLLT×LLT	EFLLT×LLT
Compression flange laterally restrained. Nominal torsional restraint against rotation about longitudinal axis.			
Both flanges fully restrained against rotation on plan	A1	0.7LLT	0.85LLT
Compression flange fully restrained against rotation on plan	A2	0.75LLT	0.9LLT
Both flanges partially restrained against rotation on plan	A3	0.8LLT	0.95LLT
Compression flange partially restrained against rotation on plan	A4	0.85LLT	1.0LLT
Both flanges free to rotate on plan	A5	1.0LLT	1.2LLT
Compression flange laterally unrestrained. Both flanges free to rotate on plan			
Partial torsional restraint against rotation about longitudinal axis provided by connection of bottom flange to supports	A6	1.0LLT + 2D	1.2LLT + 2D
Partial torsional restraint against rotation about longitudinal axis provided only by pressure of bottom flange onto supports	A7	1.2LLT + 2D	1.4LLT + 2D

Example:

```
PARAMETERS
   'DESTLDZ'      'NO'     LOAD 2
   'DESTLDZ'      'YES'    LOAD 5
   'LE'           'A3'     MEMBER 1     $  LE   =   0.8LLT  for load 2 and
                                        $  LE   =   0.95LLT  for load 5
   'LE'           'A7'     MEMBER 8     $  LE   =   1.2LLT+2D  for load 2 and
                                        $  LE   =   1.4LLT+2D  for load 5
```

---

1. D is the depth of cross-section (table property YD).
2. Default value for parameter EFLLT is equal to 1.0.
3. For cantilevers and other types of beams not in Table 00BS5950.1-6, use parameter EFLLT to specify the effective length factor (LE = EFLLT×LLT).

This page intentionally left blank.

5.2.3 GTSTRUDL Indian Standard Design Code IS800

A new steel design code named IS800 has been added.

IS800 Code
Indian Standard
IS:800-1984

IS800.1.2 IS800 Code

The IS800 code of GTSTRUDL may be used to select or check any of the following shapes:

I shapes
Channels
Single angles
Tees
Double angles
Round Bars
Pipes
Square Bars
Rectangular Bars
Structural Tubing

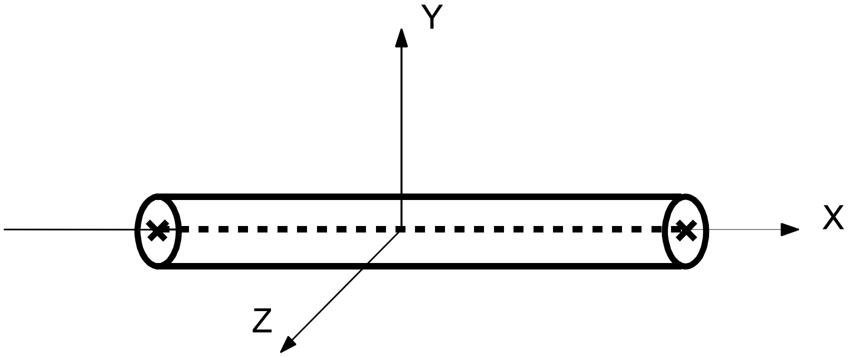
The term I shapes is used to mean ROLLED I beams and columns, universal beams and columns, W, S, M, and HP profiles with doubly symmetric cross-sections.

The code is based on the Indian Standard "Code of Practice for General Construction in Steel (Second Revision)" adopted April 25, 1984 (Twelfth Reprint December 1995, incorporating Amendments No. 1 and 2), IS:800-1984. The IS800 code utilizes the allowable stress design techniques of the Indian Standard IS:800-1984 code.

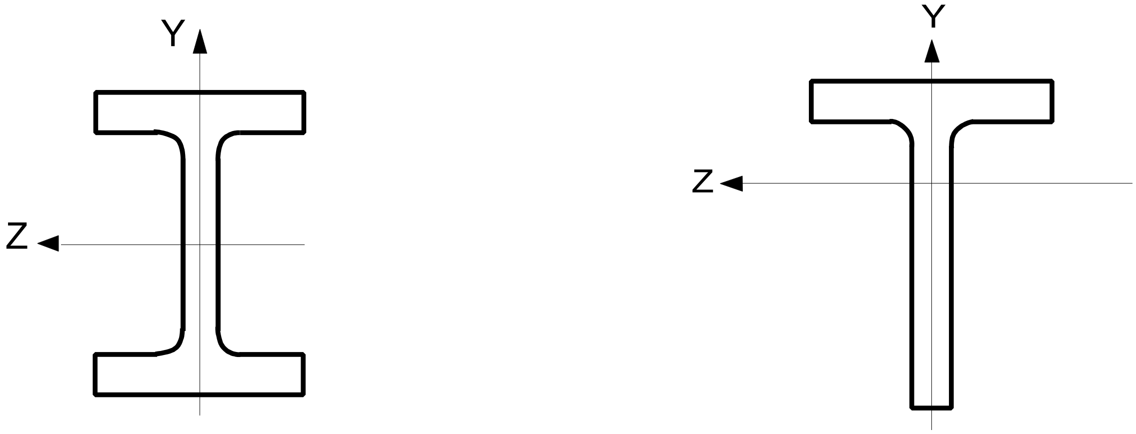
Design criteria for the above shapes are presented in Section IS800.4. A detailed discussion is presented on the allowable stresses for each of these shapes in Sections IS800.5.2 through IS800.5.10.

The following assumptions are made throughout the IS800 code.

1. The member under consideration is made of one grade of steel.
2. The modulus of elasticity of the steel is 200,000 MPa. This is of particular importance, since the computation of several constants appearing in the equations of the IS:800-1984 Specification (92) is based on this value.
3. Torsional stresses are usually small when compared to axial and bending stresses, and may be neglected. No checks are made for torsion. The designer is reminded to check the torsional stresses whenever they become significant.

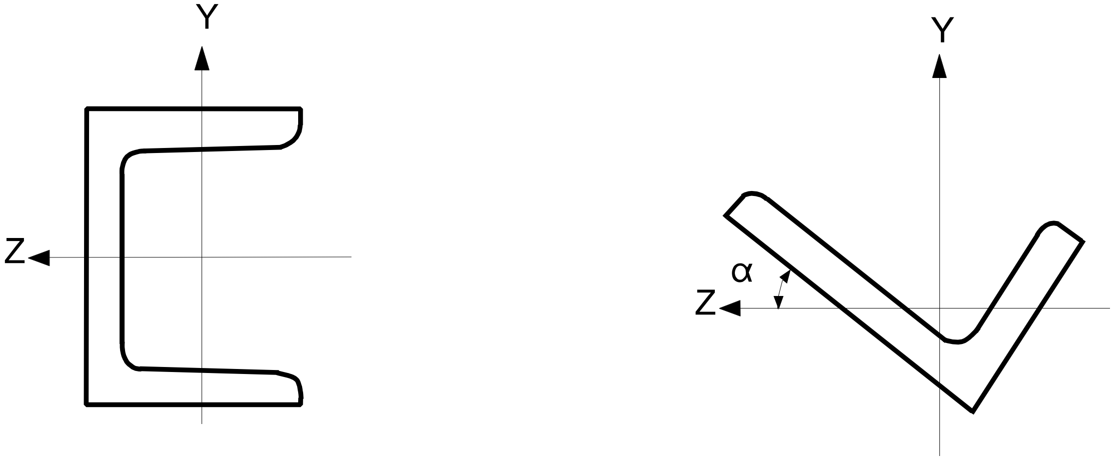


(a) LOCAL PRINCIPLE AXES ORIENTATION
(IN FIGURES BELOW X is OUT of PAPER)



(b) W SHAPES and I BEAMS

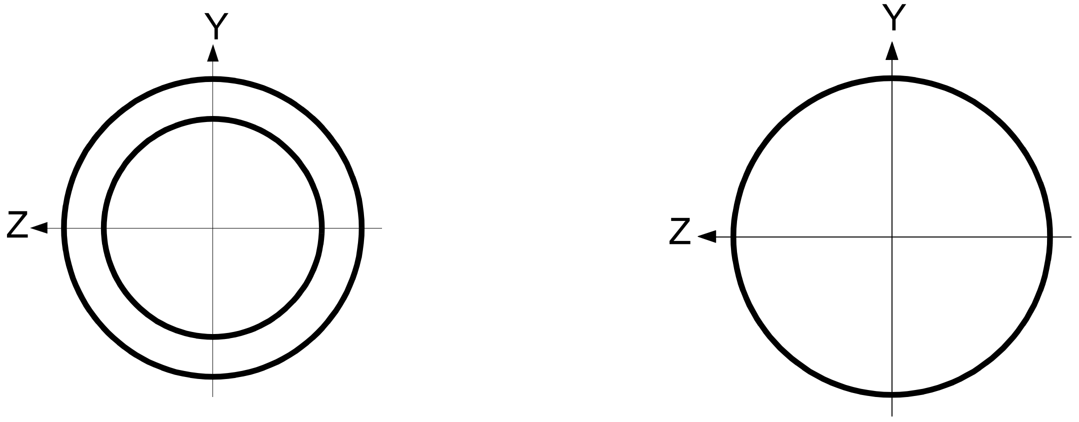
(c) TEES



(d) CHANNELS

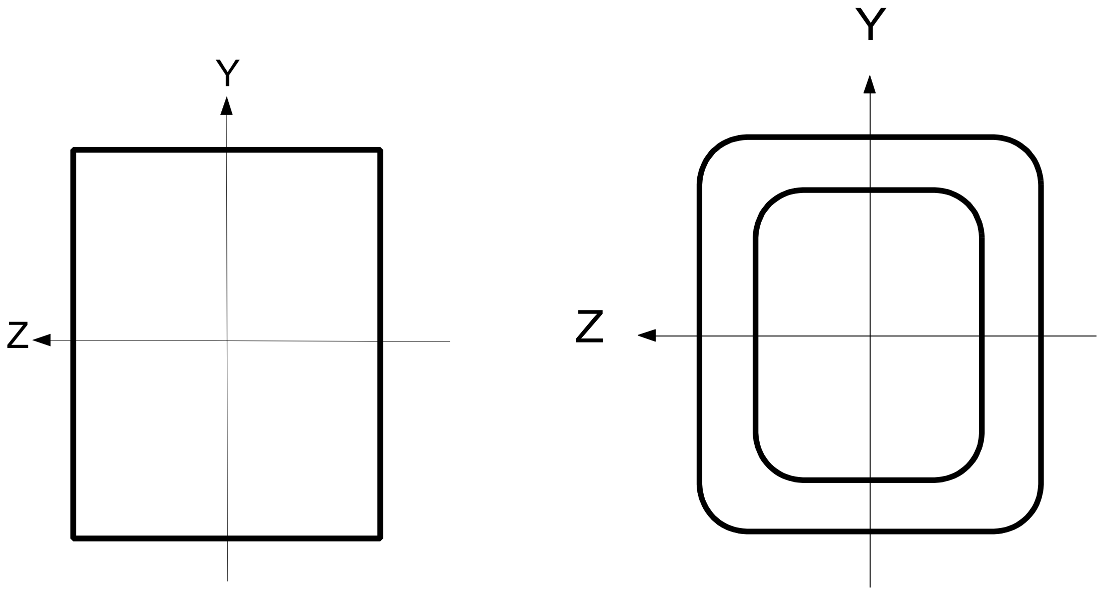
(e) SINGLE ANGLES

Figure IS800.1-1 Assumed Local Axes Direction for Hot Rolled Shapes



(f) PIPES

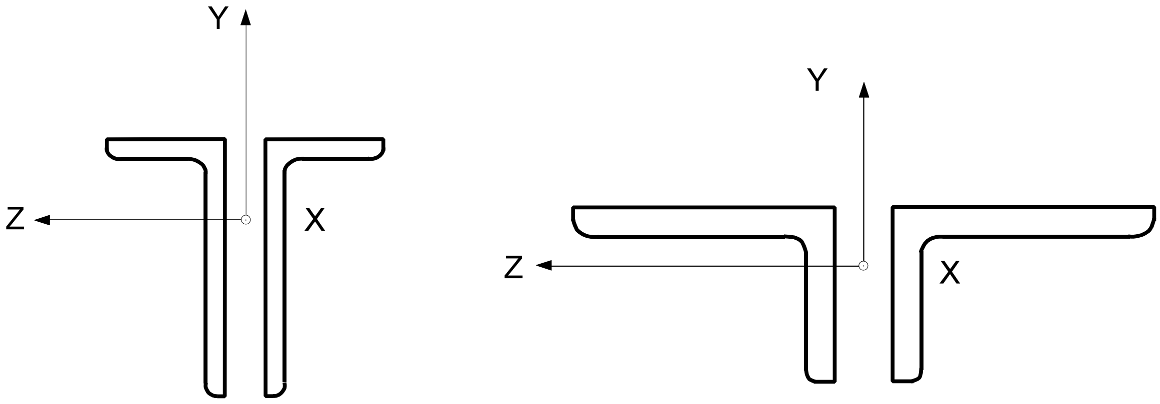
(g) ROUND SOLID BARS



(h) SQUARE and RECTANGULAR SOLID BARS

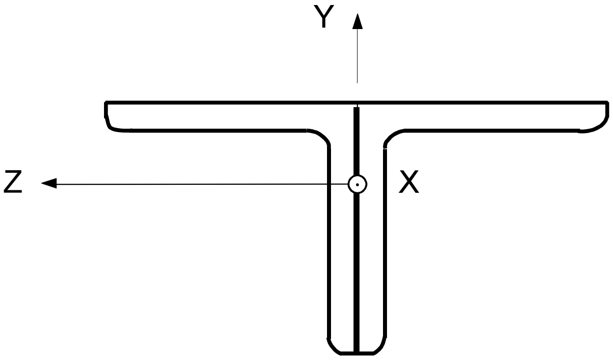
(i) TUBES

Figure IS800.1-1 (continued) Assumed Local Axes Direction for Hot Rolled Shapes



(j) Long legs back-to-back
double angle with spacing

(k) Short legs back-to-back
double angle with spacing



(l) Equal legs back-to-back
double angle in contact

Figure IS800.1-1 (continued) Assumed Local Axes Direction for Hot Rolled Shapes

4. Double angle contains an adequate number of intermediate connectors (stitch plates) which make the two angles act as one, Tee-like section.
5. The IS800 code assumes all shapes are hot rolled. In the case of a welded plate shape, the user must be certain that the section properties contained in a user created table of welded plate shapes are consistent with the requirements of the IS:800-1984 Specification (92). For example, in the case of a welded plate I-shape section, the shear area, AY, used for both analysis and shear stress checks must be equal to the web thickness times the interior distance between flanges (i.e., WBTK × INTYD).
6. In the case of welded plates, if the welded plates are not stress relieved, a value of ‘NO’ should be specified for the parameter ‘STRERELI’. For more explanation, see parameter ‘STRERELI’ and Section 3.5.2.2 of IS:800-1984.

Tensile or compressive axial stresses, bi-axial bending, shear stresses, and combined stresses are considered by IS800 code. Parameters allowing for the changes which occur in structural steel at high temperatures have been included and may be invoked at the user's discretion.

The sections of the IS:800-1984 Specifications (92) which are considered by the GTSTRUDL IS800 code are summarized below:

Section	Title
3.5	**Geometrical Properties**
3.5.2.1	Plate Thickness
3.5.2.2	Plate thickness
3.7	**Maximum Slenderness Ratio**
3.7.1	The maximum slenderness ratio
4.1	**Axial Stresses**
4.1.1	The permissible stress in axial tension
5.1.1	The permissible stress in axial compression
6	**Design of Members Subjected to Bending**
6.2.1	Maximum Bending Stresses
6.2.3	Maximum Permissible Bending Compressive Stress in Beams
6.2.4	Elastic Critical Stress
6.2.5	Beams Bent About the Axis of Minimum Strength
6.2.6	Angles and Tees

6.4 **Shear Stresses**
6.4.1 Maximum Shear Stress
6.4.2 Average Shear Stress
6.2.4 Elastic Critical Stress
6.2.5 Beams Bent About the Axis of Minimum Strength

7.1 **Combined of Direct Stress**es
7.1.1 Combined Axial Compression and Bending
7.1.2 Combined Axial Tension and Bending
7.1.3 Symbols
7.1.4 Bending and Shear

The detailed explanation of the code parameters, cross-section properties, general nomenclature, and code equations are as follows.

1. Table IS800.1-1 Shows the parameters used by the IS800 code. Table IS800.1-1 contains the applicable parameter names, their default values, and a brief description of the parameters.
2. Section IS800.2 Describes the cross-section properties used for each shape.
3. Section IS800.3 Contains detail discussion of the parameters used by the IS800 code and they are presented in alphabetic order in this section.
4. Section IS800.4 Describes the subsections in the Section IS800.4.
5. Section IS800.4.1 Defines the symbols used in the IS800 code provisions.
6. Section IS800.4.2 Contains detailed discussion of the code provisions and the equations applicable to the I shape cross-sections subjected to bending and axial forces.
7. Section IS800.4.3 Contains detailed discussion of the code provisions and the equations applicable to the Channel cross-sections subjected to bending and axial forces.
8. Section IS800.4.4 Contains detailed discussion of the code provisions and the equations applicable to the Single Angle cross-sections subjected to bending and axial forces

9.	Section IS800.4.5	Contains detailed discussion of the code provisions and the equations applicable to the Tee cross-sections subjected to bending and axial forces.
10.	Section IS800.4.6	Contains detailed discussion of the code provisions and the equations applicable to the Double Angle cross-sections subjected to bending and axial forces.
11.	Section IS800.4.7	Contains detailed discussion of the code provisions and the equations applicable to the Round Bar cross-sections subjected to bending and axial forces.
12.	Section IS800.4.8	Contains detailed discussion of the code provisions and the equations applicable to the Pipe cross-sections subjected to bending and axial forces.
13.	Section IS800.4.9	Contains detailed discussion of the code provisions and the equations applicable to the Square and Rectangular Bar cross-sections subjected to bending and axial forces.
14.	Section IS800.4.10	Contains detailed discussion of the code provisions and the equations applicable to the Structural Tubing cross-sections subjected to bending and axial forces.

This page intentionally left blank.

Table IS800.1-1

IS800 Code Parameters

Parameter Name	Default Value	Meaning
CODE	Required	Identifies the code to be used for member checking or member selection. Specify IS800 for code name. See Table IS800.1-2 and Sections IS800.2, IS800.3, and IS800.4 for a more detailed description.
TBLNAM	ISBEAMS	Identifies the table of profiles to be used during selection (SELECT command). See Table IS800.1-3 for choices.
CODETOL	0.0	Percent variance from 1.0 for compliance with the provisions of a code. The ratio of Actual/Allowable must be less than or equal to [1.0 + CODETOL/100].
PF	1.0	Area reduction factor for holesout in members subject to axial tension.
a	254000.0 (mm)	Distance between web stiffeners. This parameter is used to compute a/h ratio. The a/h ratio is the ratio of the distance between stiffeners to the web depth. An arbitrary high value of 254000.0 (mm) has been assumed as a default to indicate that web stiffeners are absent. A value is necessary to account for web stiffeners in the allowable shear stress calculation (Provision ‘6.4.2 Y’ and ‘6.4.2 Z’).
STRERELI	YES	Parameter to specify if the welded plates are stress relieved or not. This parameter is used for the computationof the effective clear depth of the web (see Section 3.5.2.2 of IS:800-1984 and Section IS800.4.2 of Volume 2 - IS800). A value of NO indicates that when the effective clear depth of the web is being computed, assume that the welded plates are not stress relieved. The default value of ‘YES’ indicates that the cross-section is stress relieved.

Table IS800.1-1 (continued)

IS800 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Material Properties		
STEELGRD	A36	Identifies the grade of steel from which a member is made. See Table IS800.1-4 for steel grades and their properties.
FY	Computed	Yield stress of member. Computed from STEELGRD if not given.
REDFY	1.0	Reduction factor for FY. This factor times FY gives the f_y value used by the code. Used to account for property changes at high temperatures.
REDE	1.0	Reduction factor for E, the modulus of elasticity. Similar to REDFY.
Slenderness Ratio		
SLENCOMP	Computed	Maximum permissible slenderness ratio (KL/r) for member subjected to axial compression. When no value is specified for this parameter, the value of 180 is used for the maximum slenderness ratio.
SLENTEN	Computed	Maximum permissible slenderness ratio (L/r) for member subjected to axial tension. When no value is specified for this parameter, the value of 400 is used for the maximum slenderness ratio.
K-Factors		
COMPK	NO	Parameter to request the computation of the effective length factors KY and KZ (Section 2.2 of Volume 2A). YES = Compute KY and KZ factors. KY = Compute KY only. KZ = Compute KZ only. NO = Use default or specified values for KY and KZ.

Table IS800.1-1 (continued)

IS800 Code Parameters

Parameter Name	**Default Value**	**Meaning**
K-Factors (continued)		
KY	1.0	Effective length factor for buckling about the local Y axis of the profile. See Section 2.2 of Volume 2A for GTSTRUDL computation of effective length factor, KY.
KZ	1.0	Effective length factor for buckling about the local Z axis of the profile. See Section 2.2 of Volume 2A for GTSTRUDL computation of effective length factor, KZ.
Print-K	YES	Parameter to print the computed K-factor values after the default code check or select command output (TRACE 4 output). The default value of 'YES' for this parameter indicates that the computed K-factor values should be printed after the code check or select command output. The column names attached to the start and end of the code checked member is also printed. This printed information allows the user to inspect the automatic detection of the columns attached to the start and end of the designed member. A value of 'NO' indicates that K-factor values and the names of the attached columns to the start and end of the designed member should not be printed.
SDSWAYY	YES	Indicates the presence or absence of sidesway about the local Y axis. YES = sidesway permitted. NO = sidesway prevented.
SDSWAYZ	YES	Indicates the presence or absence of sidesway about the local Z axis. YES = sidesway permitted. NO = sidesway prevented.

Table IS800.1-1 (continued)

IS800 Code Parameters

Parameter Name	Default Value	Meaning
CantiMem	NO	Parameter to indicate that a member or a physical member which is part of a cantilever truss should be considered as a cantilever in the K-factor computation. True cantilever members or physical members are detected automatically. NO = member of physical member is not cantilever, YES = member of physical member is cantilever.
GAY	Computed	G-factor at the start joint of the member. GAY is used in the calculation of effective length factor KY (see parameter COMPK, KY, and Section 2.2 of Volume 2A).
GAZ	Computed	G-factor at the start joint of the member. GAZ is used in the calculation of effective length factor KZ (see parameter COMPK, KZ, and Section 2.2 of Volume 2A).
GBY	Computed	G-factor at the end joint of the member. GBY is used in the calculation of effective length factor KY (see parameter COMPK, KY, and Section 2.2 of Volume 2A).

Table IS800.1-1 (continued)

Parameter Name	Default Value	Meaning
GBZ	Computed	G-factor at the end joint of the member. GBZ is used in the calculation of effective length factor KZ (see parameter COMPK, KZ, and Section 2.2 of Volume 2A).
Buckling Length		
LY	Computed	Unbraced length for buckling about the local Y axis of the profile. Computed as length of member.
LZ	Computed	Unbraced length for buckling about the local Z axis of the profile. Computed as length of member.

Table IS800.1-1 (continued)

IS800 Code Parameters

Parameter Name	Default Value	Meaning
Buckling Length (continued)		
FRLY	1.0	Fractional form of the parameter LY. Allows the unbraced length to be specified as fractions of the total length. Used only when LY is computed.
FRLZ	1.0	Fractional form of the parameter LZ, similar to FRLY. Used only when LZ is computed.
Bending Stress		
UNLCF	Computed	Unbraced length of the compression flange. Computed as length of member. In this parameter no distinction is made between the unbraced length for the top or bottom flange. See UNLCFTF or UNLCFBF.
FRUNLCF	1.0	Fractional form of the parameter UNLCF. Allows the unbraced length to be specified as a fraction of the total length. Used only when UNLCF is computed.
UNLCFTF	Computed	Unbraced length of the compression flange for the top flange. When no value is specified, UNLCF and FRUNLCF is used for this parameter.
UNLCFBF	Computed	Unbraced length of the compression flange for the bottom flange. When no value is specified, UNLCF and FRUNLCF is used for this parameter.

Table IS800.1-1 (continued)

IS800 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Combined Stresses		
AXEFF	0.0	Axial stress reduction factor indicating the amount of the axial stress which is to be deducted from a corresponding bending stress acting in the opposite direction (see Provisions ‘AXC TBEN’ and ‘AXT CBEN’ for Channels, Section IS800.4.3).
CMY	Computed	Coefficient which modifies Y axis bending stress in interaction equation (IS:800-1984 Second Ed., Section 7 (92)).
CMZ	Computed	Coefficient which modifies Z axis bending stress in interaction equation (IS:800-1984 Second Ed., Section 7 (92)).
Force Limitation		
FXMIN	2.2 (N)	Minimum axial force to be considered by the code; anything less in magnitude is taken as zero.
FYMIN	2.2 (N)	Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero.
FZMIN	2.2 (N)	Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero.
MYMIN	2260.0 (mm-N)	Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero.
MZMIN	2260.0 (mm-N)	Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero.

Table IS800.1-1 (continued)

IS800 Code Parameters

Parameter Name	Default Value	Meaning
Output Processing and System Parameters		
MXTRIALS	500.0	Maximum number of profiles to be tried when designing a member. Default is larger than the number of profiles in most tables.
PRIDTA	1.0	Flag for requesting output from selection procedure. 1 = no output 2 = output parameters
SUMMARY	NO	Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; See Sections 2.9 and 7.2 of Volume 2A for explanation.
PrintStr	NO	Parameter to request to print the section actual and allowable values for allowable stress design codes. The default output from CHECK or SELECT command prints the section force values. A value of ‘YES’ for this parameter indicates that the section actual and allowable values should be printed instead of default section forces.
TRACE	4.0	Flag indication when checks of code provisions should be output during design or code checking. See Section 7.2 of Volume 2A for explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces.
VALUES	1.0	Flag indication if parameter or property values are to be output when retrieved. See Section 7.2 of Volume 2A for explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties.

Table IS800.1-2

GTSTRUDL Indian Standard Code(s)

Code Name	Parameter Table	Application
IS800	IS800.2-1	Checks compliance of I shape, Single angle, Channel, Tee, Double angles, Solid Round bar, Pipe, Solid Square and Rectangular bar, and Structural tubing shape profiles to the Indian Standard IS:800-1984 Specification (92).

Table IS800.1-3

GTSTRUDL Profile Tables for the Design based on the IS800 Codes

Profile Shapes	**Reference**
I shapes	See Appendix C of Volume 2A for list of applicable table names for I shapes, W, S, M, HP shapes, wide flange shapes, universal beam shapes, universal column shapes, etc.
Channels	See Appendix C of Volume 2A for list of channel cross-section table names applicable to IS800 code.
Single Angles	See Appendix C of Volume 2A for list of single angle cross-section table names applicable to IS800 code.
Tees	See Appendix C of Volume 2A for list of tee cross-section table names applicable to IS800 code.
Double Angles	See Appendix C of Volume 2A for list of double angle cross-section table names applicable to IS800 code.
Solid Round Bars	See Appendix C of Volume 2A list of solid round bar cross-section table names applicable to IS800 code.
Pipes	See Appendix C of Volume 2A for list of pipe (round HSS, circular hollow section) cross-section table names applicable to IS800 code.
Solid Square Bars	See Appendix C of Volume 2A for list of solid square bar cross-section table names applicable to IS800 code.
Solid Rectangular Bars	See Appendix C of Volume 2A for list of solid rectangular bar cross-section table names applicable to IS800 code.
Structural Tubes	See Appendix C of Volume 2A for list of rectangular and square structural tube (rectangular and square HSS, rectangular and square hollow section) cross-section table names applicable to IS800 code.

Table IS800.1-4

Permissible Steel Grade Based on 1993 AISC LRFD Second Edition, 1989 AISC ASD Ninth Edition, and 1978 AISC Specification

Assumed Value of Yield Stress, F_y, and Minimum Tensile Strength, F_{ts}.

Steel Grade ASTM Designation	Group Number Per ASTM A6 F_y, Minimum Yield Stress (ksi) F_u, F_{ts}, Tensile Stress (ksi)				
	1	2	3	4	5
A36	36 58	36 58	36 58	36 58	36 58
A529	42 60	NA	NA	NA	NA
A441	50 70	50 70	46 67	42 63	42 63
A572-G42	42 60	42 60	42 60	42 60	42 60
A572-G50	50 65	50 65	50 65	50 65	50 65
A572-G60	60 75	60 75	NA	NA	NA
A572-G65	65 80	NA	NA	NA	NA
A242	50 70	50 70	46 75	42 63	42 63
A588	50 70	50 70	50 70	50 70	50 70

NA indicates that shapes in the corresponding group are not produced for that grade of steel. GTSTRUDL assumes F_y and F_{ts} to be zero in such cases and will not select profiles for these combinations of group number and steel grade. Yield strengths (F_y) and minimum tensile strengths (F_{ts}) were obtained from the summary of ASTM specifications included in the 1993 AISC LRFD Second Edition, 1989 AISC ASD Ninth Edition, and the 1978 AISC specification.

5.2.4 ACI Code 318-99

Design of beams and columns by the 1999 ACI code has been added. Only members designated as TYPE BEAM or TYPE COLUMN in a DESIGN DATA command can be PROPORTIONed when the METHOD is set to ACI318-99. When you specify ACI318-99, you will be reminded that it is a pre-release feature by a message (see the Example below). Note that CHECK is not available for codes after ACI318-77, including ACI318-99.

$$\underline{\text{MET}}\text{HOD} \ (\left\{ \begin{array}{l} \rightarrow \ \underline{\text{ULT}}\text{IMATE} \ (\underline{\text{STR}}\text{ENGTH}) \\ \ \underline{\text{WOR}}\text{KING} \ (\underline{\text{STR}}\text{ESS}) \end{array} \right\}) \left\{ \begin{array}{l} \underline{\text{ACI318-99}} \\ \underline{\text{ACI318-89}} \\ \underline{\text{ACI318-83}} \\ \underline{\text{ACI318-77}} \\ \underline{\text{ACI318-63}} \\ \underline{\text{(BSI)}} \ \underline{\text{CP110-72}} \\ \underline{\text{(BSI)}} \ \underline{\text{BS8110}} \end{array} \right\} -$$

$$(\ \underline{\text{BARS}} \left\{ \begin{array}{l} \rightarrow \ \underline{\text{ASTM}} \\ \ \underline{\text{CAN}}\text{ADIAN} \ (\underline{\text{STA}}\text{NDARD}) \\ \ \underline{\text{UNE}}\text{SCO} \\ \ \underline{\text{KOR}}\text{EAN} \ (\underline{\text{S}}\text{TANDARD}) \end{array} \right\}) \ (\left\{ \begin{array}{l} \rightarrow \ \underline{\text{NONS}}\text{EISMIC} \\ \ \underline{\text{SEIS}}\text{MIC} \\ \ \underline{\text{MODE}}\text{RATE} \ \underline{\text{SEIS}}\text{MIC} \end{array} \right\})$$

Example:

```
METHOD ACI318-99
  ****INFO_MET – 318-99 is a pre-release feature.

    DESIGN DATA FOR MEMBER 1
      TYPE BEAM RECT
    PROPORTION MEMBER 1
                        ....
    ACTIVE CODE  =  ACI 318-99
                        ....
                        (the rest of the output is the same format as previous
                        codes)
```

The table of CONSTANTS and assumed values for ACI 318-99 is shown below:

TABLE 2.4-1. CONSTANTS and Assumed Values for ACI 318-99

CONSTANT	Explanation	ACI 318-99	Assumed Value
FCP	Compressive strength of concrete, f'_c		4000 psi
FY	Yield strength of reinforcement, f_y		60000 psi
WC	Unit weight of plain concrete		145 pcf
DENSITY	Unit weight of reinforced concrete (1)		150 pcf
FC	Allow compr. stress in concrete, F_c	A.3.1	0.45(FCP)
VU	Ult. shear stress in beam with web reinf. (2)	11.5.6.9	$(8\sqrt{FCP}+v_c)$ (5)
V	Allow. shear stress in beam with web reinf.	A.3.1(b)	$(5.5\sqrt{FCP})$
RFSP	Splitting ratio, $f_{ct}/(\sqrt{f'_c})$ (3)	9.5.2.3	6.7
FYST	Yield strength of stirrups		60000 psi
FYSP	Yield strength of spiral		60000 psi
FS	Allowable tension stress in primary reinf.		20000 psi for
FSC	Allowable compressive stress in column reinf. (4)	A.3.2	Grades 40, 50
FV	Allowable tension stress in stirrups (5)		24000 psi for
			Grade 60
PHIFL	Flexure capacity reduction factor	9.3.2	0.9
PHISH	Shear capacity reduction factor	9.3.2	0.85
PHIBO	Bond capacity reduction factor	9.3.2	0.85
PHITO	Torsion capacity reduction factor	9.3.2	0.85
PHISP	Spiral column capacity reduction factor	9.3.2	0.75
PHITI	Tied column capacity reduction factor	9.3.2	0.7
BLFR	Ratio of max p, (p - p') or (p_w - p_f) to p_{bal}	10.3.3	0.75
PMAXCO	Maximum allowable reinforced ratio in columns	10.9.1	0.08
PMINCO	Minimum allowable reinforced ratio in columns	10.9.1	0.01
PMINFL	Minimum allowable reinforced ratio in flexural members	10.5.1	200/FY
ES	Modulus of elasticity for reinf. steel	8.5.2	29×10^6psi
EC	Modulus of elasticity for concrete	8.5.1	$33(WC)^{1.5}\sqrt{FCP}$
EU	Ult. strain in concrete at extreme comp. fiber	10.2.3	0.003

Notes:

1. The constant 'DENSITY' is the GTSTRUDL constant of the same name which has been set to a value of 150 pcf for reinforced concrete.

2. VU is multiplied by PHISH internally.

3. Calculations for V_c and T_c are modified by replacing $\sqrt{f_c'}$ with RFSP/6.7($\sqrt{f_c'}$) as per Section 11.2.1.1.

4. The assumed value of FSC is also limited to 30,000 psi maximum.

5. This value is defined only at the time of stirrup design.

This page intentionally left blank.

5.2.5 Rectangular and Circular Concrete Cross-Section Tables

New tables have been added for rectangular and circular concrete cross sections. The new table for rectangular sections is called CONRECT and the new table for circular sections is called CONCIR. These tables are added to facilitate the modeling and analysis of concrete cross sections but may not be used in the design of concrete cross sections. In order to design concrete sections, the MEMBER DIMENSION command must be used (see Section 2.5 of Volume 4 of the GTSTRUDL User Reference Manual).

The profiles in the CONCIR table are shown below where the name CIRxx indicates a circular cross section and xx is the diameter in inches. Thus, CIR12 is a 12 inch diameter circular cross section.

CIR12
CIR14
CIR16
CIR18
CIR20
CIR22
CIR24
CIR26
CIR28
CIR30
CIR32
CIR34
CIR36

The profiles in the CONRECT table are shown below where the name RECYYXZZ indicates a rectangular cross section with a width of YY inches and a depth of ZZ inches. Thus, REC16X24 is 16 inch wide and 24 inch deep rectangular cross section.

REC6X12	REC8X12	REC10X12	REC12X12	REC14X12	REC16X12
REC6X14	REC8X14	REC10X14	REC12X14	REC14X14	REC16X14
REC6X16	REC8X16	REC10X16	REC12X16	REC14X16	REC16X16
REC6X18	REC8X18	REC10X18	REC12X18	REC14X18	REC16X18
REC6X20	REC8X20	REC10X20	REC12X20	REC14X20	REC16X20
REC6X22	REC8X22	REC10X22	REC12X22	REC14X22	REC16X22
REC6X24	REC8X24	REC10X24	REC12X24	REC14X24	REC16X24
REC6X26	REC8X26	REC10X26	REC12X26	REC14X26	REC16X26
REC6X28	REC8X28	REC10X28	REC12X28	REC14X28	REC16X28
REC6X30	REC8X30	REC10X30	REC12X30	REC14X30	REC16X30
REC6X32	REC8X32	REC10X32	REC12X32	REC14X32	REC16X32
REC6X34	REC8X34	REC10X34	REC12X34	REC14X34	REC16X34
REC6X36	REC8X36	REC10X36	REC12X36	REC14X36	REC16X36

REC18X12
REC18X14
REC18X16
REC18X18
REC18X20
REC18X22
REC18X24
REC18X26
REC18X28
REC18X30
REC18X32
REC18X34
REC18X36
REC20X12
REC20X14
REC20X16
REC20X18
REC20X20
REC20X22
REC20X24
REC20X26
REC20X28
REC20X30
REC20X32
REC20X34
REC20X36
REC22X12
REC22X14
REC22X16
REC22X18
REC22X20
REC22X22
REC22X24
REC22X26
REC22X28
REC22X30
REC22X32
REC22X34
REC22X36
REC24X12
REC24X14
REC24X16
REC24X18
REC24X20
REC24X22
REC24X24
REC24X26
REC24X28
REC24X30
REC24X32
REC24X34
REC24X36
REC26X12
REC26X14
REC26X16
REC26X18
REC26X20
REC26X22
REC26X24
REC26X26
REC26X28
REC26X30
REC26X32
REC26X34
REC26X36
REC28X12
REC28X14
REC28X16
REC28X18
REC28X20
REC28X22
REC28X24
REC28X26
REC28X28
REC28X30
REC28X32
REC28X34
REC28X36

REC30X12
REC30X14
REC30X16
REC30X18
REC30X20
REC30X22
REC30X24
REC30X26
REC30X28
REC30X30
REC30X32
REC30X34
REC30X36
REC32X12
REC32X14
REC32X16
REC32X18
REC32X20
REC32X22
REC32X24
REC32X26
REC32X28
REC32X30
REC32X32
REC32X34
REC32X36
REC34X12
REC34X14
REC34X16
REC34X18
REC34X20
REC34X22
REC34X24
REC34X26
REC34X28
REC34X30
REC34X32
REC34X34
REC34X36
REC36X12
REC36X14
REC36X16
REC36X18
REC36X20
REC36X22
REC36X24
REC36X26
REC36X28
REC36X30
REC36X32
REC36X34
REC36X36

5.2.6 ASD9-E Code

A special Ninth Edition AISC allowable stress design code for W shapes has been implemented. The code name is ASD9-E. This code is based on the Ninth Edition AISC ASD except the equations have been modified to include modulus of elasticity (constant E). ASD9-E is applicable to W shapes only. This code is useful for structures where E and possibly other material data must be modified to account for high temperature. Parameters for the ASD9-E Code are shown below:

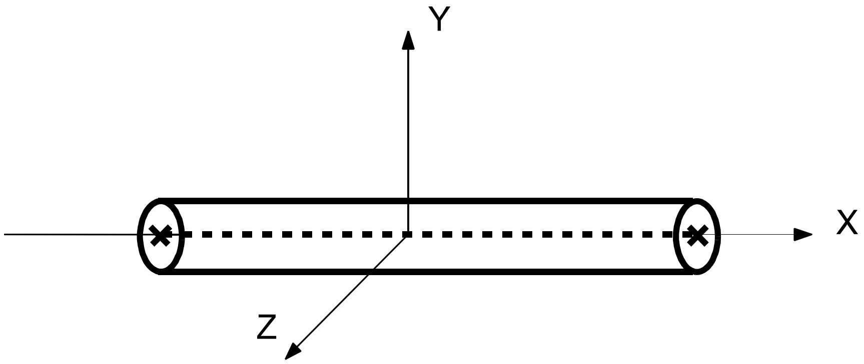
ASD9-E.1.2 ASD9-E Code

The ASD9-E Code of GTSTRUDL may be used to select or check I shape cross-section. The term I shapes is used to mean W, S, M, and HP profiles. This code is primarily based on the AISC "Specification for Structural Steel buildings, Allowable Stress Design and Plastic Design" adopted June 1, 1989. The Specification is contained in the Ninth Edition of the AISC Manual of Steel Construction, Allowable Stress Design (72). The ASD9-E code utilizes the allowable stress design techniques of the AISC Specification. The equations of the AISC have been modified to include constant E (modulus of elasticity). ASD9-E is similar to ASD9 code but it is only applicable to I shape cross-sections.

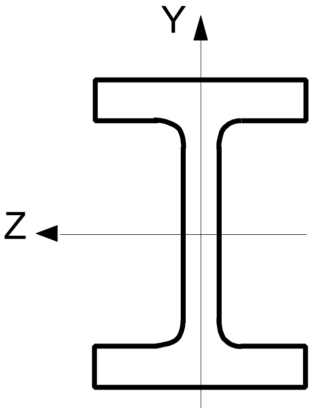
The purpose of this code is to be able to code check structures that experience high temperature or when ever the modulus of elasticity of the steel is different than standard value of 29,000 ksi (. 200 kN/mm^2). Equations in the AISC ASD specification have been simplified by assuming the modulus of elasticity of the steel is equal to 29,000 ksi (. 200 kN/mm^2). The equations in the GTSTRUDL ASD9-E code are the original equations that contains the modulus of elasticity.

Design criteria for the I shape cross-section are presented in Section ASD9-E.4. A detailed discussion is presented on the allowable stresses for I shape cross-section in Section ASD9-E.4.2. The following assumptions are made throughout the ASD9-E Code.

1. The member under consideration is made of one grade of steel.
2. Torsional stresses are usually small when compared to axial and bending stresses, and may be neglected. No checks are made for torsion. The designer is reminded to check the torsional stresses whenever they become significant.
3. Web stiffeners are considered for web shear stress, but they are not designed.



(a) LOCAL PRINCIPLE AXES ORIENTATION
(IN FIGURES BELOW X is OUT of PAPER)



(b) W SHAPES and I BEAMS

Figure ASD9-E.1-1 Assumed Local Axes Direction for Hot Rolled I Shape

Tensile or compressive axial stresses, bi-axial bending, shear stresses, and combined stresses are considered by ASD9-E. Provisions for slender compression elements, Appendix B of AISC Specification, are included when necessary. Parameters allowing for the changes which occur in structural steel at high temperatures have been included and may be invoked at the user's discretion.

The properties used for I shape cross-section is defined under Section ASD9-E.2. The parameters used by ASD9-E are discussed in detail in Table ASD9-E.1-1 and Section ASD9-E.3. Section ASD9-E.4.1 defines the general nomenclature used in describing the ASD9-E Code. The equations used in ASD9-E to determine the acceptability of a profile are described in detail for I shape cross-section in Section ASD9-E.4.2.

This page intentionally left blank.

Table ASD9-E.1-1

ASD9-E Code Parameters

Parameter Name	Default Value	Meaning
CODE	Required	Identifies the code to be used for member checking or member selection. Specify ASD9-E for code name.
TBLNAM	WSHAPES9	Identifies the table of profiles to be used during selection. See Table ASD9-E.1-2 for choices.
CODETOL	0.0	Percent variance from 1.0 for compliance with the provisions of a code. The ratio of Actual/Allowable must be less than or equal to [1.0 + CODETOL/100].
PF	1.0	Area reduction factor for holesout in members subject to axial tension.
A/H	10000.0	Ratio of clear span between transverse stiffeners to clear distance between flanges. Used in computing allowable shear stress. Default approximates infinity.
ALSTRINC	0.0	Allowable stress increase value. This parameter can be used to specify the 1/3 allowable stress increase for the wind or seismic loads. The user specified value for this parameter must be followed by the load list. An example for this parameter is to specify a value of 33.3333 followed by a load list.
Material Properties		
STEELGRD	A36	Identifies the grade of steel from which a member is made. See Table ASD9-E.1-3 for steel grades and their properties.
FYLD	Computed	Yield stress of member. Computed from STEELGRD if not given.
FTS	Computed	Minimum tensile strength of member. Computed from STEELGRD if not given.
REDFYLD	1.0	Reduction factor for FYLD. This factor times FYLD gives the FY value used by the code. Used to account for property changes at high temperatures.

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

Parameter Name	**Default Value**	**Meaning**
Material Properties (continued)		
REDFTS	1.0	Reduction factor for FTS. Similar to REDFYLD.
REDE	1.0	Reduction factor for E, the modulus of elasticity. Similar to REDFYLD.
Slenderness Ratio		
SLENCOMP	Computed	Maximum permissible slenderness ratio (KL/r) for member subjected to axial compression. When no value is specified for this parameter, the value of 200 is used for the maximum slenderness ratio.
SLENTEN	Computed	Maximum permissible slenderness ratio (L/r) for member subjected to axial tension. When no value is specified for this parameter, the value of 300 is used for the maximum slenderness ratio.
K-Factors		
COMPK	NO	Parameter to request the computation of the effective length factors KY and KZ (Sections 2.2 and 2.3 of Volume 2A). YES = Compute KY and KZ factors KY = Compute KY only. KZ = Compute KZ only. NO = Use default or specified values for KY and KZ.
KY	1.0	Effective length factor for buckling about the local Y axis of the profile. See Sections 2.2 and 2.3 of Volume 2A for GTSTRUDL computation of effective length factor, KY.
KZ	1.0	Effective length factor for buckling about the local Z axis of the profile. See Sections 2.2 and 2.3 of Volume 2A for GTSTRUDL computation of effective length factor, KZ.

Table ASD9-E.1-1 (continued)
ASD9-E Code Parameters

Parameter Name	**Default Value**	**Meaning**
K-Factors (continued)		
Print-K	YES	Parameter to print the computed K-factor values after the default code check or select command output (TRACE 4 output). The default value of 'YES' for this parameter indicates that the computed K-factor values should be printed after the code check or select command output. The column names attached to the start and end of the code checked member is also printed. This printed information allows the user to inspect the automatic detection of the columns attached to the start and end of the designed member. A value of 'NO' indicates that K-factor values and the names of the attached columns to the start and end of the designed member should not be printed.
SDSWAYY	YES	Indicates the presence or absence of sidesway about the local Y axis. YES = sidesway permitted. NO = sidesway prevented.
SDSWAYZ	YES	Indicates the presence or absence of sidesway about the local Z axis. YES = sidesway permitted. NO = sidesway prevented.
CantiMem	NO	Parameter to indicate that a member or a physical member which is part of a cantilever truss should be considered as a cantilever in the K-factor computation. True cantilever members or physical members are detected automatically. NO = member of physical member is not cantilever, YES = member of physical member is cantilever.
GAY	Computed	G-factor at the start joint of the member. GAY is used in the calculation of effective length factor KY (see parameter COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A).

Table ASD9-E.1-1 (continued)
ASD9-E Code Parameters

Parameter Name	Default Value	Meaning
K-Factors (continued)		
GAZ	Computed	G-factor at the start joint of the member. GAZ is used in the calculation of effective length factor KZ (see parameter COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A).
GBY	Computed	G-factor at the end joint of the member. GBY is used in the calculation of effective length factor KY (see parameter COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A).
GBZ	Computed	G-factor at the end joint of the member. GBZ is used in the calculation of effective length factor KZ (see parameter COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A).
Buckling Length		
LY	Computed	Unbraced length for buckling about the local Y axis of the profile. Computed as length of member.
LZ	Computed	Unbraced length for buckling about the local Z axis of the profile. Computed as length of member.
FRLY	1.0	Fractional form of the parameter LY, allows unbraced length to be specified as fractions of the total length. Used only when LY is computed.
FRLZ	1.0	Fractional form of the parameter LZ, similar to FRLY. Used only when LZ is computed.
Flexural-Torsional Buckling		
FLTORBUK	YES	Indicates the consideration of flexural-torsional buckling check. YES = check flexural-torsional buckling. NO = do not check flexural-torsional buckling.

Table ASD9-E.1-1 (continued)
ASD9-E Code Parameters

Parameter Name	Default Value	Meaning
Flexural-Torsional Buckling (continued)		
KX	1.0	Effective length factor for torsional buckling about the local X axis of the profile. This parameter is used in flexural-torsional buckling stress, F_e computations.
LX	Computed	Unbraced length for torsional buckling about the local X axis of the profile. Computed as length of member. This parameter is used in flexural-torsional buckling stress, F_e computations.
FRLX	1.0	Fractional form of the parameter LX, allows unbraced length to be specified as fractions of the total length. Used only when LX is computed.
Bending Stress		
CB	Computed	Coefficient used in computing allowable compressive bending stress (AISC ASD Ninth Ed. Section F1.3).
UNLCF	Computed	Unbraced length of the compression flange. Computed as length of member. In this parameter no distinction is made between the unbraced length for the top or bottom flange. See UNLCFTF or UNLCFBF.
FRUNLCF	1.0	Fractional form of the parameter UNLCF, allows unbraced length to be specified as fractions of the total length. Used only when UNLCF is computed.
UNLCFTF	Computed	Unbraced length of the compression flange for the top flange. When no value is specified, UNLCF and FRUNLCF is used for this parameter.
UNLCFBF	Computed	Unbraced length of the compression flange for the bottom flange. When no value is specified, UNLCF and FRUNLCF is used for this parameter.

Table ASD9-E.1-1 (continued)
ASD9-E Code Parameters

Parameter Name	**Default Value**	**Meaning**
Combined Stresses		
AXEFF	0.0	Axial stress reduction factor indicating the amount of the axial stress which is to be deducted from a corresponding bending stress acting in the opposite direction.
CMY	Computed	Coefficient which modifies Y axis bending stress in interaction equation (AISC ASD Ninth Ed. Section H1).
CMZ	Computed	Coefficient which modifies Z axis bending stress in interaction equation (AISC ASD Ninth Ed. Section H1).
Force Limitation		
FXMIN	0.5(lb)	Minimum axial force to be considered by the code; anything less in magnitude is taken as zero.
FYMIN	0.5(lb)	Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero.
FZMIN	0.5(lb)	Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero.
MYMIN	20.0(in-lb)	Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero.
MZMIN	20.0(in-lb)	Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero.
Output Processing and System Parameters		
MXTRIALS	500.0	Maximum number of profiles to be tried when designing a member. Default is larger than the number of profiles in most tables.
SUMMARY	NO	Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; See Sections 2.9 and 7.2 of Volume 2A for explanation.

Table ASD9-E.1-1 (continued)
ASD9-E Code Parameters

Parameter Name	**Default Value**	**Meaning**
Output Processing and System Parameters (continued)		
PrintStr	NO	Parameter to request to print the section stress values for allowable stress design codes. The default output from CHECK or SELECT command prints the section force values. A value of ‘YES’ for this parameter indicates that the section stress values should be printed instead of default section forces.
TRACE	4.0	Flag indication when checks of code provisions should be output during design or code checking. See Section 7.2 of Volume 2A for explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces.
VALUES	1.0	Flag indication if parameter or property values are to be output when retrieved. See Section 7.2 of Volume 2A for explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties.

This page intentionally left blank.

Table ASD9-E.1-2

GTSTRUDL I shape Profile Tables for the Design based on the ASD9-E Code*

(I shapes, Universal Beams, Universal Columns, Joists, Piles, etc.)

Table Name	Reference
AISC Tables	(American Institute of Steel Construction)
W-LRFD3	W shapes from 1999 AISC LRFD Third Edition (96)
M/S/HPL3	M, S, and HP shape profiles from 1999 AISC LRFD Third Edition (96)
WSHAPES9	W shapes from 1989 AISC ASD Ninth Edition (72)
M/S/HP9	M, S, and HP shapes from 1989 AISC ASD Ninth Edition (72)
WBEAM9	W shapes commonly used as beams from 1989 AISC ASD Ninth Edition (72)
WCOLUMN9	W shapes commonly used as columns from 1989 AISC ASD Ninth Edition (72)
WSHAPESM	W shape from AISC Metric “WSHAPES” table (83)
M/S/HPM	M, S, and HP shape profiles from AISC Metric “M SHAPES, S SHAPES, and HP SHAPES” table (83)
WBEAMM	W shape profiles commonly used as beams from AISC Metric “WSHAPES” table (83)
WCOLUMNM	W shape profiles commonly used as columns from AISC Metric “WSHAPES” table (83)
STEELW78	W shapes from 1978 AISC ASD Eighth Edition (33)
HP/S/M	HP, S, and M shapes from 1978 AISC ASD Eighth Edition (33)
W78BEAM	W shapes commonly used as beams from 1978 AISC ASD Eighth Edition (33)
W78COLUM	W shapes commonly used as columns from 1978 AISC ASD Eighth Edition (33)
STEELW	W shapes from 1969 AISC ASD Seventh Edition (16)
WCOLUMN	W shapes commonly used as columns from 1969 AISC ASD Seventh Edition (16)

* See design code for applicable cross-sections
* See Appendix C of Volume 2A for Table description and profile names
* Also see Appendix C of Volume 2A for additional Table names

Table ASD9-E.1-2 (continued)

GTSTRUDL I shape Profile Tables for the
Design based on the ASD9-E Code*
(I shapes, Universal Beams, Universal Columns, Joists, Piles, etc.)

Table Name	**Reference**
Brazilian Standard Tables, NBR 5884 2000	
CS	I shapes from Brazilian Standard, ABNT, NBR 5884:2000
CVS	I shapes from Brazilian Standard, ABNT, NBR 5884:2000
VS	I shapes from Brazilian Standard, ABNT, NBR 5884:2000
British Standard Tables, BS 5950	
UNIBEAMS	British Universal Beam profiles from 1996 BS 5950 Section Properties, 4th Edition (82)
UNICOL	British Universal Column profiles from 1996 BS 5950 Section Properties, 4th Edition (82)
JOISTS	British Joist profiles from 1996 BS 5950 Section Properties, 4th Edition (82)
UBPILES	I shape profiles from British “UNIVERSAL BEARING PILES” table (82)
European Tables	
HEA	H shaped (HE-A) profiles from Breite I-Träger, Reihe HE-A. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997"
HEB	H shaped (HE-B) profiles from Breite I-Träger, Reihe HE-B. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997"
HEM	H shaped (HE-M) profiles from Breite I-Träger, Reihe HE-M. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997"

* See design code for applicable cross-sections

* See Appendix C of Volume 2A for Table description and profile names

* Also see Appendix C of Volume 2A for additional Table names

Table ASD9-E.1-2 (continued)

GTSTRUDL I shape Profile Tables for the
Design based on the ASD9-E Code*
(I shapes, Universal Beams, Universal Columns, Joists, Piles, etc.)

Table Name	**Reference**
European Tables (Continued)	
IPE	I shaped (IPE) profiles from Mittelbreite I-Träger, IPE-Reihe. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997"
EUROPEAN	This table contains profiles from IPE, HEA, HEB, and HEM tables
Indian Standard Tables from IS808:1989	
ISBEAMS	I shape beam sections (medium flange beams, junior and light weight beams) from Tables 2.1 and 2.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision (93)
ISCOLUMN	I shape column/heavy weight beam sections (column and heavy weight beams) from Table 3.1 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision (93)

* See design code for applicable cross-sections
* See Appendix C of Volume 2A for Table description and profile names
* Also see Appendix C of Volume 2A for additional Table names

Table ASD9-E.1-3

Permissible Steel Grade Based on 1989 AISC ASD Ninth Edition Specification

Assumed Value of Yield Stress, F_y, and Minimum Tensile Strength, F_{ts}.

Steel Grade ASTM Designation	Group Number Per ASTM A6 F_y, Minimum Yield Stress (ksi) F_u, F_{ts}, Tensile Stress (ksi)				
	1	2	3	4	5
A36	36 58	36 58	36 58	36 58	36 58
A529	42 60	NA	NA	NA	NA
A441	50 70	50 70	46 67	42 63	42 63
A572-G42	42 60	42 60	42 60	42 60	42 60
A572-G50	50 65	50 65	50 65	50 65	50 65
A572-G60	60 75	60 75	NA	NA	NA
A572-G65	65 80	NA	NA	NA	NA
A242	50 70	50 70	46 67	42 63	42 63
A588	50 70	50 70	50 70	50 70	50 70

NA indicates that shapes in the corresponding group are not produced for that grade of steel. GTSTRUDL assumes F_y and F_{ts} to be zero in such cases and will not select profiles for these combinations of group number and steel grade. Yield strengths (F_y) and minimum tensile strengths (F_{ts}) were obtained from the summary of ASTM specifications included in the 1989 AISC ASD Ninth Edition specification.

5.2.7 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

The goal of the DESIGN SLAB command is to select reinforcing steel for concrete flat plate systems using finite elements as a tool for the determination of design moments.

Instead of dealing with results on an element-by-element basis, the user will be able to design the reinforcing steel for slab systems based on cuts. Here, the term *cut* refers to the cross-section of a strip at a particular location to be designed. A cut is defined by two nodes identifying the start and end of the cut, and by an element in the plane of the cut.

Once the definition of the cut has been determined, the resultant forces along the cut are computed using either moment resultants (otherwise known as the Wood and Armer method) or element force results (using the CALCULATE RESULTANT command, as described in Section 2.3.7.3 of Volume 3 of the Reference Manuals). The final design moment is determined by computing the resultant moment acting on the cut for each loading condition, and reducing these moments to a design envelope.

Once the design envelope is computed, the cross-section is designed according to ACI 318-05 either using default design parameter or with certain user specified design parameters such as the bar size or spacing.

An important distinction is to note that each cut is designed independently from all other cuts. That is, a cut specified in one region is independent with respect to a design in another region. As such, if the user wishes to use the same bar size over multiple adjacent cuts, this information must be specified for each cut.

The form of the command is as follows:

$$\text{\underline{DES}IGN \underline{SLAB} (\underline{REIN}FORCEMENT) (\underline{US}ING)} -$$

$$\left\{\begin{array}{l} \text{\underline{WOOD} (\underline{AND}) (\underline{ARM}ER)}\left\{\begin{array}{l}\rightarrow \text{\underline{AVE}RAGE} \\ \quad \text{\underline{MAX}IMUM}\end{array}\right\} \\ \text{\underline{CALC}ULATE (\underline{RES}ULTANT) (\underline{ELE}MENT) (\underline{FOR}CES)} \end{array}\right\} \text{ (\underline{AL}ONG)} -$$

$$\text{(\underline{CUT}}\left\{\begin{array}{c}\text{'a'} \\ \text{i}_1\end{array}\right\}\text{)} \left\{\begin{array}{l}\text{\underline{JOI}NTS} \\ \text{\underline{NOD}ES}\end{array}\right\} \text{ list}_1 \text{ \underline{ELE}MENT list}_2 \text{ (\underline{TABLE}} \left\{\begin{array}{l}\rightarrow \text{\underline{ASTM}} \\ \quad \text{\underline{UNESCO}}\end{array}\right\}\text{)} -$$

$$^{*}\left\{\begin{array}{l} \text{\underline{TOP} (\underline{FACE}) (\underline{BAR}S i}_2\text{) (\underline{SPA}CING v}_1\text{)} \\ \text{\underline{BOT}TOM (\underline{FACE}) (\underline{BAR}S i}_3\text{) (\underline{SPA}CING v}_2\text{)} \\ \text{\underline{BOTH} (\underline{FACE}S) (\underline{BAR}S i}_4\text{) (\underline{SPA}CING v}_3\text{)} \end{array}\right\} -$$

$$\left\{\begin{array}{l}\rightarrow \text{\underline{IN}NER (\underline{LAY}ER)} \\ \quad \text{\underline{OUT}ER (\underline{LAY}ER)}\end{array}\right\} \text{ (\underline{COV}ER v}_4\text{) (\underline{LIN}EAR (\underline{TOL}ERANCE) v}_5\text{)} -$$

$$\text{(\underline{TOR}SIONAL (\underline{MOM}ENT) (\underline{WARN}ING) v}_6\text{)}$$

where,

‘a’ or i_1 refer to an optional alphanumeric or integer cut name

list_1 = list containing ID’s of the start and end node of the cut

list_2 = list containing the ID of an element in the plane of the cut

i_2 = bar size to be used for bars on the top surface of the slab

i_3 = bar size to be used for bars on the bottom surface of the slab

i_4 = bar size to be used for both the top and bottom surfaces of the slab

v_1 = reinforcing bar spacing to be used on the top surface of the slab

v_2 = reinforcing bar spacing to be used on the bottom surface of the slab

v_3 = reinforcing bar spacing to be used on both surfaces of the slab

v_4 = optional user-specified cover distance for reinforcing bars

v_5 = linear tolerance used in element selection rules for moment computation

v_6 = optional ratio of torsion to bending moment allowed on the cross-section

TOP = element surface with +ZPLANAR coordinate

BOTTOM = element surface with -ZPLANAR coordinate

Explanation:

The DESIGN SLAB command allows the user to communicate all data necessary for the reinforcing steel design. This information is processed and a design is calculated based on the input. The command is designed to provide varying levels of control for the user so as to make the command as broadly applicable as possible.

The user must first define the cut. A cut is defined by a start and end node ID, and an element ID in the plane of the cut. The user has the option of giving each cut an alphanumeric name for organizational purposes. The purpose of the required element ID is to determine the appropriate plane to design in the event that multiple planes of finite elements intersect along the cut, as defined by the start and end node. An example where this might occur is the intersection of a slab with a shear wall. In this case, a misleading design could be generated if the slab was designed using the forces in the shear wall. The cut definition constitutes all information required to compute the resultant forces acting along the cut.

The total moment acting on a cut cross-section is computed using one of two methods. The use of moment resultants, also known as the Wood and Armer method, is implemented as the default method. In this method, the moment resultants MXX, MYY, and MXY are resolved on a per node basis along the cut, and either the average effect or the maximum effect on the cut is applied to the entire cross-section.

The other option for moment computation is based on the use of element forces. In this method, the total resultant moment acting on the cross-section is computed using the CALCULATE RESULTANT command, and the element force nodal moments are resolved for each node of each element adjacent to the cut.

Once the cut has been defined, the user may indicate parameters to be used to design the system. The user may constrain the bar size or spacing to a certain value, either for the top face, bottom face, or for both faces. In this case, the final design will utilize the information provided. If the bar size is constrained, the appropriate spacing of bars is determined. If the bar spacing is constrained, the appropriate bar size is determined. In the case that the user supplies a bar size and spacing for the cut, the application will simply check the strength of the cross-section against the computed design envelope according to ACI 318. If the user specifies no design constraints, the application assumes a bar size and designs the section to satisfy ACI 318. As such, the user maintains explicit control over the function of the application.

The user may also specify which layer of bars to be designed, using the modifier INNER or OUTER. These refer to the location of reinforcing bars on each surface. At most slab locations, reinforcement is placed in two perpendicular directions on both surfaces of the slab. Since each layer of reinforcement cannot occupy the same space, one layer must be placed on top of the other. OUTER refers to the layer closest to the surface, while INNER refers to the layer nearest the center of the slab.

All user-specified constraints, such as concrete compressive strength, yield strength, cover, and spacing are checked against ACI minimum/maximum values, as specified in ACI 318-02. The thickness of the cross-section is determined internally based on the modeled thickness of the user-specified element.

With respect to the interpretation of results, “top” always refers to the face of the slab on the +ZPLANAR side of the element, and “bottom” always refers to the face of the slab on the -ZPLANAR side of the element. “Positive bending” refers to bending that produces tension on the bottom face of the slab and compression on the top face, as defined previously. “Negative bending” produces tension on the top face and compression on the bottom face, as defined previously.

Requirements:

The MATERIAL REINFORCED CONCRETE command must be specified before the DESIGN SLAB. The MATERIAL REINFORCED CONCRETE command initializes the RC capabilities of GT STRUDL and sets the relevant material and design quantities to their default values for design. At this point, the user can issue the CONSTANTS command to modify any material properties to be used in the design. The default values are:

ECU = 0.003
ES = 29,000,000 psi
FCP = 4000 psi
FY = 60,000 psi
PHIFL = 0.9

The STIFFNESS command must be issued prior to the DESIGN SLAB command. The STIFFNESS command solves the global equilibrium equation and computes the quantities required for the determination of the bending moments that the DESIGN SLAB command uses.

Only elements known to appropriately model the behavior of slab systems are included in the computation of design forces. For a flat plate system, only plate bending and plate elements are used. Thus, if the user models the system using plane stress / plane strain elements, and then issues the DESIGN SLAB command, a warning message is output and the command is ignored.

Plate bending elements supported include the BPHT, BPR, BPHQ, CPT, and IPBQQ finite elements. General plate elements supported include the SBCT, SBCR, SBHQ, SBHQCSH, SBHT, SBHT6, and SBHQ6 finite elements.

Usage:

Studies have shown that the CALCULATE RESULTANT ELEMENT FORCE option of the DESIGN SLAB command is only applicable in regions where the cut orientation is generally orthogonal to the directions of principle bending. If the geometry of a region dictates that a cut be oriented non-orthogonally to the principal bending directions, a significant torsional effect may occur. In this case, the Wood and Armer method must be employed due to its ability to correctly compute the ultimate moment in a strong torsion field. In the DESIGN SLAB command, the user is warned if the element force implementation computes a resultant torsion greater than 10% of the resultant bending moment on a particular cross-section. The user may modify the torsion warning threshold via the modifiers TORSIONAL MOMENT WARNING. If there is any question of the orientation of the cut with respect to the directions of principal bending, the user should investigate the behavior in the finite element results section of GTMENU.

Usage Example: Description of Example Structure

The example structure is a rectangular slab system, shown in Figure 5.2.7-1. The clear span of the structure is thirty feet, and the slab strip has a width of ten feet. The two ends of the slab are fully fixed, while the thirty foot sides are free, resembling a fixed-fixed beam. The slab is one foot thick and constructed of normal strength concrete with FCP = 4000 psi. The example structure can be idealized as a subset of a larger slab system, perhaps the design strip running between two column faces in an interior region. The structure is loaded with a distributed surface pressure of 150 psf over the entire surface of the slab.

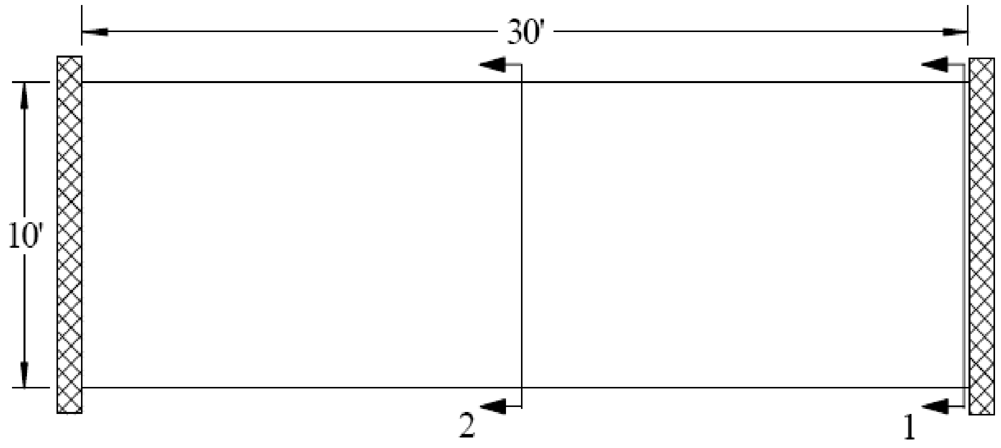
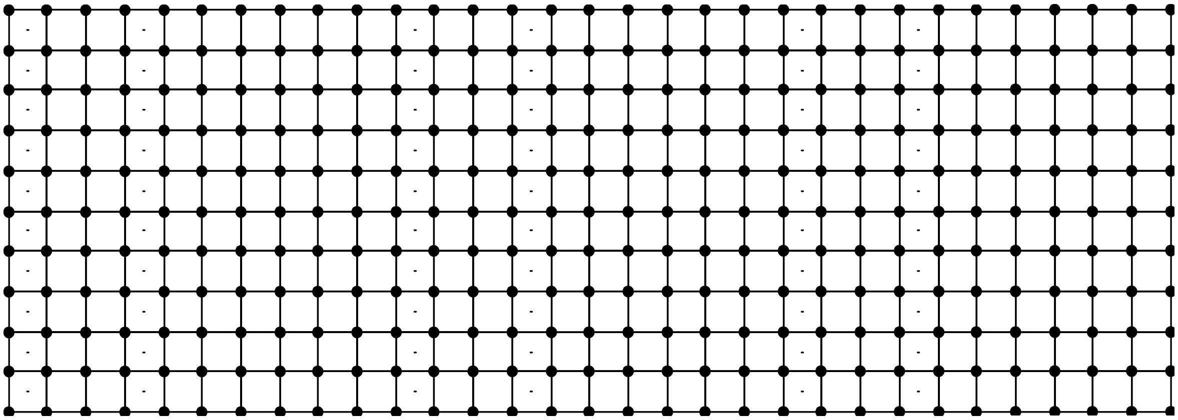


Figure 5.2.7-1 Example Flat Plate Structure (PLAN)

GT STRUDL Finite Element Model

The example structure was modeled in GT STRUDL using PLATE BENDING finite elements. The BPHQ element was utilized, and the configuration modeled corresponded to a mesh of ten elements by thirty elements. The model contained 300 finite elements and 341 nodes. The material properties were the default values associated with the MATERIAL REINFORCED CONCRETE command. All 6 degrees of freedom were restrained at each node along the supported ends of the slab system. Each element was loaded with a surface pressure of 150 psf, resulting in a confirmed summation of vertical reaction of 45,000 lb.

Figure 5.2.7-2 Example Finite Element Model



Definition of Cut Cross-Sections

Two “cuts” are considered for the verification example, as shown in Figure 5.2.7-1.

Cut 1-1:

The cross-section Cut 1-1 is defined along the fixed support at the end of the slab strip and represents the maximum “negative moment” section in the slab where top reinforcing steel would be required. Cut 1-1 originates at node #1 and terminates at node #11. The elements along Cut 1-1 are elements #1-#10. The command given for Cut 1-1 is:

“design slab using calculate resultant joi 1 11 ele 1 top bar 5"

In this case, the user requests that a slab cross-section beginning at node #1, ending at node #11, and in the plane of element #1 be reinforced according to the section moment computed using the CALCULATE RESULTANT command. The user has specified that #5 bars are to be used on the top surface, indicating that spacing is to be computed. The results of the DESIGN SLAB command are shown in the following table.

Calculation	Surface	Bar	Spacing	Area Prov.	Moment Strength	Moment Required
		#	in	sq. in.	lb-in	lb-in
DESIGN SLAB	Top	5	13.0	2.862	1561006.4	1354381.5
DESIGN SLAB	Bottom	NA	NA	NA	NA	NA

The GTSTRUDL output for this example is as follows:

```
  ** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **

       PROBLEM - VFE103    TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS


       RELEVANT ACTIVE UNITS:  INCH LB

       NUMBER OF ACTIVE LOADINGS:     1

       REINFORCEMENT ORIENTATION PERPENDICULAR TO A CUT BEGINNING AT NODE 1
         AND TERMINATING AT NODE 11       AND IN THE PLANE OF ELEMENT 1


  ** ELEMENT FORCE IMPLEMENTATION **


  ** DESIGN MOMENT ENVELOPE **

       NEGATIVE MOMENT =      -1354381.48   DUE TO LOAD   150psf
       POSITIVE MOMENT =             0.00   DUE TO LOAD   (none)

       NOTE:
        - Negative moment produces tension on the positive PLANAR Z surface, requiring TOP
bars.
        - Positive moment produces compression on the positive PLANAR Z surface, requiring
BOTTOM bars.


  ** SLAB CROSS-SECTION **

       Width        Depth          FCP            FY         Cover         Layer
      ________________________________________________________________________

      120.00       12.00        4000.00       60000.00     0.750         Inner


  ** DESIGN RESULTS (per ACI 318-05) **

       Face       Bar    Spacing   AS PROV'D    MOMENT STRENGTH    MOMENT REQ'D    STATUS

__________________________________________________________________________________________

       TOP        # 5    13.000      2.862        1561006.4280    1354381.4844    PASSES

       BOTTOM             ( Reinforcement Not Required )
```

Cut 2-2:

The cross-section Cut 2-2 is defined along the center line in the middle region of the slab strip and represents the maximum “positive moment” section in the slab where bottom reinforcing steel would be required. Cut 2-2 originates at node #166 and terminates at node #176. The elements along Cut 2-2 are elements #141-#150 on one side and #151-#160 on the other side. The command given for Cut 2-2 Case 1 is:

> “design slab wood and armer joi 166 176 ele 141 table unesco bottom spacing 10 outer layer"

In this case, the user requests that a slab cross-section beginning at node #166, ending at node #176, and in the plane of element #141 be reinforced according to the average effect produced by the Wood and Armer method. The user has specified that UNESCO metric reinforcing bars are to be used. The bottom reinforcement spacing has been constrained to 10 inches, and the reinforcement to be designed is located in the outer layer. The results of the DESIGN SLAB command are shown in the following table:

Calculation	Surface	Bar	Spacing	Area Prov.	Moment Strength	Moment Required
		#	in	sq. in.	lb-in	lb-in
DESIGN SLAB	Bottom	M14	10.0	2.864	1664920.7	671358.2
DESIGN SLAB	Top	NA	NA	NA	NA	NA

The GTSTRUDL output for this example is as follows:

```
 ** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **

      PROBLEM - VFE103    TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS


      RELEVANT ACTIVE UNITS:  INCH LB

      NUMBER OF ACTIVE LOADINGS:     1

      REINFORCEMENT ORIENTATION PERPENDICULAR TO A CUT BEGINNING AT NODE 166
        AND TERMINATING AT NODE 176      AND IN THE PLANE OF ELEMENT 141


 ** WOOD & ARMER IMPLEMENTATION **

      Design using average result acting on section.


 ** DESIGN MOMENT ENVELOPE **

      NEGATIVE MOMENT =            0.00   DUE TO LOAD   150psf
      POSITIVE MOMENT =       671358.19   DUE TO LOAD   150psf
```

```
NOTE:
       - Negative moment produces tension on the positive PLANAR Z surface, requiring TOP
bars.
       - Positive moment produces compression on the positive PLANAR Z surface, requiring
BOTTOM bars.

  ** SLAB CROSS-SECTION **

       Width        Depth          FCP           FY        Cover        Layer
      ______________________________________________________________________

      120.00       12.00        4000.00       60000.00     0.750        Outer

  ** DESIGN RESULTS (per ACI 318-05) **

       Face       Bar    Spacing   AS PROV'D    MOMENT STRENGTH    MOMENT REQ'D    STATUS
___________________________________________________________________________________________

       TOP                 ( Reinforcement Not Required )

       BOTTOM     M14    10.000      2.864        1664920.7190     671358.1875     PASSES
```

This page intentionally left blank.

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

$$\underline{\text{CAL}}\text{CULATE } \underline{\text{ERR}}\text{OR } (\underline{\text{EST}}\text{IMATE}) \ (\underline{\text{BAS}}\text{ED } \underline{\text{O}}\text{N}) \ \ -$$

$$* \left\{ \begin{array}{l} \underline{\text{ENERGY}} \ (\underline{\text{NOR}}\text{M}) \\ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \\ \underline{\text{PERCENT}} \ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{PERCENT}} \ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \\ \underline{\text{NORM}}\text{ALIZED} \ \underline{\text{PERCENT}} \ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{NORM}}\text{ALIZED} \ \underline{\text{PERCENT}} \ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \end{array} \right\} \ -$$

$$(\underline{\text{AT}}) \ \overset{*}{\left\{ \begin{array}{l} \underline{\text{TOP}} \\ \underline{\text{MID}}\text{DLE} \\ \underline{\text{BOT}}\text{TOM} \end{array} \right\}} \ (\underline{\text{SUR}}\text{FACES}) \ (\underline{\text{FOR}}) \ \left\{ \begin{array}{l} \rightarrow \underline{\text{ALL}} \\ \\ \underline{\text{ELE}}\text{MENT list} \end{array} \right\}$$

The results from this command provide an estimate of the errors in the finite element discretization of the problem. Energy norm (L_2 norm) and nodal error estimates are available.

The L_2 norm is given by:

$$\left\| e_\sigma \right\|_{L2} = \left(\int_\Omega \left(e_\sigma \right)^T \left(e_\sigma \right) d\Omega \right)^{1/2}$$

where e_σ is the error in stress and Ω is the domain of the element. The error stress is the difference between the average stress, σ^*, and element stress at the nodes, σ. The stress norm is obtained by using the shape functions used for displacements, thus,

$$\left\| e_\sigma \right\|_{L2} = \left(\int_\Omega (\sigma^* - \sigma)^T N^T \cdot N (\sigma^* - \sigma) \ d\Omega \right)^{1/2}$$

where N is the shape functions used for the assumed displacement field of the element.

The stress norm uses the average stresses and is given by:

$$\|\sigma\|_{L2} = \left(\int_{\Omega} \left(\sigma^*\right)^T N^T \cdot N\left(\sigma^*\right) d\Omega \right)^{1/2}$$

The relative percentage error which is output for each element is given by:

$$\eta = \frac{\|e_\sigma\|}{\|\sigma\| + \|e_\sigma\|} \times 100$$

The nodal error estimates estimate the accuracy of the data in a selected nodal output vector. Six nodal error estimation methods are available:

- Maximum Difference.
- Difference from Average.
- Percent Maximum Difference.
- Percent Difference from Average.
- Normalized Percent Maximum Difference.
- Normalized percent Difference from Average.

These error estimates look at the variations in stresses at the nodes. An error estimate of nodal output data will be based on the gradients that data produces in each element. That is, how the data varies across that node based on the different data values from the elements connected at that node. The calculation of error estimates for nodal output is fairly straightforward, the values at each node connected at an element are simply compared. The six nodal error measures are outlined in more detail below:

Maximum Difference Method

$$\left| \text{Value}_{\text{Max}} - \text{Value}_{\text{Min}} \right|$$

Difference from Average Method

$$\text{MAX}\ (\left|\text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}}\right|, \left|\text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}}\right|)$$

Percent Maximum Difference Method

$$\left| \frac{\text{Value}_{\text{Max}} - \text{Value}_{\text{Min}}}{\text{Value}_{\text{Avg}}} \right| \times 100\%$$

Percent Difference from Average Method

$$\frac{\text{MAX}\left(\left|\text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}}\right|, \left|\text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}}\right| \right)}{\left|\text{Value}_{\text{Avg}}\right|} \times 100\%$$

Normalized Percent Maximum Difference

$$\left| \frac{\text{Value}_{\text{Max}} - \text{Value}_{\text{Min}}}{\text{Value}_{\text{VectorMax}}} \right| \times 100\%$$

Normalized Percent Difference from Average Method

$$\frac{\text{MAX}\left(\left|\text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}}\right|, \left|\text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}}\right| \right)}{\left|\text{Value}_{\text{VectorMax}}\right|} \times 100\%$$

In each of these calculations, the “Min”, “Max”, and “Avg” values refer to the minimum, maximum, and average output values at the node. The “Vector Max” values refer to the maximum value for all nodes in the output vector. All error estimates are either zero or positive, since all use the absolute value of the various factors.

The choice of an appropriate error estimation method largely depends on the conditions in the model. As many error estimates as required may be calculated. In general, the Max Difference method is good at pointing out the largest gradients in the portions of your model with the largest output values. The Difference from Average Method will also identify areas with the largest output values. In this case however, areas where only one or a few values are significantly different will be accentuated. The Max Difference method will identify the steepest gradients in the most critical portions of your model. The Difference from Average Method will identify just the steepest non-uniform gradients, the ones that vary in only a single direction. The two percentage methods identify the same type of gradients, but do not make any distinction between large and small output values. These methods are to be used only if the magnitude of the output is less important than the changes in output. The two percentage methods estimate the error as a percent of the average stress. However, at nodes where there is a change in sign of the stress, the average stress becomes very small and often close to zero. As a result, the value of the error becomes enormous. In order to quantify this error, the error at such nodes is given a value of 1,000 percent. The final two normalized percentage methods are usually the best at quantifying overall errors in area with peak stress values.

The results produced by the CALCULATE ERROR ESTIMATE command may also be contoured in GTMenu. To produce a contour of the error estimate in GTMenu, follow the steps below after performing a STIFFNESS ANALYSIS for a static loading:

1. Enter GTMenu.

2. Select Results, Finite Element Contours, and then Energy & Stress Error Estimates.

3. Select the Estimate Method including Value, Surface, and Stress Component.

4. Select the Loading.

5. Select Display (solid colors or lines) to produce a contour of the error estimate.

6. Select Legend to place a legend on the screen indicating the type of error estimate, loading, and surface.

5.3.2 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

The Sections shown below are numbered as they will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.4.3.7 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

This section describes the commands that are used to incorporate the viscous damper element (dash pot) into a structural model that is used for linear and nonlinear dynamic analysis by the direct integration procedure. The commands that are used for this purpose include:

1. DAMPER ELEMENT DATA, described in Section 2.4.3.7.1.
2. PRINT DAMPER ELEMENT DATA, described in Section 2.4.3.7.2.
3. DELETE DAMPER ELEMENT DATA, described in Section 2.4.3.7.3.

2.4.3.7.1 The DAMPER ELEMENT DATA Command

Tabular form:

$$\underline{\text{DAMP}}\text{ER}\ \ \underline{\text{ELE}}\text{MENT}\ \ (\underline{\text{DAT}}\text{A})$$

$$\begin{Bmatrix} i_D \\ 'a_D' \end{Bmatrix} \underline{\text{INC}}\text{IDENCES} \begin{Bmatrix} i_S \\ 'a_S' \end{Bmatrix} (\begin{Bmatrix} i_E \\ 'a_E' \end{Bmatrix}) \begin{Bmatrix} \rightarrow \underline{\text{GLO}}\text{BAL} \\ \underline{\text{LOC}}\text{AL} \end{Bmatrix} -$$

$$[\underline{\text{CTX}}]\ v_{CTX}\ \ [\underline{\text{CTY}}]\ v_{CTY}\ \ [\underline{\text{CTZ}}]\ v_{CTZ}\ \ [\underline{\text{CRX}}]\ v_{CRX}\ \ [\underline{\text{CRY}}]\ v_{CRY}\ \ [\underline{\text{CRZ}}]\ v_{CRZ}$$

•

•

•

$$\begin{Bmatrix} i_D \\ 'a_D' \end{Bmatrix} \underline{\text{INC}}\text{IDENCES} \begin{Bmatrix} i_S \\ 'a_S' \end{Bmatrix} (\begin{Bmatrix} i_E \\ 'a_E' \end{Bmatrix}) \begin{Bmatrix} \rightarrow \underline{\text{GLO}}\text{BAL} \\ \underline{\text{LOC}}\text{AL} \end{Bmatrix} -$$

$$[\underline{\text{CTX}}]\ v_{CTX}\ \ [\underline{\text{CTY}}]\ v_{CTY}\ \ [\underline{\text{CTZ}}]\ v_{CTZ}\ \ [\underline{\text{CRX}}]\ v_{CRX}\ \ [\underline{\text{CRY}}]\ v_{CRY}\ \ [\underline{\text{CRZ}}]\ v_{CRZ}$$

$$\underline{\text{END}}\ \ (\underline{\text{O}}\text{F}\ \ \underline{\text{D}}\text{AMPER}\ \ \underline{\text{E}}\text{LEMENT}\ \ \underline{\text{D}}\text{ATA})$$

Elements:

i_D/’a_D’ = integer or alphanumeric name of the new damper element. The name must be unique among all previously defined damper elements and is restricted to no more than eight digits or alphanumeric characters.

i_S/’a_S’ = integer or alphanumeric name of a previously defined joint to be the starting incident joint of the new damper element.

i_E/’a_E’ = optional integer or alphanumeric name of the previously defined joint to be the ending incident joint of the new damper element. The starting joint and ending joint names must be different.

v_{CTX} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL X direction. Active force, length, and time units apply [force/(length/time)].

v_{CTY} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL Y direction. Active force, length, and time units apply [force/(length/time)].

v_{CTZ} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL Z direction. Active force, length, and time units apply [force/(length/time)].

v_{CRX} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].

v_{CRY} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].

v_{CRZ} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].

Explanation:

The DAMPER ELEMENT DATA command is used to create new viscous damper elements and define their joint connectivity and damping force and moment properties. The viscous damper element data are entered by giving the DAMPER ELEMENT DATA command header first, followed by one or more tabular element data entry lines of the form:

$$\begin{Bmatrix} i_D \\ 'a_D' \end{Bmatrix} \underline{\text{INC}}\text{IDENCES} \begin{Bmatrix} i_S \\ 'a_S' \end{Bmatrix} \left(\begin{Bmatrix} i_E \\ 'a_E' \end{Bmatrix}\right) \begin{Bmatrix} \rightarrow \underline{\text{GLO}}\text{BAL} \\ \underline{\text{LOC}}\text{AL} \end{Bmatrix} -$$

$$[\underline{\text{CTX}}]\ v_{CTX} \quad [\underline{\text{CTY}}]\ v_{CTY} \quad [\underline{\text{CTZ}}]\ v_{CTZ} \quad [\underline{\text{CRX}}]\ v_{CRX} \quad [\underline{\text{CRY}}]\ v_{CRY} \quad [\underline{\text{CRZ}}]\ v_{CRZ}$$

for each new damper element. This data entry line consists of the element name, the element incidences, the element orientation, and the element viscous damping coefficients, which are described in greater detail as follows:

Element name $\begin{Bmatrix} i_D \\ 'a_D' \end{Bmatrix}$

Each new damper element must be given an integer or alphanumeric name that is unique among all other existing damper element names. The name may not exceed eight digits or alphabetic characters. The name may be a duplicate of a previously defined member or finite element name.

$\underline{\text{INC}}\text{IDENCES} \begin{Bmatrix} i_S \\ 'a_S' \end{Bmatrix}$

The damper element connectivity is defined by one or two incident joints. The first incident joint, i_S/’a_S’, defines the start of the element. The second incident joint, i_E/’a_E’, is optional and defines the end of the element. If only one joint is given, the second joint is taken as a totally fixed support joint; it is fictitious and invisible. The specified joints must have been previously defined and if two are specified, they must be different. However, they may be coincident. The only restriction on the selection of incident joints is that they may not be slave joints.

$$\left\{\begin{matrix}\rightarrow \underline{\text{GLO}}\text{BAL} \\ \underline{\text{LOC}}\text{AL}\end{matrix}\right\}$$

The GLOBAL and LOCAL options are used to specify the coordinate reference frame for the damper element. The GLOBAL option, which is the default, means that the element is a global element and that the six element damping degrees-of-freedom are defined with respect to the global coordinate system. The LOCAL option means that the element damping degrees-of-freedom are defined with respect to the element local coordinate system, which is identical to the local joint-to-joint coordinate system for frame members. The only difference between the frame member and damper element local coordinate systems is that the damper element does not support the Beta angle. If the LOCAL option is specified, but the joint-to-joint length of the element is equal to 0 (# 10^{-5} inches), then GLOBAL is assumed. In addition, GLOBAL is automatically assumed for any damper element for which only one incident joint is specified.

$[\underline{\text{CTX}}]\ v_{CTX}\ [\underline{\text{CTY}}]\ v_{CTY}\ [\underline{\text{CTZ}}]\ v_{CTZ}\ [\underline{\text{CRX}}]\ v_{CRX}\ [\underline{\text{CRY}}]\ v_{CRY}\ [\underline{\text{CRZ}}]\ v_{CRZ}$

These decimal data values represent the damping coefficient values on the diagonal of the uncoupled element damping matrix, which has the following form:

$$\begin{bmatrix} \text{CTX} & 0 & 0 & 0 & 0 & 0 \\ & \text{CTY} & 0 & 0 & 0 & 0 \\ & & \text{CTZ} & 0 & 0 & 0 \\ & (\text{sym}) & & \text{CRX} & 0 & 0 \\ & & & & \text{CRY} & 0 \\ & & & & & \text{CRZ} \end{bmatrix}$$

These values refer to the element damping translational and rotational degrees-of-freedom with respect to the specified coordinate system, GLOBAL, the default, or LOCAL. Only non-zero values need be specified.

Command processing is completed when the END option is given.

The damping properties from the viscous damper elements are assembled into the total global system damping matrix of the equations of motion that are solved using the direct integration methods executed by the DYNAMIC ANALYSIS PHYSICAL and DYNAMIC ANALYSIS NONLINEAR commands. The viscous damper element data are used only by the execution of these two commands

Modifications:

The DAMPER ELEMENT DATA command operates only in the ADDITIONS mode. If the command is given when the active input mode is CHANGES or DELETIONS, then the command execution is terminated and the command data are ignored. If it is necessary to change the data for an existing damper element, then use the DELETE DAMPER ELEMENT command described in Section 2.4.3.7.3 to delete the damper element to be changed, followed by the re-specification of the new data in the DAMPER ELEMENT DATA command. All of these steps are performed in ADDITIONS mode.

Example:

The following example illustrates the creation of two damper elements DAMP1 and DAMP2. DAMP1 spans from joint 2 to joint 10 and has one damping coefficient equal to 10^7 kips/(inches/second) corresponding to translation in the local y direction of the element. DAMP2 spans from joint 1 to joint 2 and has global damping factors CTX = 100 kips/(inches/second) and CRZ = 1000 kip-inches/(radians/second). The damping coefficients for element DAMP2 are referenced with respect to the global coordinate system because the GLOBAL/LOCAL option was not given. The execution of this example depends on DAMP1 and DAMP2 not having been previously defined and joints 1, 2, and 10 being valid joints.

```
UNITS KIPS INCHES RADIANS
DAMPING ELEMENT DATA
  'DAMP1' INC 2 10 LOCAL CTY 1.E7
  'DAMP2' INC 1 2 CTX 100.0 CRZ 1000.0
END
```

Errors:

1. When two or more damper elements are defined with the same name, the following warning message is printed. Command processing is terminated for the offending element and continues for subsequent elements.

```
{   10} > DAMPING ELEMENT DATA
{   11} >   'DAMP1' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{   12} >   'DAMP1' INC 2 4 GLOBAL CTY 1.E7

**** WARNING_STDELD -- Damper element DAMP1 previously defined.
                       Command ignored.

{   13} >   'DAMP3' INC 3 3 GLOBAL CTY 1.E7
{   14} > END
```

Element DAMP1 is successfully created by the first tabular command entry. The warning message for DAMP1 is printed for the second tabular entry for DAMP1. Command processing continues with the tabular entry for DAMP3.

2. The following warning message is printed if one or both of the specified element incidence joints are not defined. Command processing continues with the tabular entry for the next element.

```
{   10} > DAMPING ELEMENT DATA
{   11} >   'DAMP1' INC 2 10 LOCAL CTY 1.E7

**** WARNING_STDELD -- Damper element incidence joint not defined.
                       Command ignored.

{   12} >   'DAMP2' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{   13} > END
```

The warning message indicates that one or both of the specified element incidences for element DAMP1 are not defined.

3. The following warning message is printed when the starting and ending element incidence joints are the same. Command processing continues with the tabular entry for the next element.

```
{   10} > DAMPING ELEMENT DATA
{   12} >   'DAMP1' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{   13} >   'DAMP2' INC 2 4 GLOBAL CTY 1.E7
{   14} >   'DAMP3' INC 3 3 GLOBAL CTY 1.E7

**** WARNING_STDELD -- Damper element starting and ending incident joints are the
                      same.  Command ignored.

{   15} >   'DAMP4' INC 4 5 CTY 1.E7
{   16} > END
```

2.4.3.7.2 The PRINT DAMPER ELEMENT DATA Command

General form:

PRINT DAMPER (ELEMENT DATA)

Explanation:

The PRINT DAMPER ELEMENT DATA is used to print a table of the damper element data for all existing damper elements. The following is an example of the printed output from this command:

Example:

The following example illustrates the format for the output from the PRINT DAMPER ELEMENT command.

```
{   17} > PRINT DAMPING ELEMENT DATA

********************************
* RESULTS FROM LATEST ANALYSIS *
********************************


ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
    LENGTH          WEIGHT          ANGLE          TEMPERATURE          TIME
     FEET            LB              RAD              DEGF             SEC


Damping Element Data
====================

Element       Start Jnt    End Jnt     CTX       CTY            CTZ         CRX            CRY        CRZ
-------       ---------    -------     -----     -----          -----       -----          -----      -----

DAMP1    LOC  1            2           100.0    0.0000E+00    0.0000E+00   0.0000E+00   0.0000E+00     1000.
DAMP2    GLO  2            4         0.0000E+00 0.1000E+08    0.0000E+00   0.0000E+00   0.0000E+00   0.0000E+00
```

Errors:

The following warning message is printed when no damper element data exists.

```
{    9} > PRINT DAMPING ELEMENT DATA

 ********************************
 * RESULTS FROM LATEST ANALYSIS *
 ********************************


 ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
     LENGTH          WEIGHT          ANGLE          TEMPERATURE          TIME
      FEET             LB             RAD               DEGF             SEC


 Damping Element Data
 ====================

 Element           Start Jnt   End Jnt         CTX          CTY     ...       CRZ
 -------           ---------   -------        -----        -----             -----

 **** INFO_STPDED -- Damper element data have not been defined.
```

2.4.3.7.3 The DELETE DAMPER ELEMENT DATA Command

General form:

DELETE DAMPER (ELEMENT DATA) $\left\{\begin{matrix} i_D \\ 'a_D' \end{matrix}\right\} \ldots \left\{\begin{matrix} i_D \\ 'a_D' \end{matrix}\right\}$

Elements:

i_D/’a_D’ = integer or alphanumeric name of damper element to be deleted. The name is limited to no more that eight digits or characters.

Explanation:

This command is used to delete previously defined damper elements. The names of the elements to be deleted are given in the list of individually named damper elements. No other list construct, such as “1 TO 10" is permitted. Specified damper elements that are not defined are ignored.

This page intentionally left blank.

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

The ROTATE LOAD command will rotate an existing loading and create a new loading condition in order to model a different orientation of the structure or the loading. The ROTATE command is described below and is numbered as it will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.11.4.6 The ROTATE LOAD Command

General form:

$$\underline{\text{ROT}}\text{ATE}\ \ \underline{\text{LOA}}\text{DING}\ \left\{\begin{matrix} i_R \\ \\ 'a_R' \end{matrix}\right\}\ (\ \underline{\text{ANG}}\text{LES}\ )\ \left[\underline{\text{T1}}\right] r_1\ \left[\ \underline{\text{T2}}\ \right] r_2\ \left[\ \underline{\text{T3}}\right] r_3$$

Elements:

i_R/'a_R' = integer or alphanumeric name of the existing independent loading condition whose global components are to be rotated.

r_1, r_2, r_3 = values in current angle units of the load component rotation angles 2_1, 2_2, 2_3 as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

In many instances, loading conditions are defined for a structure having a given orientation in space, but then the same structure may need to be analyzed for different additional orientations. Applied loading components that are defined with respect to local member or element coordinate systems remain unchanged regardless of the structure's orientation. However, loading components that are defined with respect to the global coordinate system may need to be rotated in order to properly reflect a new orientation for the structure. This is particularly true for self-weight loads, buoyancy loads, etc.

The ROTATE LOADING command is used to take the global applied loading components from an existing loading condition, rotate them through a set of rotation angles, and copy the new rotated global components to a new or modified different destination loading condition. The existing independent loading condition, the ROTATE load, from which the rotated global load components are computed is specified by the loading name i_R/'a_R'. The angles of rotation are specified by the values r_1, r_2, r_3. These rotation angles are defined according to the same conventions as those that define the local support release directions in the JOINT RELEASE command described in Section 2.1.7.2, Volume 1 of the GTSTRUDL User Reference Manual, and illustrated in Figure 2.1.7-1.

The ROTATE LOADING command is always used in conjunction with one of the following loading definition commands: LOADING, DEAD LOAD, and FORM LOAD. These commands will define the name (and title) of a new or existing destination loading condition into which the ROTATE LOADING results are copied. The ROTATE LOADING command may be given with any additional applied loading commands such as JOINT LOADS, MEMBER LOADS, ELEMENT LOADS, etc.

Taking the specified loading i_R/'a_R', the ROTATE LOADING command performs the following operations and copies the results into the destination loading condition:

1. Rotate all joint loads, including applied joint support displacements.
2. Rotate all member force and moment loads defined with respect to the global coordinate system. Member force and moment loads defined with respect to the member local coordinate system are simply copied without rotation.
3. Rotate all element force loads defined with respect to the global coordinate system. Element force loads defined with respect to any applicable local or planar coordinate systems are copied without rotation.
4. All other types of loads such as member temperature loads, member distortions, joint temperatures, etc. are copied without changes.

Examples:

1.
```
UNITS DEGREES
LOADING 2 ‘ROTATED LOADING’
MEMBER DISTORTIONS
     1 TO 10 UNIFORM FR LA 0.0 LB 1.0 DISPL X 0.001
ROTATE LOADING 1 ANGLES T1 45.0
```

The applied loads from previously defined loading 1 will be processed according to Steps 1 to 4 above and copied into the new destination loading 2, which includes the specified member distortion loads applied to members 1 to 10.

2.
```
UNITS DEGREES
CHANGES
LOADING 3
ADDITIONS
ROTATE LOAD 4 ANGLES T2 -30.0
```

Previously defined loading 3 is specified in CHANGES mode, followed by a return to ADDITIONS mode. The ROTATE LOAD command is then given to add the components of load 4, including appropriate rotations, to loading 3.

Error Messages:

Incorrect data given in the ROTATE LOADING command will cause the following error conditions to be identified and error messages printed:

1. The following error message is printed if the ROTATE loading name is identical to the name of the destination load. An example of the commands that produce this error are also included:

```
{   114} > LOADING 201
{   115} > ROTATE LOAD 201 T1 45.0

**** ERROR_STROLO –  The  ROTATE  loading  is  illegally  the  same as the
                     destination loading.
                     Command ignored.
```

Loading 201 is illegally named as both the destination load and the loading whose components are rotated.

2. In the following error example, loading 51 is undefined.

```
{   111} > LOADING 201
{   112} > ROTATE LOAD 51 T1 45.0

**** ERROR_STROLO –  Loading to be rotated undefined.
                     Command ignored.
```

3. The following error message is produced because loading 4, specified as the ROTATE load, is a load combination, or dependent loading condition. The ROTATE load must be an independent loading condition.

```
{   141} > LOADING 108
{   142} > ROTATE LOADING 4 T3 45.0

**** ERROR_STROLO –   Rotated Loading 4 is an illegal dependent load.
                      Command ignored.
```

4. This error condition and message is caused by the fact that the destination load 108 is defined as a loading combination.

```
{   144} > LOAD COMB 108 ‘ALL’ COMBINE 1 1.5 2 1.0 3 1.0
{   145} > ROTATE LOADING  1 T3 45.0

**** ERROR_STROLO –   Destination independent loading not defined.
                      Rotated load components not computed.
```

5.4.2 COUTPUT Command

The COUTPUT command now can replace (overwrite) an existing output file. Previously, an existing file could be appended only.

$$\underline{\text{COUTPUT}}\quad (\ \left\{\begin{array}{l} \rightarrow\underline{\text{APP}}\text{END} \\ \underline{\text{REP}}\text{LACE} \\ \underline{\text{STA}}\text{NDARD} \end{array}\right\}\ )\ (\text{'file\_name'})$$

where,

'file_name' is a new or existing text file. 'file_name' is limited to 256 characters and must be enclosed in quotes (apostrophes).

Explanation:

APPEND is the default action, so "COUTPUT 'file1'" and "COUTPUT APPEND 'file1'" are equivalent. APPEND tells GTSTRUDL to add subsequent output to the end of the specified file. If APPEND is requested, 'file_name' must be given.

REPLACE tells GTSTRUDL to delete the contents of the specified file and the write subsequent output to the specified file. If REPLACE is requested, 'file_name' must be given.

APPEND and REPLACE act identically when 'file_name' does not already exist. While GTSTRUDL is in the APPEND or REPLACE state, only input (commands) are echo printed in the text window - all generated output will be placed in the specified output file.

STANDARD tells GTSTRUDL to stop writing to the specified output file and direct subsequent output to the text window. This is the output state when GTSTRUDL starts.

Usage:

COUTPUT APPEND 'file1'

All subsequent output, from PRINT, LIST, etc., will be written to 'file1' and will not appear in the text window, although the actual command will be displayed in the text window. If 'file1' existed previously to this COUTPUT request, the new output will appear at the end of the existing contents.

COUTPUT REPLACE 'file2'

All subsequent output, from PRINT, LIST, etc., will be written to 'file2' and will not appear in the text window, although the actual command will be displayed in the text window. If 'file2' existed previously to this COUTPUT request, the existing contents will be deleted and only the new output will appear in 'file2'.

COUTPUT STANDARD

Stop writing output to an output file and write all output to the text window.

5.4.3 Reference Coordinate System Command

General form:

$$\underline{\text{REF}}\text{ERENCE (}\underline{\text{COO}}\text{RDINATE) (}\underline{\text{SYS}}\text{TEM)} \begin{Bmatrix} i_1 \\ \text{'}a_1\text{'} \end{Bmatrix} \ -$$

$$\left\{ \begin{array}{l} (\underline{\text{ORI}}\text{GIN [}\underline{\text{X}}\text{] } v_x \text{ [}\underline{\text{Y}}\text{] } v_y \text{ [}\underline{\text{Z}}\text{] } v_z\text{) (}\underline{\text{ROT}}\text{ATION [}\underline{\text{R1}}\text{] } v_1 \text{ [}\underline{\text{R2}}\text{] } v_2 \text{ [}\underline{\text{R3}}\text{] } v_3\text{)} \\ \left\{ \begin{array}{l} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ \\ \text{'}a_2\text{'} \end{Bmatrix} \\ \underline{\text{X}}\ v_4 \ \ \underline{\text{Y}}\, v_5 \ \ \underline{\text{Z}}\ v_6 \end{array} \right\} \left\{ \begin{array}{l} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ \\ \text{'}a_2\text{'} \end{Bmatrix} \\ \underline{\text{X}}\ v_4 \ \ \underline{\text{Y}}\, v_5 \ \ \underline{\text{Z}}\ v_6 \end{array} \right\} \left\{ \begin{array}{l} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ \\ \text{'}a_2\text{'} \end{Bmatrix} \\ \underline{\text{X}}\ v_4 \ \ \underline{\text{Y}}\, v_5 \ \ \underline{\text{Z}}\ v_6 \end{array} \right\} \end{array} \right\}$$

Explanation:

The REFERENCE COORDINATE SYSTEM is a right-handed three-dimensional Cartesian coordinate system. The Reference Coordinate System's origin may be shifted from the origin (X=0.0, Y=0.0, Z=0.0) of the overall global coordinate system. The Reference Coordinate System axes may also be rotated from the corresponding orthogonal axes of the overall global coordinate system.

At the present time, this command is used to specify additional coordinate systems which may be used in GTMenu (see Volume 2 of the GTSTRUDL Release Guide) to facilitate the creation of the structural model. Reference Coordinate systems created using the above command will be available as Local systems in GTMenu. In a future release, the user will be able to output results such as joint displacements and reactions in a Reference Coordinate System.

There are two optional means of specifying a Reference Coordinate System:

(1) Define the origin and rotation of coordinate axes of the reference system with respect to the global coordinate system, and

(2) define three joints or the coordinates of three points in space.

In either case, i_1 or 'a_1' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x, v_y, and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x, v_y, and v_z, are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive directions of these axes; v_1, v_2, and v_3 are the rotation angles R_1, R_2, and R_3 in active angular units between the orthogonal axes of this system and the axes of the overall global coordinate system. The description of these angles is the same as given in Section 2.1.7.2 of Volume 1 of the GTSTRUDL User Reference Manuals for rotated joint releases (2_1, 2_2, and 2_3).

In the second case, three joints are required. Each of the three joints may be defined either by a joint identifier using the JOINT option of the command or by its global X, Y, and Z coordinates. If the joint identifier option is used, however, the coordinates of the joint must be specified previously by the JOINT COORDINATES command. The first time (i_2 or 'a_2' or v_4, v_5, and v_6) defines the origin of the reference system; the X-axis of the reference system is determined by the first and second joints (i_3 or 'a_3' or v_7, v_8, and v_9). The positive X-axis is directed from the first to the second joint. The third joint (i_4 or 'a_4' or v_{10}, v_{11}, and v_{12}) is used to define the XY-plane of the reference system. The positive Y-axis is directed toward the third joint. The Z-axis then is determined by the right-hand rule.

Only one reference system can be specified in one command, but the command may be used any number of times.

Modifications of Reference Systems:

In the changes mode, the translations of the origin and/or the rotations of the axes of the reference system from those of the overall global system can be changed. Only that information supplied in the command is altered. The other data that might be supplied in the command remains unchanged. The CHANGES mode, however, does not work for the second option discussed above (i.e., define a reference coordinate system by three joints or the coordinate of three points in space). The reason is that data for these joints are not stored permanently in GTSTRUDL. When this option is used, a reference system is created and its definitions of the system origin, rotation angles, as well as the transformation matrix between the global coordinate system and the reference system are generated and stored as would be for the first option. Therefore, if any of the coordinates for the joints used to specify a reference system is changed after the REFERENCE COORDINATE SYSTEM command has been given, the definition of the reference system remains unchanged. For this reason, care must be taken in using the three joints option in conjunction with the changes of joint coordinates. The reference system should be deleted first if any of the coordinates of the joints used to define the reference system are to be changed. Under the DELETIONS mode, the complete definition of the reference coordinate system is destroyed.

Examples:

```
a)  UNITS DEGREES
    REFERENCE COORDINATE SYSTEM ‘FLOOR2’  -
                  ORIGIN 0.0 15.0 0.0 R1 30.
```

This command creates a Reference Coordinate System called FLOOR2 at Y=15 with the axes rotated 30 degrees about global Z.

```
b)  REF COO 1  -
                  X    120   Y   120   Z   -120  -
                  X    120   Y   240   Z      0  -
                  X   -120   Y   120   Z      0
```

This command creates Reference Coordinate System 1 with its origin at 120, 120, -120 and its X-axis from this origin to 120, 240, 0 and its Y axis is the plane defined by the two previous coordinates and the third coordinate, -120, 120, 0, with the positive Y-axis directed toward the third coordinate.

```
c)  REFERENCE COORDINATE SYSTEM 2  -
                  JOINT 10 JOINT 20 JOINT 25
```

This command creates Reference Coordinate System 2 with its origin located at Joint 10 and its X-axis directed from Joint 10 toward Joint 20. The XY plane is defined by Joints 10, 20, and 25 with the positive Y-axis directed toward Joint 25.

```
d)  CHANGES
    REFERENCE COORDINATE SYSTEM ‘FLOOR2’  -
                  ORIGIN   10   20   30
    ADDITIONS
```

The above commands change the origin of the Reference System FLOOR2 defined in a) above. The rotation RI = 30 remains unchanged.

```
e)  DELETIONS
    REFERENCE  SYSTEM 2
    ADDITIONS
```

The above command deletes Reference System 2.

5.4.3-1 Printing Reference Coordinate System Command

General form:

$$\text{\underline{PRI}NT \underline{REF}ERENCE (\underline{COO}RDINATE) (\underline{SYS}TEM)} \begin{Bmatrix} \rightarrow \text{ALL} \\ \text{list} \end{Bmatrix}$$

Explanation:

The PRINT REFERENCE COORDINATE SYSTEM command will output the Reference Systems. The origin and rotation angles will be output.

5.4.4 Hashing Algorithm to Accelerate Input Processing

An advanced data-structuring technique called HASHING can now be used when storing and searching lists of joints and/or elements. The command to control this feature is as follows:

$$\underline{\text{SET}}\ \underline{\text{ELE}}\text{MENTS} \left\{ \begin{array}{l} \underline{\text{HAS}}\text{HED} \\ \\ \underline{\text{SEQ}}\text{UENTIAL} \end{array} \right\}$$

The following points concern HASHING:

1) The benefit of HASHING is that it GENERATES large structures faster. The disadvantage is that it is more complex internally.

2) HASHING is disabled by GTMenu. The GTSTRUDL database is usually not modified extensively in GTSTRUDL after invoking GTMenu, so this has minimal affect. However, the SET ELEMENTS HASHED command, when given with an existing database, builds hashing data structures for the existing database.

3) The order of a joint and/or element listing is the same for HASHED and SEQUENTIAL unless the structural database has been edited in DELETIONS mode and then in ADDITIONS mode again. Then SEQUENTIAL will place the latest addition in the deleted slot whereas HASHING will append the addition to the end of the list.

This page intentionally left blank.

5.4.5 GTMenu Point and Line Incidences Commands

GTMenu can now write construction geometry commands to an input file, which can be read later into GTSTRUDL in order to initialize the construction geometry of GTMenu. The two commands written are “GTMenu POINT COORDINATES” and “GTMenu LINE INCIDENCES”.

(1) GTMenu POINT COORDINATES

General Form:

GTMenu POINT COORDINATES

$$\left\{ \begin{array}{c} i_1 \\ 'a_1' \end{array} \right\} \text{coordinate-specs}_1$$

A
A
A

$$\left\{ \begin{array}{c} i_n \\ 'a_n' \end{array} \right\} \text{coordinate-specs}_n$$

Elements:

coordinate-specs = [X] v_1 [Y] v_2 [Z] v_3

Where,

$i_1, i_2, ..., i_n$ = unsigned integer Point identifiers.

‘a_1', ‘a_2', ..., ‘a_n’ = 1 to 8 character alphanumeric Point identifiers.

v_1, v_2, v_3 = Cartesian Point coordinates (integer or real).

(2) GTMenu LINE INCIDENCES

General Form:

$$\underline{\text{GTMenu}}\ \underline{\text{LIN}}\text{E}\ \underline{\text{INC}}\text{IDENCES}$$

$$\left\{\begin{matrix} i_1 \\ \\ 'a_1' \end{matrix}\right\}\ \text{type}_1\ \ \text{incidence-specs}_1$$

A
A
A

$$\left\{\begin{matrix} i_n \\ \\ 'a_n' \end{matrix}\right\}\ \text{type}_n\ \ \text{incidence-specs}_n$$

Elements:

$$\text{type} \quad = \quad \left\{\begin{array}{l} \rightarrow\ \underline{\text{LIN}}\text{E} \\ \underline{\text{POL}}\text{YNOMINAL}\ \ (\underline{\text{CUR}}\text{VE}) \\ \underline{\text{ARC}}\ \ (\underline{\text{TEM}}\text{PLATE}) \\ \underline{\text{CEN}}\text{TERED}\ \ (\underline{\text{ARC}})\ \ \underline{\text{PER}}\text{CENT}\ \ v_1 \\ \underline{\text{BEZ}}\text{IER}\ \ (\underline{\text{CUR}}\text{VE}) \\ \underline{\text{SPL}}\text{INE}\ \ (\underline{\text{CUR}}\text{VE})\ \ (\underline{\text{ORD}}\text{ER}\ \ k_2) \end{array}\right\}$$

$$\text{incidence-specs} \quad = \quad \left\{\begin{matrix} i_1 \\ \\ 'a_1' \end{matrix}\right\} \left\{\begin{matrix} i_2 \\ \\ 'a_2' \end{matrix}\right\} . \ . \ . \ \left\{\begin{matrix} i_p \\ \\ 'a_p' \end{matrix}\right\}$$

Where,

$i_1, i_2, ..., i_n$ = unsigned integer Line/Curve identifiers.

$'a_1', 'a_2', ..., 'a_n'$ = 1 to 8 character alphanumeric Line/Curve identifiers.

$i_1, i_2, ..., i_p$ = unsigned integer Point identifiers used.

$'a_1', 'a_2', ..., 'a_p'$ = 1 to 8 character alphanumeric Point identifiers.

v_1 = positive number (integer or real).

k_2 = integer between 2 and the number of incidences.

1, 2, ...,p = Point subscripts for a Line/Curve. The following table gives the number of Points used to specify different types of Line/Curve:

type	number of incidences
LINE	2 - 500
POLYNOMIAL CURVE	2 - 10
ARC TEMPLATE	3
CENTERED ARC	3
BEZIER CURVE	2 - 10
SPLINE CURVE	2 - 10

End of Document.