GT STRUDL ®

GT STRUDL
Release Guide Version 28

Volume 1 of 2

January 2005

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® Release Guide is applicable to Version 28, with a release date in the GTSTRUDL title block of January 2005.

The GTSTRUDL® 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

Commercial Software Rights Legend

Table of Contents

Chapter 1

Introduction

Version 28 covers GTSTRUDL operating on PC's under the Windows XP, Windows 2000, Windows NT, Windows ME, and Windows 98 operating systems. Chapter 2 presents the new features and enhancements which have been added since the Version 27 release. Chapter 3 provides you with details regarding error corrections that have been made since the Version 27 release. Chapter 4 describes known problems with Version 28. 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 in Version 28 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 LRFD3 Tables
- 5.2.3 BS5950 Steel Design Code and Parameters
- 5.2.4 Steel Design by Indian Standard Code IS800
- 5.2.5 IS800 Tables
- 5.2.6 Steel Deflection Check and Design
- 5.2.7 Brazilian Table
- 5.2.8 ACI Code 318-99
- 5.2.9 Rectangular and Circular Concrete Cross Section Tables
- 5.2.10 ASD9-E Code

5.3 Analysis Prerelease Features
- 5.3.1 Calculate Error Estimate Command
- 5.3.2 Output of Response Spectrum Results
- 5.3.3 Form Static Load Command
- 5.3.4 Form UBC97 Load Command
- 5.3.5 Form IS1893 Load Command
- 5.3.6 Element Properties Command for Nonlinear Hysteretic Spring Element
- 5.3.7 Nonlinear Dynamic Analysis
- 5.3.8 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

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

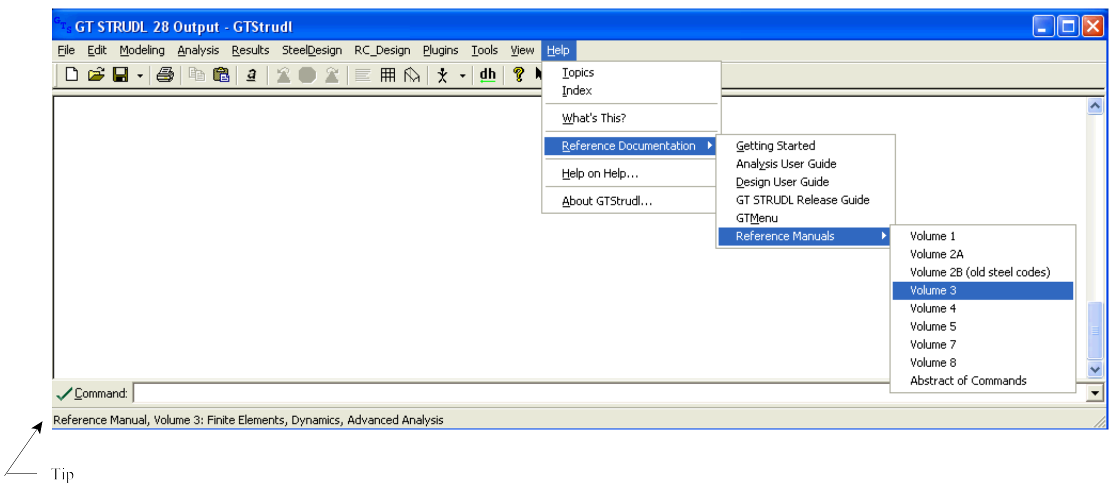
Note that GTMenu is described in Volume 2 of this Release Guide.

Chapter 2
New Features in Version 28

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 28. This release guide is also available online upon execution of GTSTRUDL under Help/Reference Documentation/GTSTRUDL Release Guide. Other documentation has also changed in Version 28. You should also review the following online documentation which is also available under Help:

- GTMenu
- Getting Started
- Analysis Guide
- Design Guide

Furthermore, the GTSTRUDL Reference Manuals are available under Help as shown below:

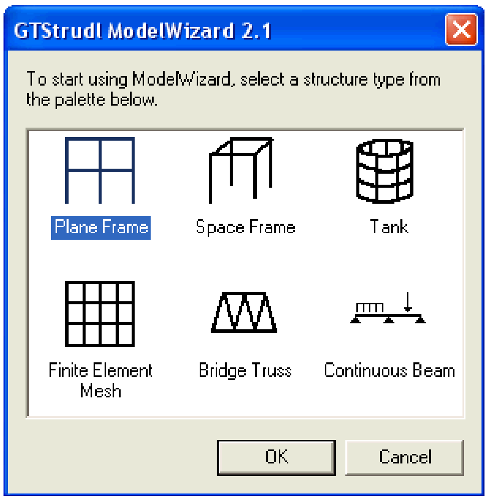


A tip on the contents of the various Reference Manuals is shown at the bottom of the above window under the Command line. In the above example, the tip indicates “Reference Manual, Volume 3: Finite Elements, Dynamics, Advanced Analysis.”

2.1 GTSTRUDL Startup

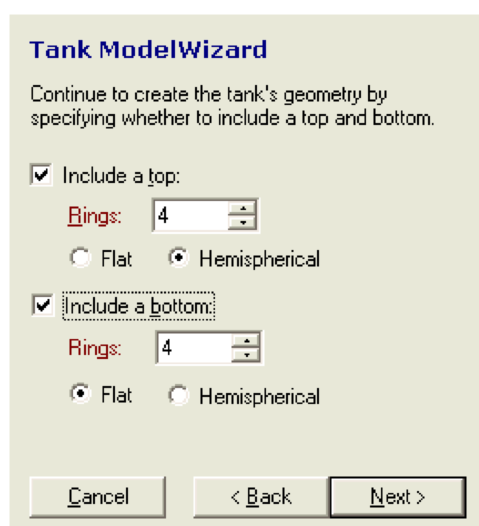
Model Wizard

1. A new Tank option has been added to the Model Wizard as shown below:

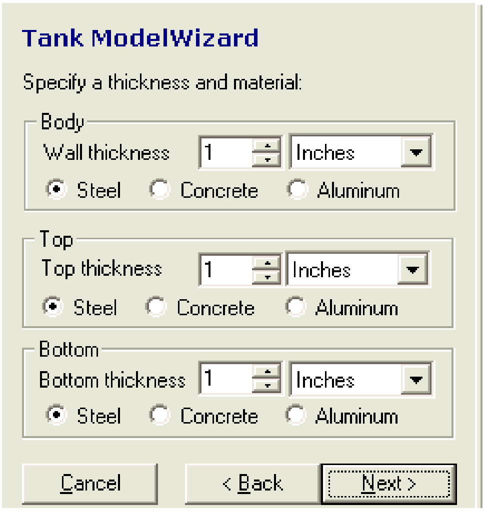


The new Tank Wizard can be used to generate finite element meshes of circular tanks with flat or hemispherical shaped top and bottom surfaces using SBHQ6 or SBHT6 elements. Self weight, internal pressure hydrostatic loads are also created. Furthermore, supports may also be created including soil springs at the bottom of the tank. Examples of several of the Tank Wizard dialogs are shown below:

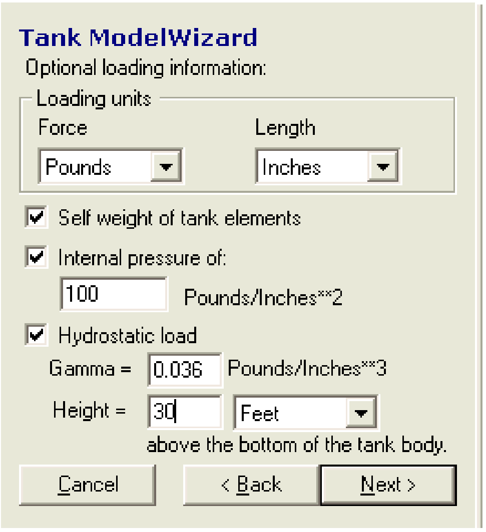
Geometry and top and bottom surfaces dialogs;



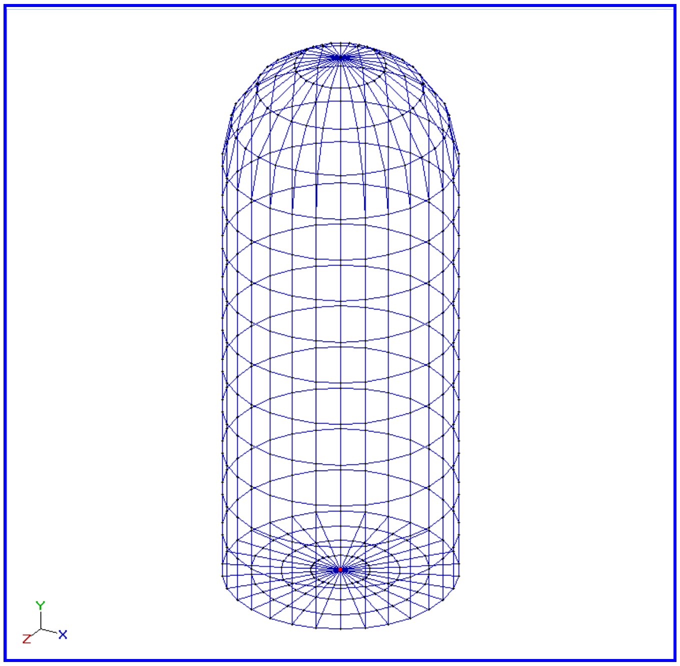
Thickness and Support Specification dialogs including soil springs on the bottom surface of tank;



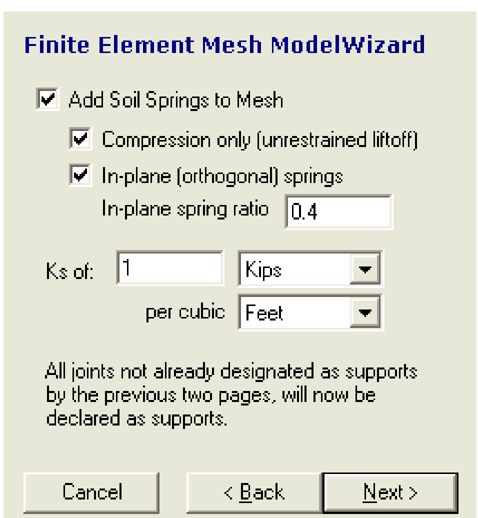
Self weight, internal pressure, and hydrostatic load dialog;



An example of a tank created using the new Tank Wizard is shown below;

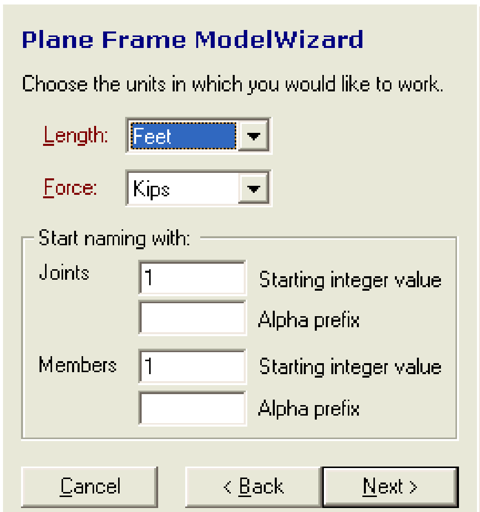


The Finite Element Mesh Model Wizard now includes In-Plane springs and Compression Only options for springs normal to the plane of the element. The Compression Only spring option will create a nonlinear spring curve and nonlinear spring elements. In-plane springs will be elastic springs. An example of the new dialog for springs in the Finite Element Model Wizard is shown below:

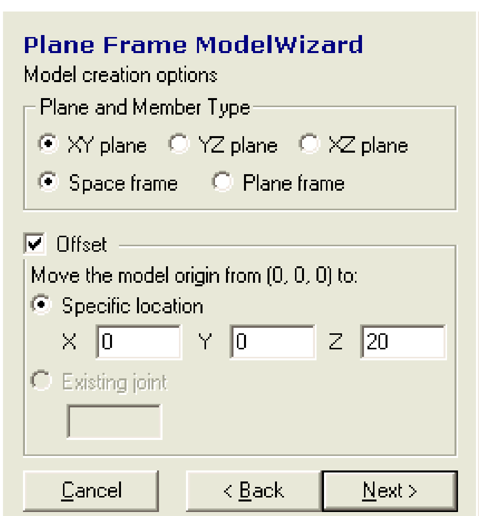


If the Compression Only option is used in the above dialog, a nonlinear compression only spring curve is created and nonlinear spring elements are connected to every node with the equivalent stiffness in the compression region of the nonlinear spring curve and zero stiffness in the tensile region of the spring curve.

All of the Wizards available in the Model Wizard now allow you to control the starting names of joints, members, or elements created in the various Wizards including the ability to specify alphanumeric names. A dialog similar to the one shown below is available in all of the Wizards:



The Plane Frame Model Wizard now provides you the ability to create the model with either Space Frame or Plane Frame members with Space Frame being the default. In addition, you may now specify an offset from the origin in all of the Wizards. This offset will be added to the coordinates created in the Wizard. An example of one of the Plane Frame Model Wizard dialogs is shown below illustrating the member type and offset specifications:

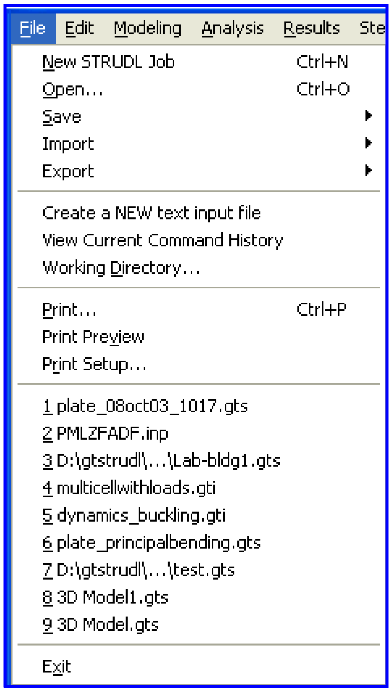


2.2 GTSTRUDL Output Window

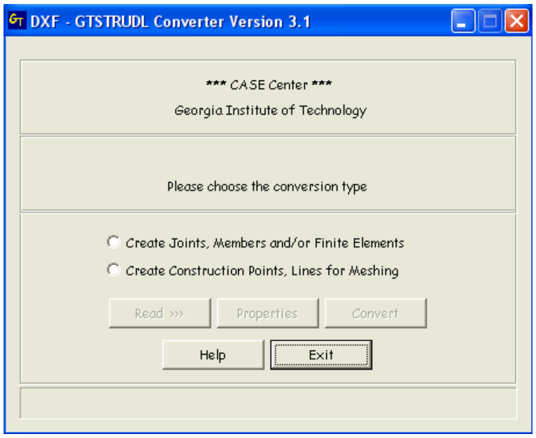
Numerous new features and enhancements have been added to the GTSTRUDL Output Window. These additions are presented over the following pages by the menu headings located at the top of the GTSTRUDL Output Window.

File Pulldown

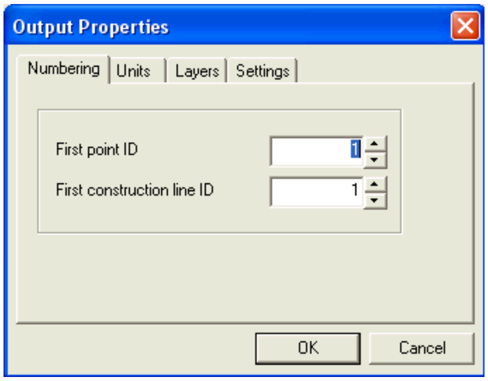
The new File pulldown is shown:



The Import DXF option has been enhanced to convert lines and polylines into GTMenu Construction Points and Lines which can later be used to create the model in GTMenu. The initial dialog that appears now gives you the option to create Joints, Members, and Elements, or Construction Points and Lines as shown below:



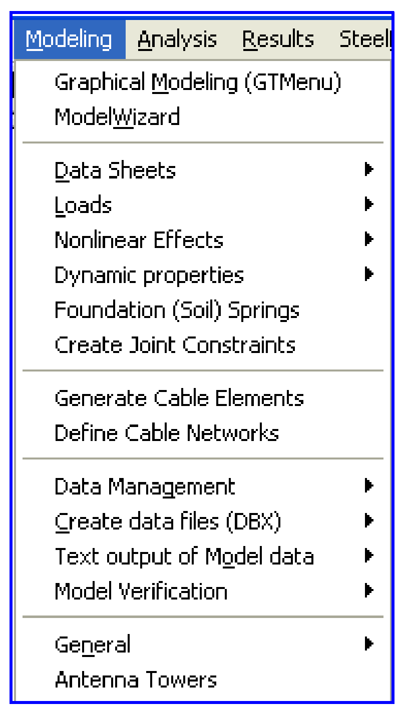
The Properties option in the above dialog allows you to specify the starting names (IDs) for the Construction Points and Lines as shown below:



Other tabs in the dialog allow you to specify the Units for the Construction Points, the Layers to be used in the DXF file, and the Settings to be used.

The "Create a New Text Input File" option is a new feature added to the File pulldown. This allows you to create a text input file as you would from GTMenu without requiring you to enter GTMenu. You will be prompted for the input file name and the input file created using this option has the same limitations as described in the GTMenu Release Guide. You should review the GTMenu Release Guide under Help 6 Reference Documentation 6 GTMenu to understand these limitations.

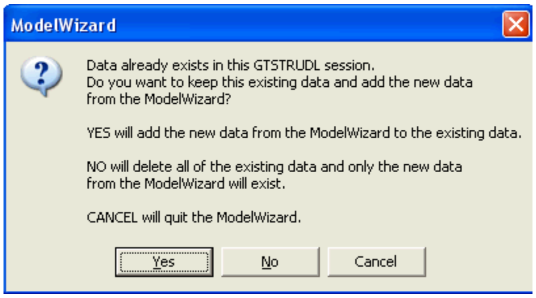
Modeling Pulldown



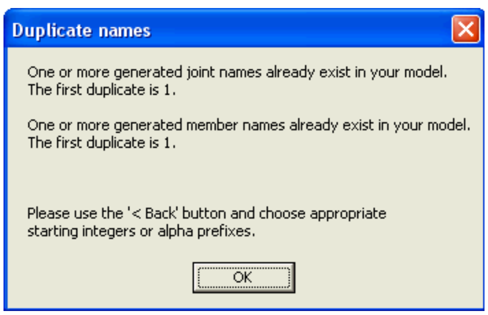
The Modeling pulldown has been restructured and new features have been added. The modified Modeling pulldown is shown :

A feature that has been requested by a number of users is the ability to create additions to an existing model using the Model Wizard. As shown above, the Model Wizard is now the second line in the Modeling pulldown. When using this new feature with an existing model, you should verify that you have specified the starting names (joint, member, and element IDs) for the new portion of the model so as to avoid duplicate names with the existing model.

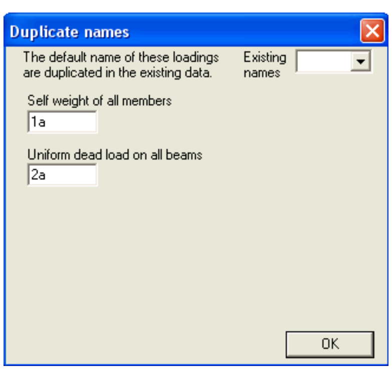
If data already exist when the Model Wizard is initiated, the following popup dialog will appear:



The new options in the Model Wizard should be used to specify new starting names for joints, members, or elements to avoid duplicate names with the existing model. A check is made at the end of the Model Wizard to determine if there are any conflicts with the joint, member, or element names. If duplicates exist, a dialog such as the one shown below will appear:

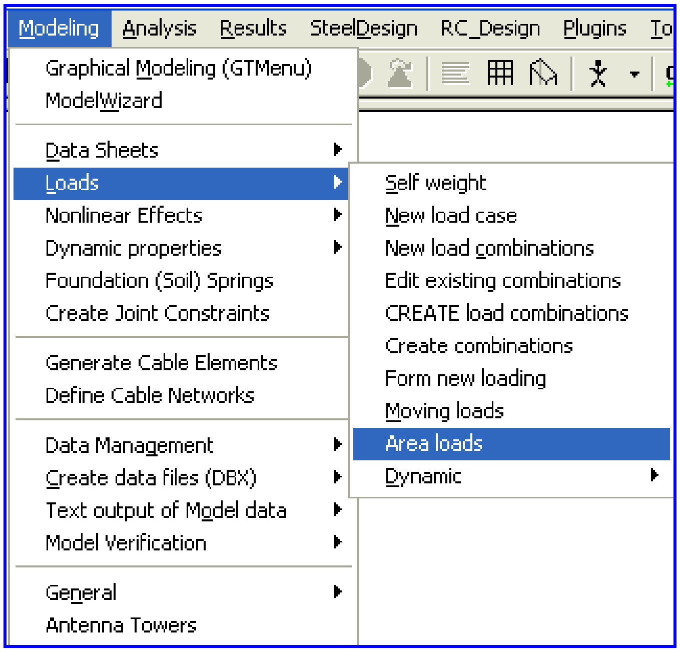


If duplicate load names exist, the following dialog will appear which will allow you to rename the new loadings:

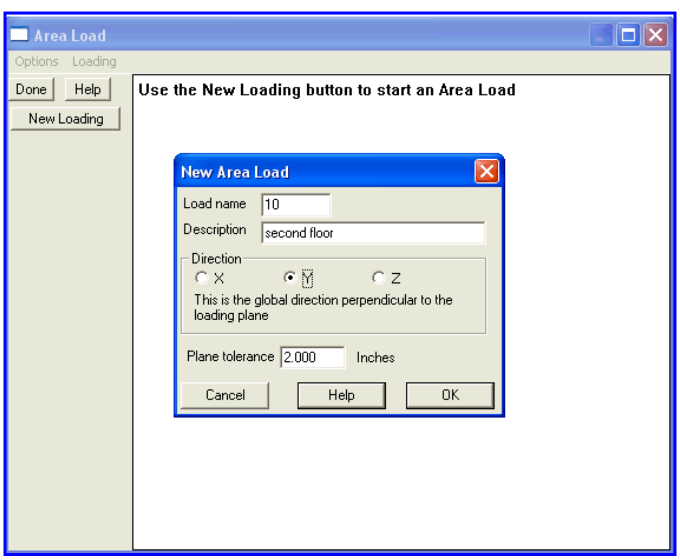


You can not append to existing loadings but you can use the Form Load command to merge various independent loadings.

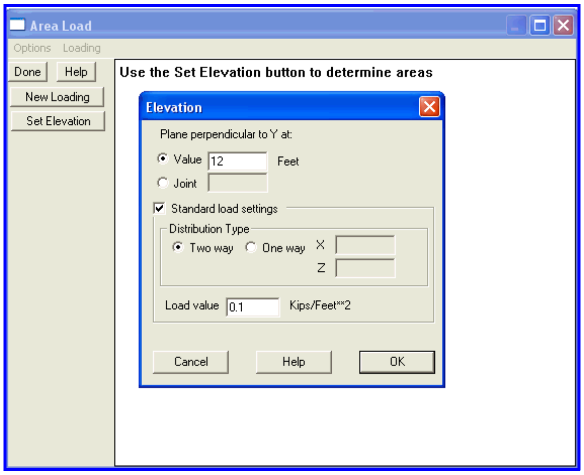
A new option is available under Loads to create Area Loads as shown below:



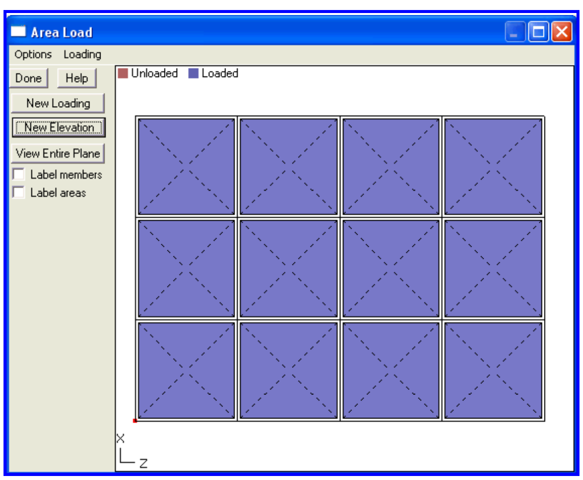
An example of the new Area Load dialog is shown below:



After establishing the loading name, description, and direction, you will then set elevations for the area load using either the coordinates of the plane, or a joint in the plane. The loads will be distributed to the members on a particular elevation using either one way or two way distribution. An example of this dialog is shown below:



For two way systems, the area load is distributed to surrounding members by tributary area. As shown below, division lines in the loaded areas indicate the tributary areas.

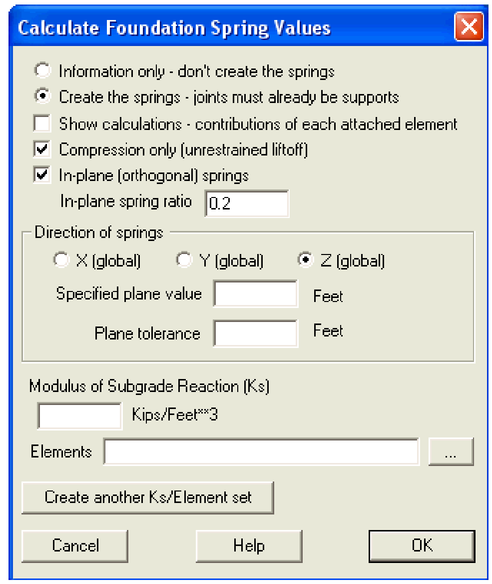


For one way systems, the area load is distributed to the surrounding members in the direction specified. This will leave members parallel to the one way direction unloaded. You will see lines in the one way areas parallel to the loading direction. The values in the direction boxes (labeled **X** and **Z** in the example) are ratios, so any two equal numbers will yield a 45 degree angle. Any number in the **X** box and 0 in the **Z** box would result in a one way distribution in the X global direction.

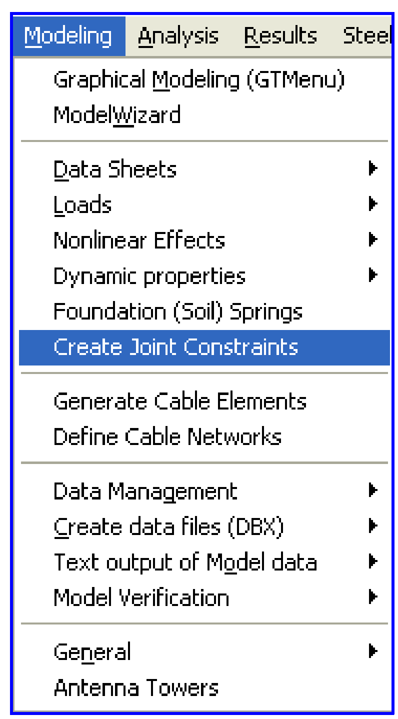
The loads and regions may also be edited to remove any loadings or change the value of loadings in selected regions.

The new Area Load feature is further described in Section 2.5.

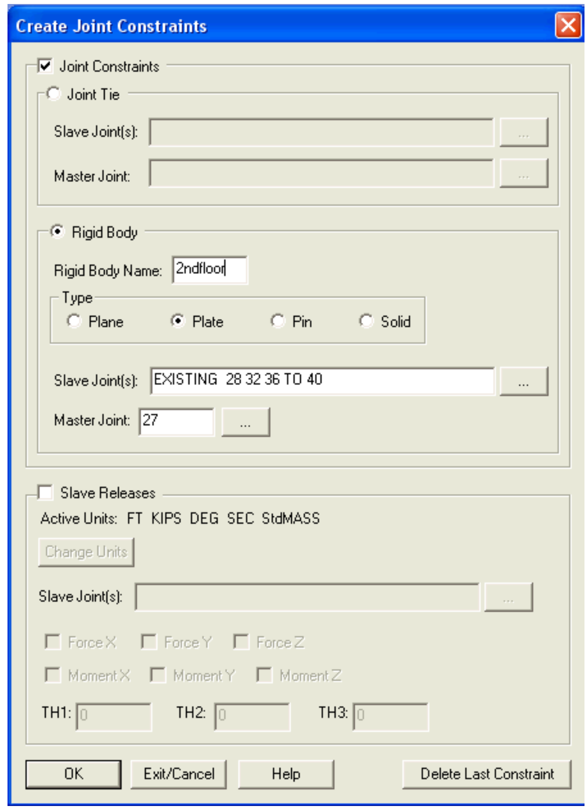
The Foundation (Soil) Springs option under the Modeling pulldown now has the ability to create Compression Only springs and to also create In-plane springs which are a ratio of the normal springs. The new Calculate Foundation Spring Values dialog is shown:



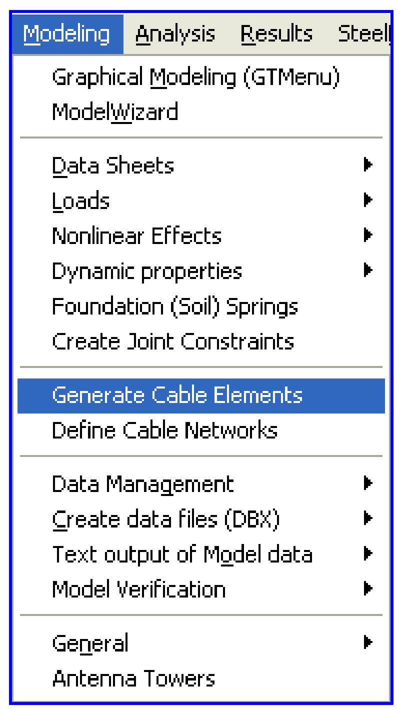
Joint Constraints may now be created under the Modeling pulldown as shown:



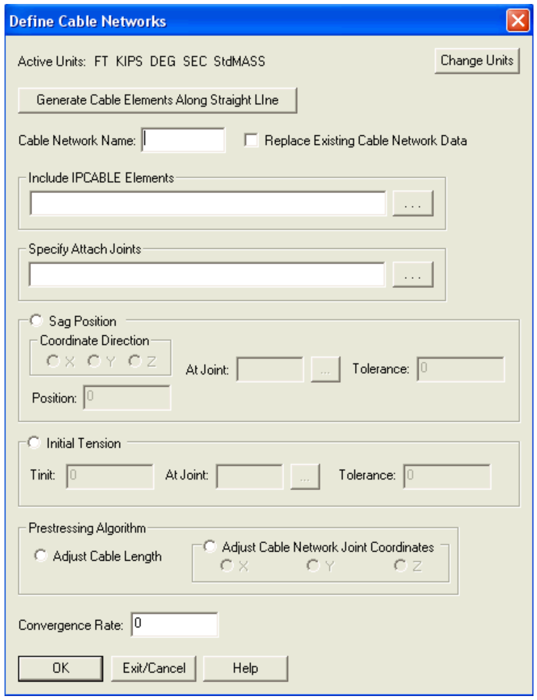
This figure illustrates the new Joint Constraint dialog:

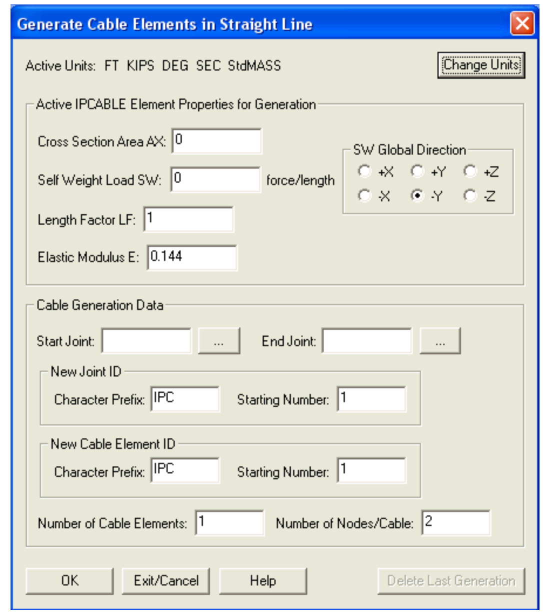


Cable elements and cable networks may now be created under the Modeling pulldown as shown:



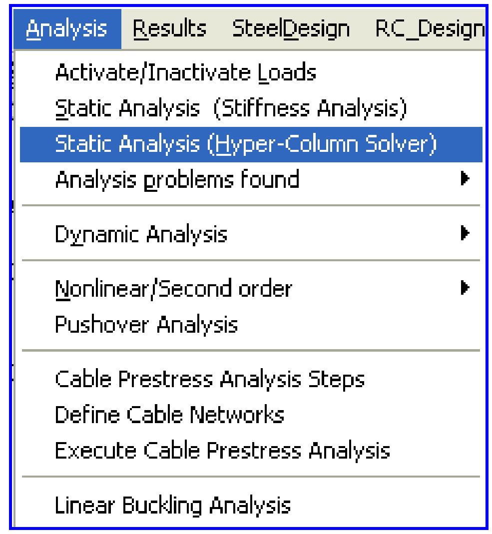
The new Cable Element and Cable Network dialogs are shown below:





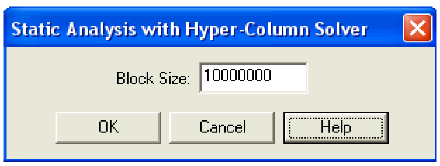
The Antenna Towers option shown at the bottom of the Modeling pulldown is a separately licensed feature of GTSTRUDL. If you are interested in licensing this feature, contact your GTSTRUDL representative.

Analysis Pulldown



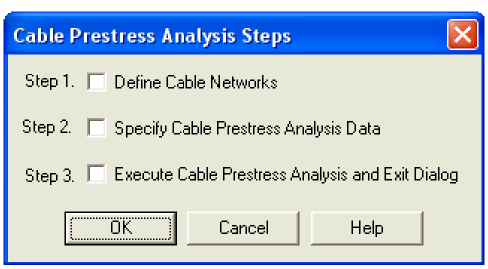
A new static analysis equation solver (Hyper-Column Solver) has been implemented in Version 28 and added to the Analysis pulldown as shown:

This new equation solver will solve extremely large problems much faster than the standard equation solver used in STIFFNESS ANALYSIS. In addition, this new equation solver will use external files during the solution procedure, thus alleviating the Windows limitation on virtual storage (page file). The Hyper-Column Solver dialog is shown below:

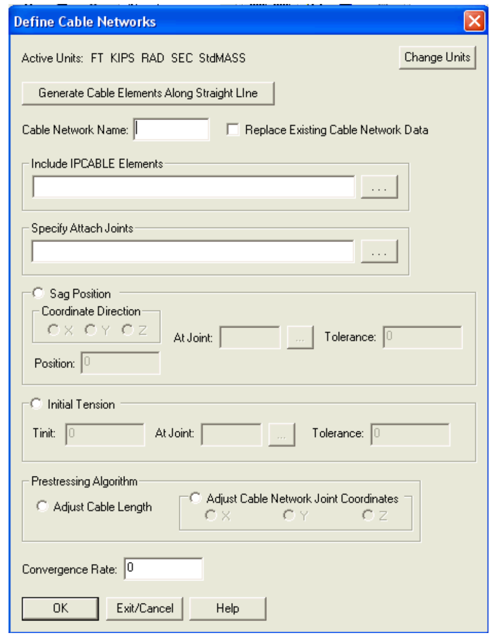


The new Hyper-Column Solver is further discussed in Section 2.12 along with some examples which illustrate the solver's efficiency and ability to solve problems which could not be solved in previous versions of GTSTRUDL due to the Windows virtual memory limitation.

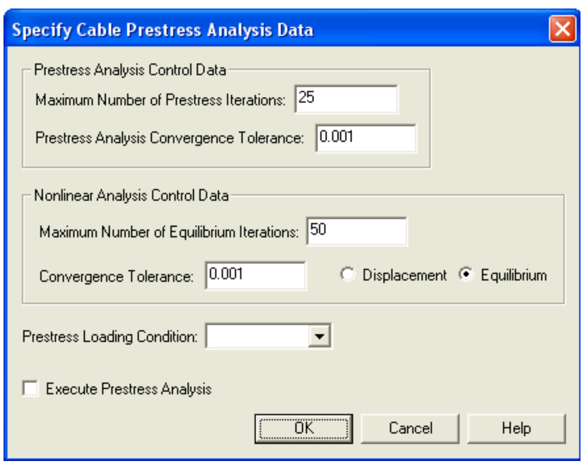
Also added to the Analysis pulldown are dialogs for the cable definition, solution control, and prestress analysis for cable problems. These options are labeled Cable Prestress Analysis Steps, Define Cable Networks, and Execute Cable Prestress Analysis. The Cable Prestress Analysis Steps dialog shown below will step through the procedure for a cable prestress analysis:



The Define Cable Networks dialog is used to define the cable networks and is shown:

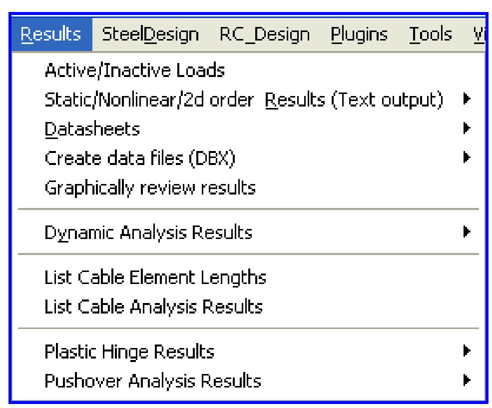


The Specify Cable Prestress Analysis Data dialog shown below is used to specify cable prestress analysis data and to execute the cable prestress analysis:

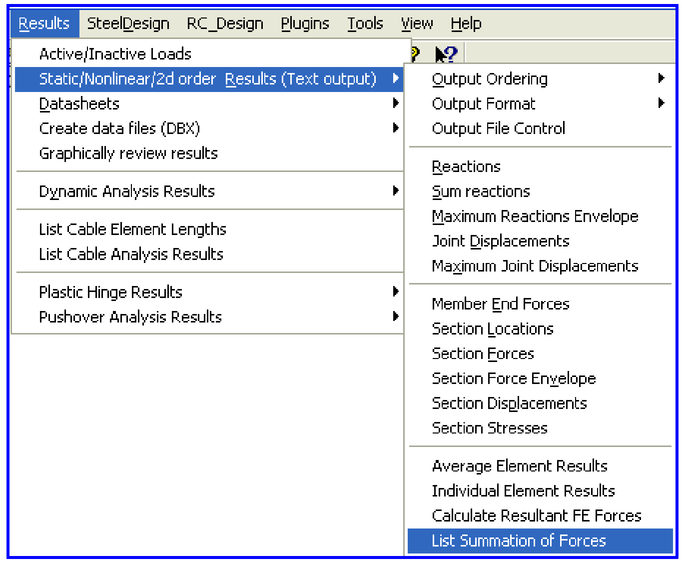


Results Pulldown

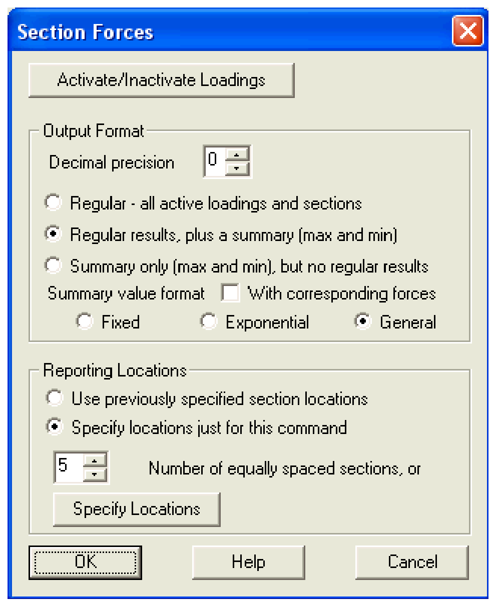
The Results pulldown has been modified and new options have been added in Version 28. The revised Results pulldown is shown:



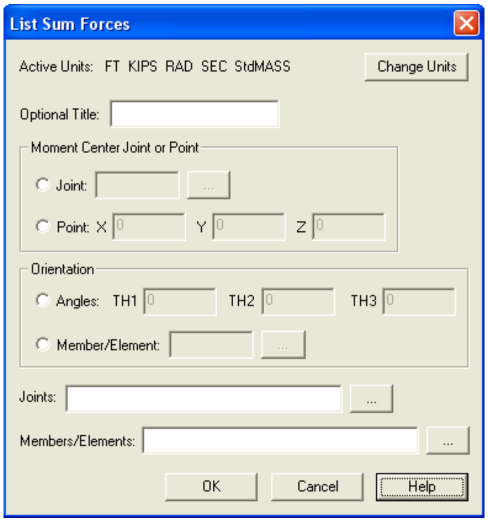
The options available under Static Results have been modified as shown in the following pulldown:



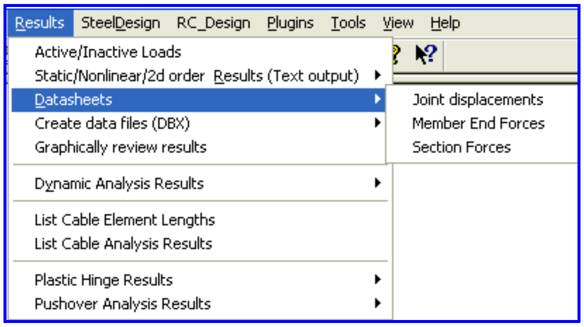
A new option has been added to the Section force option to include the output of corresponding forces. For example, for the section location and loading that causes the maximum value of MZ, the associated values of other force and moment components will also be output. The modified Section Forces dialog is shown below:



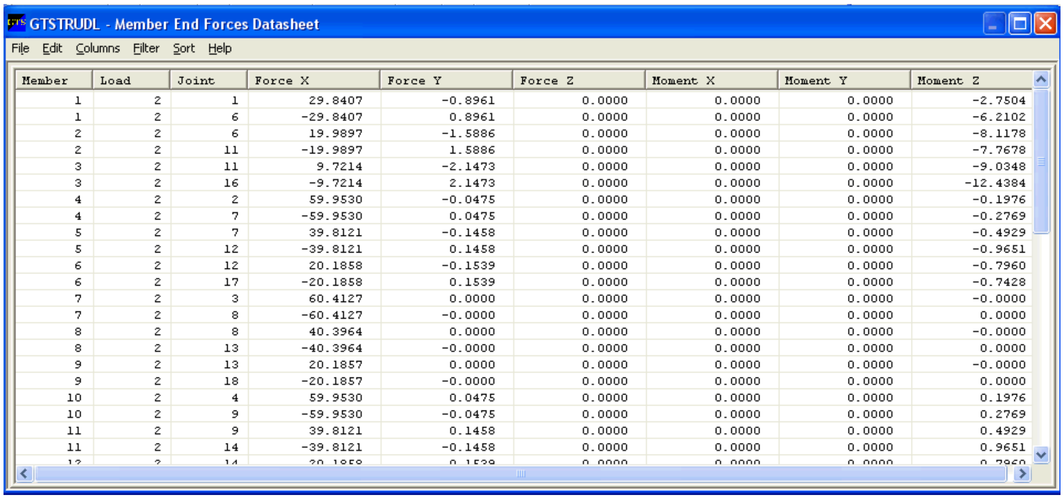
A new option has been added called List Summation of Forces to report the resultant forces and moments from a set of joints and members/elements at a joint or point. The force and moment resultants will be reported for all active loading conditions. The List Sum Forces dialog is shown below:



Datasheets for Joint Displacements, Member End Forces, and Member Section Forces are now available in Version 28 as shown below:

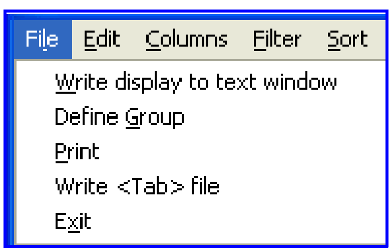


Previously, only the Joint displacement Datasheet was available in Version 27. An example of the new Datasheet for Member End Forces is shown below:



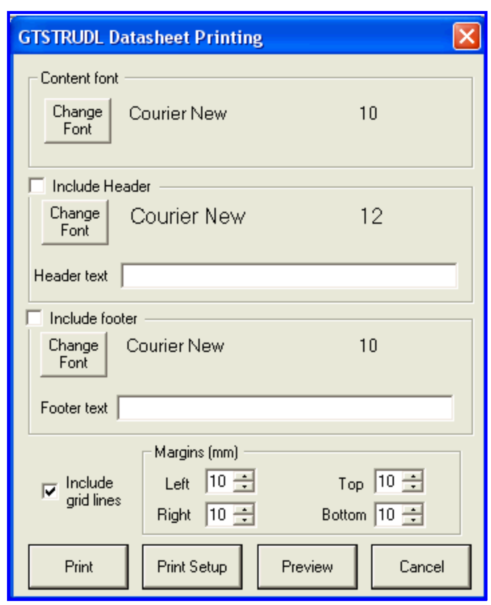
Member	Load	Joint	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z
1	2	1	29.8407	-0.8961	0.0000	0.0000	0.0000	-2.7504
1	2	6	-29.8407	0.8961	0.0000	0.0000	0.0000	-6.2102
2	2	6	19.9897	-1.5886	0.0000	0.0000	0.0000	-8.1178
2	2	11	-19.9897	1.5886	0.0000	0.0000	0.0000	-7.7678
3	2	11	9.7214	-2.1473	0.0000	0.0000	0.0000	-9.0348
3	2	16	-9.7214	2.1473	0.0000	0.0000	0.0000	-12.4384
4	2	2	59.9530	-0.0475	0.0000	0.0000	0.0000	-0.1976
4	2	7	-59.9530	0.0475	0.0000	0.0000	0.0000	-0.2769
5	2	7	39.8121	-0.1458	0.0000	0.0000	0.0000	-0.4929
5	2	12	-39.8121	0.1458	0.0000	0.0000	0.0000	-0.9651
6	2	12	20.1858	-0.1539	0.0000	0.0000	0.0000	-0.7960
6	2	17	-20.1858	0.1539	0.0000	0.0000	0.0000	-0.7428
7	2	3	60.4127	0.0000	0.0000	0.0000	0.0000	-0.0000
7	2	8	-60.4127	-0.0000	0.0000	0.0000	0.0000	0.0000
8	2	8	40.3964	0.0000	0.0000	0.0000	0.0000	-0.0000
8	2	13	-40.3964	-0.0000	0.0000	0.0000	0.0000	0.0000
9	2	13	20.1857	0.0000	0.0000	0.0000	0.0000	-0.0000
9	2	18	-20.1857	-0.0000	0.0000	0.0000	0.0000	0.0000
10	2	4	59.9530	0.0475	0.0000	0.0000	0.0000	0.1976
10	2	9	-59.9530	-0.0475	0.0000	0.0000	0.0000	0.2769
11	2	9	39.8121	0.1458	0.0000	0.0000	0.0000	0.4929
11	2	14	-39.8121	-0.1458	0.0000	0.0000	0.0000	0.9651

The Columns, Filter, and Sort pulldowns in the new Datasheets have the same functionality as in Version 27. However, the File pulldown has several new features as shown below:

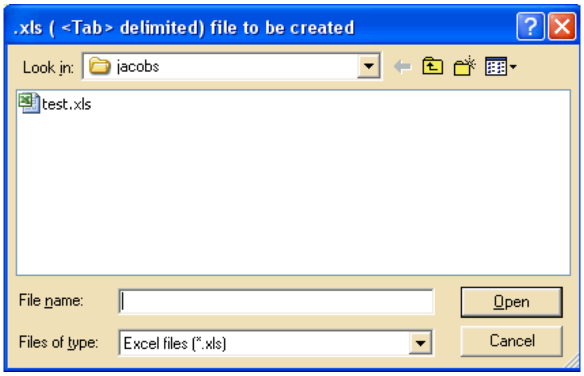


The Print option in the above pulldown allows you to print the datasheet using the dialog:

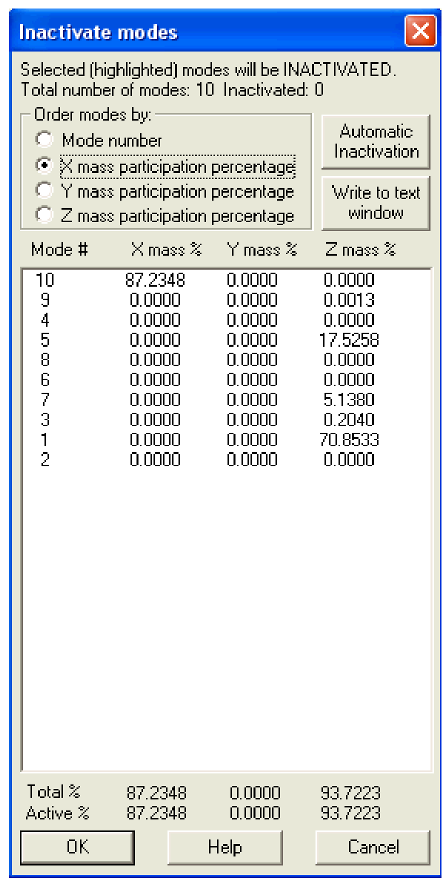
In the Print dialog, you may change the font, add a header and footer, and print the dialog with or without the gridlines in the Datasheet.



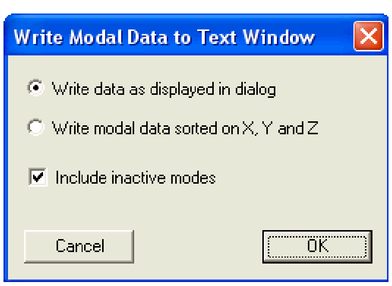
Another new feature added to the Results Datasheets is the ability to write a tab delimited file. This file may then be opened in Excel. You may include the column headings in the file in order to retain them in Excel. The following dialog appears when you select the Write file option under the File pulldown:



The Inactive Modes dialog under Dynamic Analysis Results has been enhanced to allow you to write the information in the dialog to the GTSTRUDL Output (text) Window. The revised Inactive Modes dialog with the new "Write to text window" option is shown:



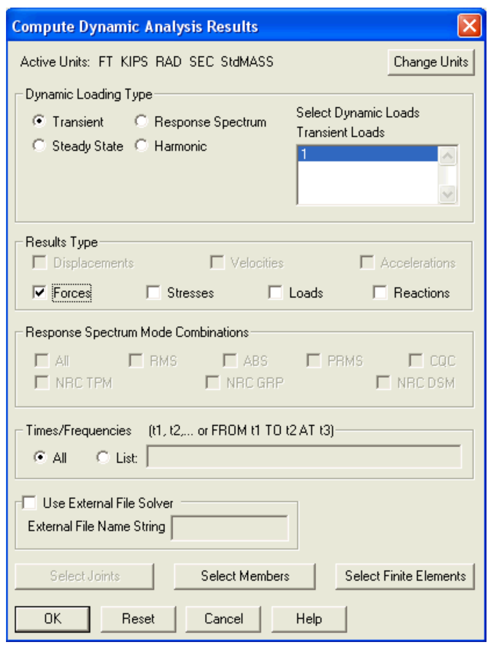
If the "Write to text window" option is selected, the following dialog will appear which allows you to either write the data as displayed, or write the modal data sorted on X, Y, and Z, and to include inactive modes.



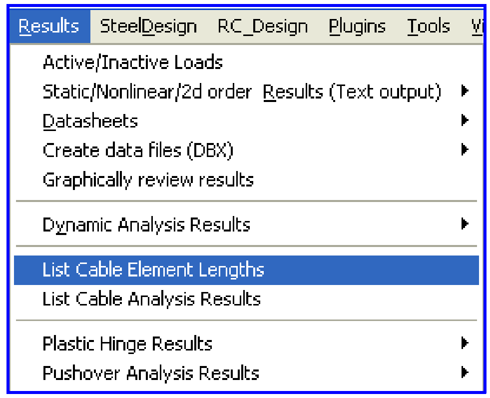
An example of the output which appears in the GTSTRUDL Output Window is shown below if the option to sort on X, Y, and Z is selected:

```
 -----------------------------------------------
| Output from the Inactivate Modes Dialog      |
 -----------------------------------------------
 Results for all modes

    Sorted on                          Sorted on                          Sorted on
    X mass %                           Y mass %                           Z mass %

   Mode          % Cum. sum          Mode          % Cum. sum          Mode          % Cum. sum
  ----- -------- --------           ----- -------- --------           ----- -------- --------
     10  87.2348  87.2348              10   0.0000   0.0000                1  70.8533  70.8533
      9   0.0000  87.2348               9   0.0000   0.0000                5  17.5258  88.3791
      4   0.0000  87.2348               8   0.0000   0.0000                7   5.1380  93.5171
      5   0.0000  87.2348               4   0.0000   0.0000                3   0.2040  93.7211
      8   0.0000  87.2348               5   0.0000   0.0000                9   0.0013  93.7223
      6   0.0000  87.2348               6   0.0000   0.0000                2   0.0000  93.7223
      7   0.0000  87.2348               7   0.0000   0.0000               10   0.0000  93.7223
      3   0.0000  87.2348               3   0.0000   0.0000                6   0.0000  93.7223
      1   0.0000  87.2348               1   0.0000   0.0000                8   0.0000  93.7223
      2   0.0000  87.2348               2   0.0000   0.0000                4   0.0000  93.7223

 Total%  87.2348                        0.0000                            93.7223
Active%  87.2348                        0.0000                            93.7223
```

The List and Compute dialogs under Dynamic Analysis results have also been modified. Separate buttons for the selection of Members and Elements now appear at the bottom of the dialogs where previously the same button was used for Members and Elements. The modified dialog is shown:



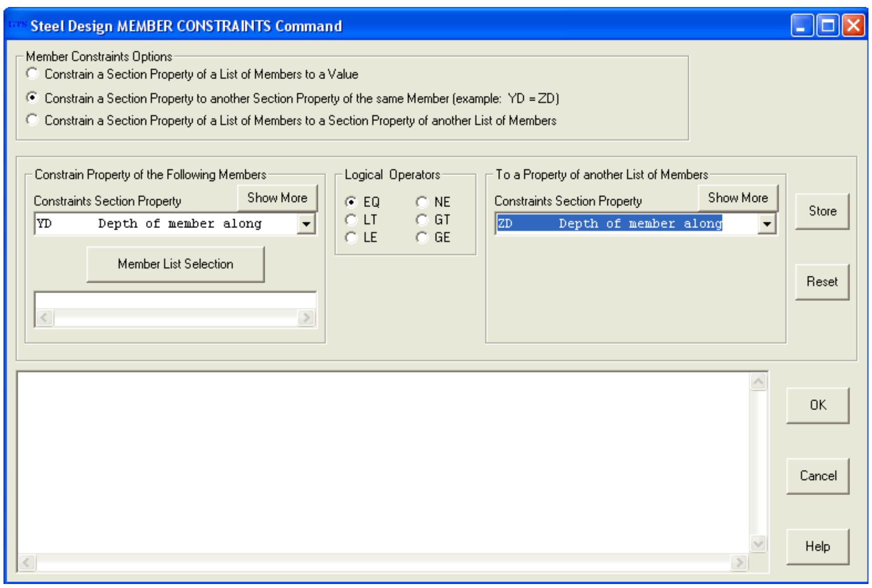
The Results pulldown now includes options to List Cable Element Lengths and to List Cable Analysis Results as shown:



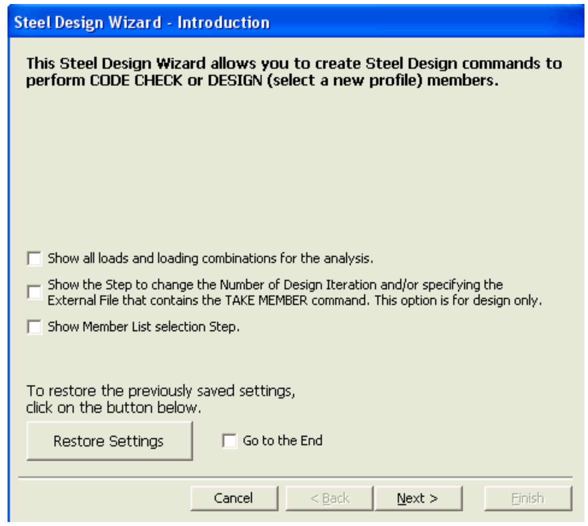
Steel Design Pulldown

New parameters have been added to the Parameters dialog for the LRFD3 code. More information on the LRFD3 parameters is available in Section 5.2.1. New LRFD3 tables have been added to the LRFD3 parameter dialog. Furthermore, the ASD9-E code, which is a prerelease feature described in Section 5.2.10, now appears in the Parameters dialog.

The Member Constraints dialog has a new option which allows a member's section property to be constrained to another section property of the same member (i.e. YD = ZD). The revised dialog is shown below:

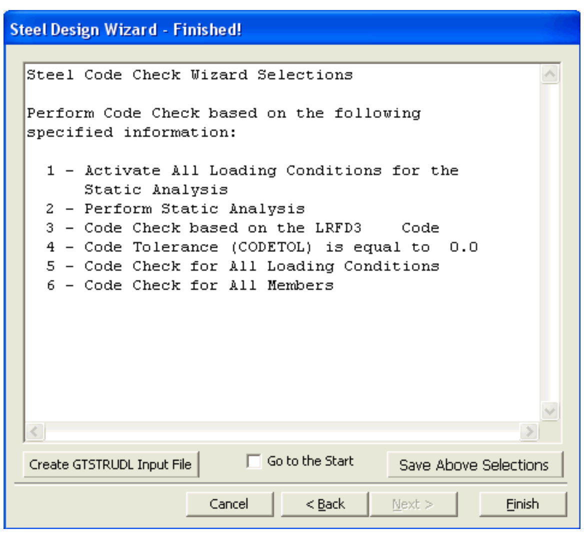


The Steel Design Wizard has been modified so that it no longer displays the Steps in the Wizard that do not usually change. An option has been added to the first dialog in the Wizard to show the Steps that are no longer being shown by default. An example of the first dialog in the Wizard is shown:



When the Load Combination button on the Design Wizard is used and new loading combinations have been added, the load dialogs are now updated to show the new loading combinations.

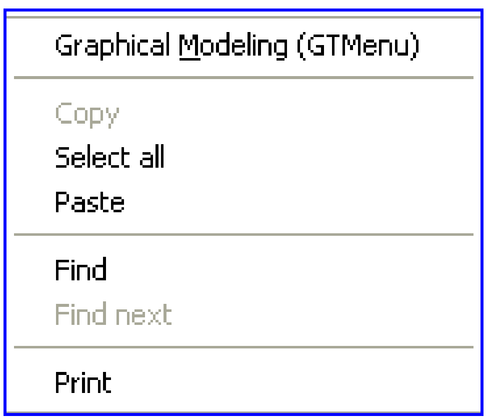
A new button has been added to the end of the Steel Design Wizard to create an input file (*.gti) and database file (*.gts). The new button to Create a GTSTRUDL Input File on the last dialog of the Wizard is shown below:



The input file contains the structural model including the new properties if a design has been performed as well as the steel design parameters and constraints and the analysis and code check commands.

A new option has been added to the Create Cross Section Property Database dialog to allow the display of results generated during the creation of the Property Database (profile table). An example of this new option is shown in the following dialog:

A new feature has been added to the GTSTRUDL Output Window. Right clicking the mouse in the GTSTRUDL Output Window will bring up the following pop-up:



As can be seen in the above pop-up, selections include entering GTMenu, Copy items to the clipboard if they have been selected with the cursor, Select all, Paste (this assumes that the information on the clipboard contains valid GTSTRUDL commands that can be pasted into the command line), a search feature using Find or Find next, and Printing.

2.3 Dynamics

1. The INERTIA OF JOINTS FROM LOAD command has been extended to include the conversion of translational force load components of both member and finite element loads to translational joint masses. Member temperature loads, member distortion loads, and finite element joint temperature loads are not converted.

2. Efficiency improvements have been made with regard to the assembly and use of the damping matrix for dynamic analyses. When damping data have been specified, the damping matrix is now assembled on demand and only when needed:

   a. The ASSEMBLE FOR DYNAMICS command, the PERFORM EIGENVALUE ANALYSIS command, and the DYNAMIC ANALYSIS EIGENVALUE command do not cause the damping matrix to be assembled.

   b. The PERFORM PHYSICAL ANALYSIS command, the DYNAMIC ANALYSIS PHYSICAL command, and the DYNAMIC ANALYSIS NONLINEAR command cause the damping matrix to be assembled if appropriate damping data have been specified.

   c. The COMPUTE MODAL DAMPING RATIOS command causes the damping matrix to be assembled for the computation of composite modal damping ratios if the appropriate damping data have been specified.

   d. The PERFORM MODE SUPERPOSITION ANALYSIS command and the DYNAMIC ANALYSIS MODE SUPERPOSITION command cause the damping matrix to be assembled for the computation of composite modal damping ratios if the appropriate damping data have been specified.

3. The assembly of the mass matrix for dynamic analysis is no longer automatically forced to be consistent when joint constraints are present in the model. The INERTIA OF JOINTS LUMPED command is now honored when joint constraints are present in the model.

4. The COMPUTE COMPOSITE DAMPING RATIOS command has been extended as shown below whereby the composite modal damping ratios are computed on the basis of the assumption that the damping ratios are directly proportional to total mass and stiffness.

$$\underline{\text{COM}}\text{PUTE } \underline{\text{MOD}}\text{AL } \underline{\text{DAMP}}\text{ING } (\underline{\text{RAT}}\text{IOS}) \left\{ \begin{array}{l} \rightarrow \underline{\text{AVE}}\text{RAGE} \\ \quad \underline{\text{PRO}}\text{PORTIONAL} \end{array} \right\} -$$

$$(\underline{\text{INC}}\text{LUDE } (\underline{\text{COU}}\text{PLING } \underline{\text{FRO}}\text{M}) \left\{ \begin{array}{l} \underline{\text{MOD}}\text{ES mlist} \\ \underline{\text{ALL}} \end{array} \right\})$$

5. Improvements have been made to the efficiency of memory allocation and management for the assembly of dynamic mass and composite modal damping matrices. Dynamic analysis jobs that contained rigid bodies and/or large numbers of specified damping ratios for joint inertias and were too large for available memory now execute to completion.

6. The UNITS command has been extended to include mass units as follows:

$$\underline{\text{UNI}}\text{TS} \left\{ \begin{array}{l} \underline{\text{STAN}}\text{DARD} \\ \underline{\text{STDM}}\text{ASS} \\ \underline{\text{GRAM}} \\ \underline{\text{KGM}} \\ \underline{\text{MTNM}} \end{array} \right\}$$

The mass unit STANDARD/STDMASS is the default and indicates that the active mass unit is taken as force-time^2/length, based on the active force, time, and length units. For example, if the active force and length units are specified as KIPS and FEET respectively, and the active time unit is SECONDS, then the active STANDARD/STDMASS mass unit is KIP-SECOND^2/FEET.

The default internal mass unit to which all mass data are converted when they are stored is lb-second^2/inch. Therefore, based on the previous example, the conversion factor from STANDARD active units of KIP-SECOND^2/FEET to internal units of lb-second^2/inch is (1000 lbs/kip)/(12 inches/foot) = 83.333333.

This implementation is designed so that when STANDARD/STDMASS is the active mass unit, mass data processing with respect to active units, including text output, remains unchanged from previous versions of GTSTRUDL.

See Section 2.1.2.6, Volume 1, GTSTRUDL Reference Manual for further information.

2.4 Finite Elements

1. The element name specified in the ELEMENT PROPERTIES command is no longer required to be upper case or to have single quotes around the name as shown in the examples below:

   Examples:

   ```
   Element Properties
       1 to 10 type sbhq thick 2.

   Element Properties type sbht6
       11 to 15  thick 2.5
       16 to 20  thick 3
   ```

2. Temperature gradient loading capabilities on the following elements have been brought to a release status:

Plate Bending elements	BPHT, BPHQ
Plate elements	SBHT, SBHQ, SBHQCSH, SBHT6, SBHQ6

   The finite element dictionary shown in Section 2.3.4 of Volume 3 of the Reference Manuals now indicates that temperature gradient loading is available for the elements listed above.

2.5 General

1. A new AREA LOAD command has been implemented to distribute loadings over defined regions to the surrounding members. The member loads are applied in the specified global direction X, Y or Z. A loaded area must be completely enclosed by members.

General form:

$$\underline{\text{ARE}}\text{A}\ \ \underline{\text{LOA}}\text{D}\ \ \text{'id'}\ \ \text{('title')}\ \ \underline{\text{DIR}}\text{ECTION} \left\{\begin{matrix}\underline{X}\\ \underline{Y}\\ \underline{Z}\end{matrix}\right\} (\underline{\text{PLA}}\text{NE}\ \ \underline{\text{TOL}}\ \ v_t)$$

where,

id	=	loading identifier, limited to 8 characters. If id is an integer, then the quotes (‘’) are not needed.
title	=	loading title, limited to 64 characters.
v_t	=	tolerance above and below the specified elevation.

Tabular repeat

$\underline{\text{ELE}}\text{VATION}$ v_e (region ($\underline{\text{EXC}}\text{EPT}$ region2)) $\underline{\text{VAL}}\text{UE}$ load_value dist_type
C
C
C
$\underline{\text{END}}$ $\underline{\text{ARE}}\text{A}$ $\underline{\text{LOA}}\text{D}$

where,

v_e	=	elevation along the specified axis.
region	=	all the structure at elevation if not given, otherwise

$$\underline{\text{LIM}}\text{ITS} \left\{\begin{matrix} \left\{\begin{matrix}\underline{X}\\ \underline{Y}\\ \underline{Z}\end{matrix}\right\} v_4\ v_5 \left\{\begin{matrix}\underline{X}\\ \underline{Y}\\ \underline{Z}\end{matrix}\right\} v_6\ v_7 \\ \underline{\text{JOI}}\text{NTS j\_list} \\ \underline{\text{MEM}}\text{BERS m\_list} \end{matrix}\right\}$$

v_4, v_5, v_6, v_7	=	max and min of an orthogonal "bounding box" that encloses the area. Only areas completely inside the "bounding box" will be loaded.
j_list	=	a list of joints that form a "bounding box" defined by the

minimum and maximum coordinates of the joints in j_list in the loading plane. Only areas completely inside the "bounding box" will be loaded.

m_list = a list of members that form the perimeter of the region. These members must form a contiguous line along the perimeter, but need not be in any order.

load_value = distributed load of force per length squared, in current units

dist_type =

$$\text{dist\_type} = \left\{ \begin{array}{l} \rightarrow \underline{\text{TWO}}\ \underline{\text{WAY}} \\ \\ \underline{\text{ONE}}\ \underline{\text{WAY}} \left\{ \begin{array}{l} \underline{X} \\ \underline{Y} \\ \underline{Z} \\ \\ \underline{X}\ v_8\ \underline{Z}\ v_9 \end{array} \right\} \end{array} \right\}$$

X, Y, Z = Global direction in which to distribute the one way loading. For example, if you select “DIRECTION Y” in the AREA LOAD command, you can select either X or Z as the direction to distribute the one way load.

v_8, v_9 = Ratio of one way load distribution. This is used when the desired one way distribution is not aligned with a global axis. X and Z correspond to the local, regional coordinate system. The local and global coordinate systems are identical for DIR Y. For DIR X, X corresponds to global Y and Z corresponds to global Z. For DIR Z, X corresponds to global X and Z corresponds to global Y.

region2 = same as region, except LIMITS are required, not optional.

You may specify as many ELEVATION commands as needed. Complete the AREA LOAD definition with the END AREA LOAD command.

The AREA LOAD command is used to apply a uniform distributed load to selected regions of the structure. Areas within the specified region are automatically determined, where an area is a part of the structure perpendicular to one of the global axes, and bounded by members on all sides. An independent load is created and the area load is distributed to the bounding members of an area as MEMBER LOAD components. If geometry or changing load values require, multiple ELEVATION commands can be used to describe the area load on a single floor. An area may be loaded by more than one ELEVATION command. All of the loading components created by the ELEVATION command between an AREA LOAD command and the following END AREA LOAD command are contained in the single independent load. Use the PRINT LOAD DATA command to review the values of the created loading components.

The AREA LOAD command is described in Section 2.1.11.3.7 of Volume 1 of the Reference Manual.

2.. The DEAD LOAD and SELF WEIGHT commands have been enhanced to include two and three-dimensional finite elements in the computation of member/element and joint dead loads. The INCLUDE FINITE ELEMENT option has been added to the SELF WEIGHT and DEAD LOAD commands as shown below:

$$\underline{\text{SEL}}\text{F}\ \underline{\text{WEI}}\text{GHT}\ (\underline{\text{LOA}}\text{DING})\begin{Bmatrix} i_1 \\ 'a_1' \end{Bmatrix}\ (\text{'title'})\ \underline{\text{DIR}}\text{ECTION}\begin{Bmatrix} \underline{X} \\ \underline{-X} \\ \underline{Y} \\ \underline{-Y} \\ \underline{Z} \\ \underline{-Z} \end{Bmatrix}\ -$$

$$([\underline{\text{FAC}}\text{TOR}]\ v_1)\ (\underline{\text{INC}}\text{LUDE}\ (\underline{\text{F}}\text{INITE})\ (\underline{\text{E}}\text{LEMENTS}))\ \text{clist}$$

$$\underline{\text{DEA}}\text{D}\ \underline{\text{LOA}}\text{DING}\begin{Bmatrix} i_1 \\ 'a_1' \end{Bmatrix}\ (\text{'title'})\ \underline{\text{DIR}}\text{ECTION}\begin{Bmatrix} \underline{X} \\ \underline{-X} \\ \underline{Y} \\ \underline{-Y} \\ \underline{Z} \\ \underline{-Z} \end{Bmatrix}\ -$$

$$([\underline{\text{FAC}}\text{TOR}]\ v_1)\ (\underline{\text{INC}}\text{LUDE}\ (\underline{\text{F}}\text{INITE})\ (\underline{\text{E}}\text{LEMENTS}))\ \text{clist}$$

If the INCLUDE FINITE ELEMENTS option is not specified in the above commands, the commands function as they did in earlier releases of GTSTRUDL - the self-weight of the finite elements will not be considered in the computation of the DEAD or SELF WEIGHT loadings. The revised commands are documented in Section 2.1.11.3 of Volume 1 of the Reference Manuals.

3. A new command has been implemented as a prerelease feature to calculate and report the summation of forces and moments at a specified point from a specified set of member and finite element nodal forces for all active static loading conditions. The new command is LIST SUM FORCES and is similar to, but more general than, the CALCULATE RESULTANT command described in Section 2.3.7.3, Volume 3 of the Reference Manual. The force summation is computed for all active loading conditions. The form of the command is shown below and is further described in Section 5.4.8:

$$\underline{\text{LIST}}\ \ \underline{\text{SUM}}\ \ \underline{\text{FOR}}\text{CES}\ \ (\underline{\text{TITL}}\text{E}\ 'a_T')\ (\underline{\text{ABO}}\text{UT} \left\{ \begin{array}{l} \underline{\text{JOI}}\text{NT} \left\{\begin{array}{l} i_c \\ 'a_c' \end{array}\right\} \\ [\underline{\text{X}}]\ v_X\ [\underline{\text{Y}}]\ v_Y\ [\underline{\text{Z}}]\ v_Z \end{array} \right\})\ -$$

$$(\underline{\text{ORI}}\text{ENTATION} \left\{\begin{array}{l} [\underline{\text{TH1}}]\ v_1\ [\underline{\text{TH2}}]\ v_2\ [\underline{\text{TH3}}]\ v_3 \\ \left\{\begin{array}{l}\underline{\text{MEM}}\text{BER} \\ \underline{\text{ELE}}\text{MENT}\end{array}\right\} \left\{\begin{array}{l} i_O \\ 'a_O' \end{array}\right\} \end{array}\right\})\ \ \underline{\text{JOI}}\text{NTS}\ list_j\ \ \underline{\text{MEM}}\text{BERS}\ list_m$$

4. New commands have been added to delete joints or members that do not have data associated with them. The new commands are as follows:

$$\underline{\text{DEL}}\text{ETE}\ \underline{\text{JOI}}\text{NTS}\ \underline{\text{WIT}}\text{HOUT}\ (\underline{\text{JOI}}\text{NT})\ \underline{\text{COO}}\text{RDINATES}$$

$$\underline{\text{DEL}}\text{ETE}\ \underline{\text{MEM}}\text{BERS}\ \underline{\text{WIT}}\text{HOUT}\ (\underline{\text{MEM}}\text{BER})\ \underline{\text{INC}}\text{IDENCES}$$

$$\underline{\text{DEL}}\text{ETE}\ (\underline{\text{FIN}}\text{ITE})\ \underline{\text{ELE}}\text{MENTS}\ \underline{\text{WIT}}\text{HOUT}\ (\underline{\text{ELE}}\text{MENT})\ \underline{\text{INC}}\text{IDENCES}$$

Joints, members, or elements can be created when a user accidently specifies joint or member names in commands such as PRINT or LIST. Now, you may use the above commands to delete these joints, members or elements.

See Section 5.4.7 for more information on these commands.

5. Elastic member end connections may now be deleted using the DELETIONS mode. In previous versions of GTSTRUDL, elastic member end connections could not be deleted. Section 2.1.8.2 of Volume 1 of the Reference Manual has been revised to indicate the functioning of the DELETIONS mode as it pertains to elastic member end connections.

6. The CALCULATE SOIL SPRING VALUES command has been brought to release status. Furthermore, the command has been extended since its introduction in Version 27 on a prerelease status to include the ability to create Compression Only springs and also In-plane springs. The command is shown below:

$$\underline{\text{CALC}}\text{ULATE } (\underline{\text{SOIL}})\ \underline{\text{SPRI}}\text{NG } (\underline{\text{VAL}}\text{UES}) \ (\underline{\text{COM}}\text{PRESSION } (\underline{\text{ONL}}\text{Y})) -$$

$$\underline{\text{DIR}}\text{ECTION} \begin{Bmatrix} \underline{\text{X}} \\ \underline{\text{Y}} \\ \underline{\text{Z}} \end{Bmatrix} (v_1)((\underline{\text{PLA}}\text{NE})\ \underline{\text{TOL}}\text{ERANCE } v_2) -$$

$$((\underline{\text{PLA}}\text{NE})\ \underline{\text{ANG}}\text{LE } (\underline{\text{TOL}}\text{ERANCE}) \ v_3) \ \text{value\_sets} -$$

$$(\underline{\text{APP}}\text{LY}) \ (\underline{\text{SHO}}\text{W } (\underline{\text{CAL}}\text{CS})) \ (\underline{\text{INP}}\text{LANE } (\underline{\text{RATI}}\text{O}) \ v_6)$$

The command is documented in Section 2.1.12.13 of Volume 1 of the GTSTRUDL Reference Manuals.

7. The SUMMARY option for LIST SECTION FORCES has a new option to include the output of corresponding forces. For example, for the section location and loading that produces the maximum value of MZ, the command will now also output the associated values of FX and FY. The modified command is documented in Section 2.1.14.6 of Volume 1 of the GTSTRUDL Reference Manual.

An example illustrating the modified command is shown below:

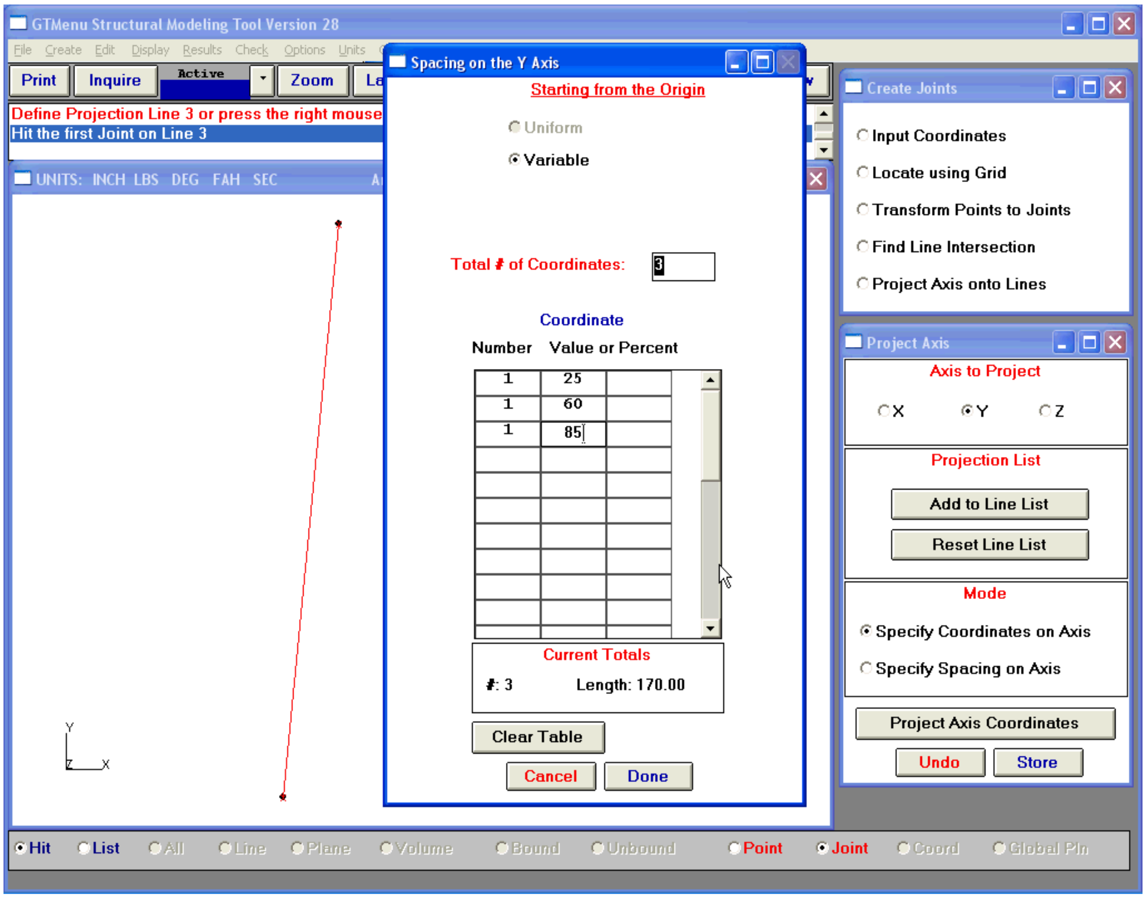
LIST SECTION FORCES SUMMARY AND CORRESPONDING FORCES MEMBER 27

The following is the output produced by the above command:

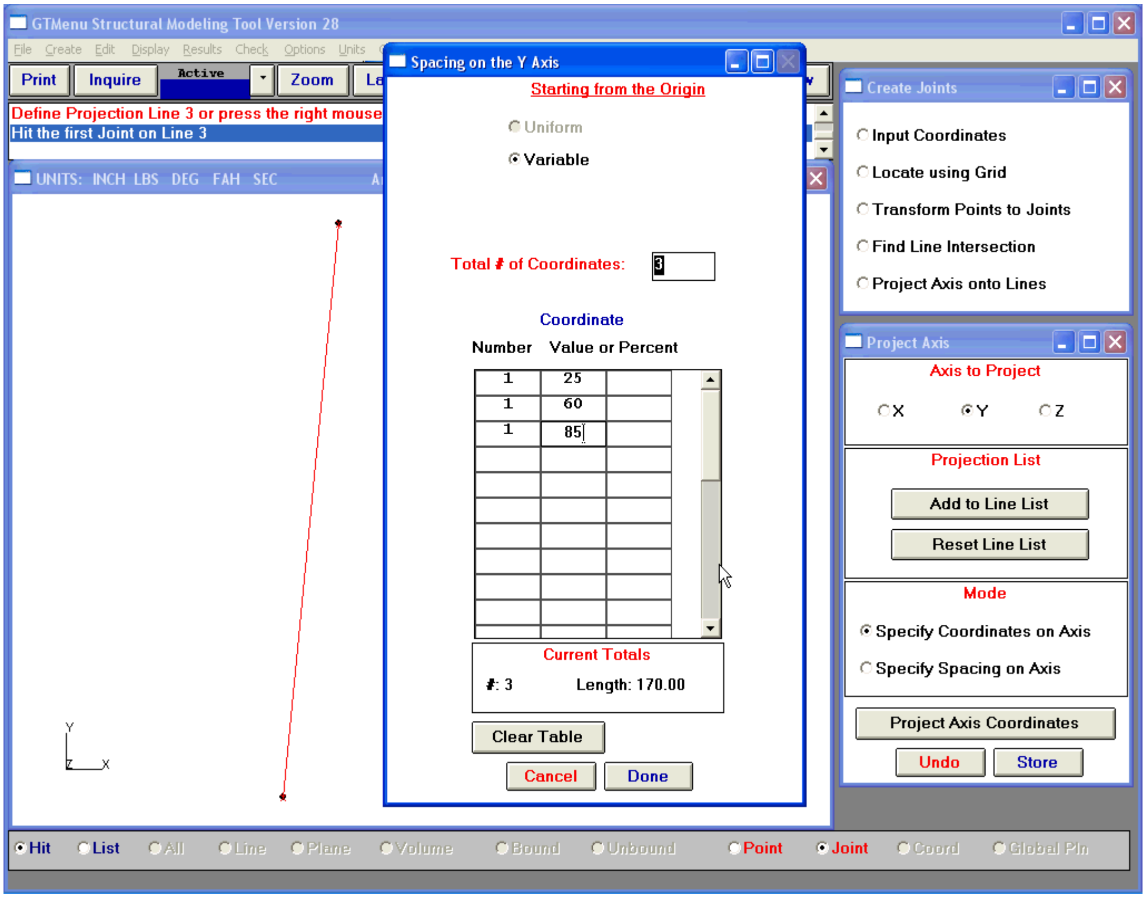
```
----------------------------------------------------------------------------------------------
Max/min Section Forces for member 27       , locations are fractional.
        Value        Load     Location          Value        Load      Location
        -------------  --------   --------          -------------  --------         --------
   Max FX:      -0.5         4    1.0000      Min FX:        -2.          2        1.0000
   Max FY:       10.         2    1.0000      Min FY:      -0.1E+02  2       0.0000
   Max FZ:      0.0E+00  4    1.0000      Min FZ:        0.0E+00  4       1.0000
   Max MX:     0.0E+00  4    1.0000      Min MX:        0.0E+00  4       1.0000
   Max MY:     0.0E+00  4    1.0000      Min MY:        0.0E+00  4       1.0000
   Max MZ:      3.           4    0.0000      Min MZ:      -0.2E+02  2       0.0000

Corresponding Forces for FX FY FZ MX MY MZ
   Max FX        -0.5040967     0.6305282     0.0000000E+00  0.0000000E+00  0.0000000E+00  -3.548704
   Min FX        -2.147321      9.721370      0.0000000E+00  0.0000000E+00  0.0000000E+00  -12.43836
   Max FY        -2.147321      9.721370      0.0000000E+00  0.0000000E+00  0.0000000E+00  -12.43836
   Min FY        -2.147321     -10.27863      0.0000000E+00  0.0000000E+00  0.0000000E+00  -15.22468
   Max FZ        -0.5040967     0.6305282     0.0000000E+00  0.0000000E+00  0.0000000E+00  -3.548704
   Min FZ        -0.5040967     0.6305282     0.0000000E+00  0.0000000E+00  0.0000000E+00  -3.548704
   Max MX        -0.5040967     0.6305282     0.0000000E+00  0.0000000E+00  0.0000000E+00  -3.548704
   Min MX        -0.5040967     0.6305282     0.0000000E+00  0.0000000E+00  0.0000000E+00  -3.548704
   Max MY        -0.5040967     0.6305282     0.0000000E+00  0.0000000E+00  0.0000000E+00  -3.548704
   Min MY        -0.5040967     0.6305282     0.0000000E+00  0.0000000E+00  0.0000000E+00  -3.548704
   Max MZ        -0.5040967     0.6305282     0.0000000E+00  0.0000000E+00  0.0000000E+00   2.756577
   Min MZ        -2.147321     -10.27863      0.0000000E+00  0.0000000E+00  0.0000000E+00  -15.22468
```

2.6 GTMenu

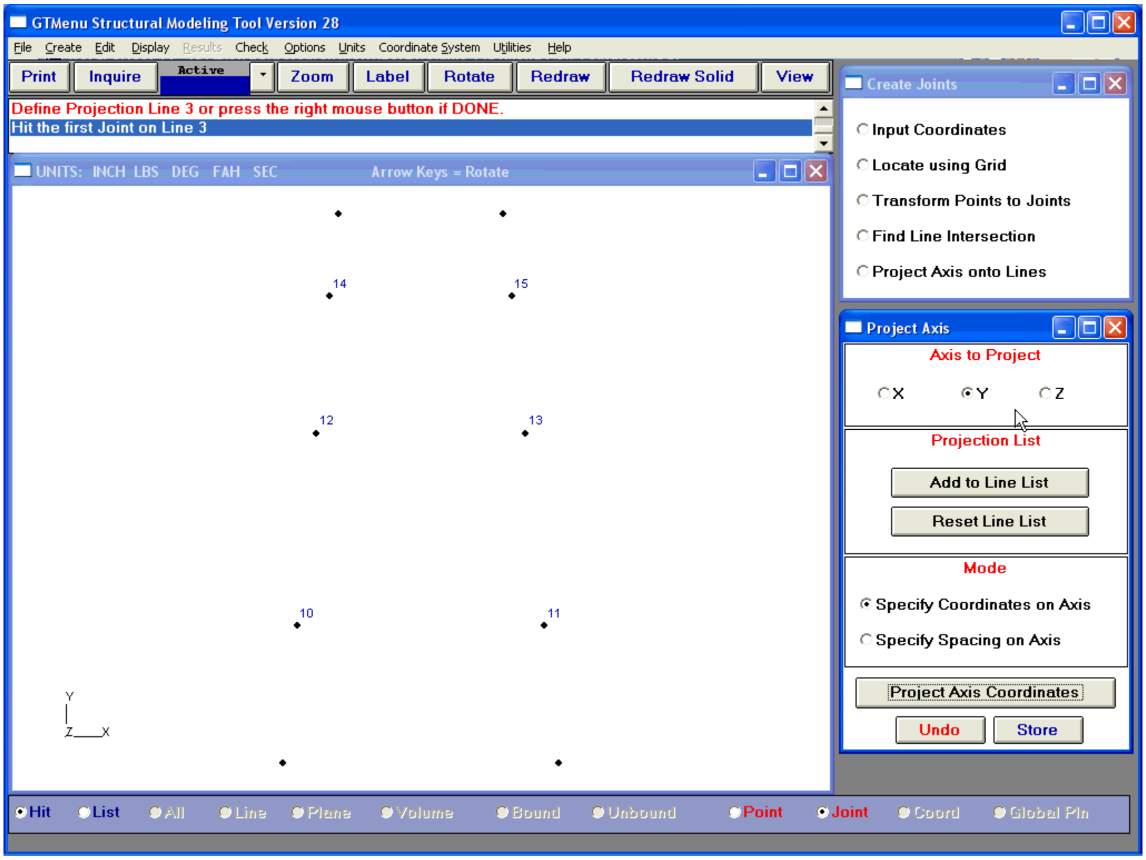
1. A new option called "Project Axis onto Lines" has been added to Create Joints dialog as shown below:



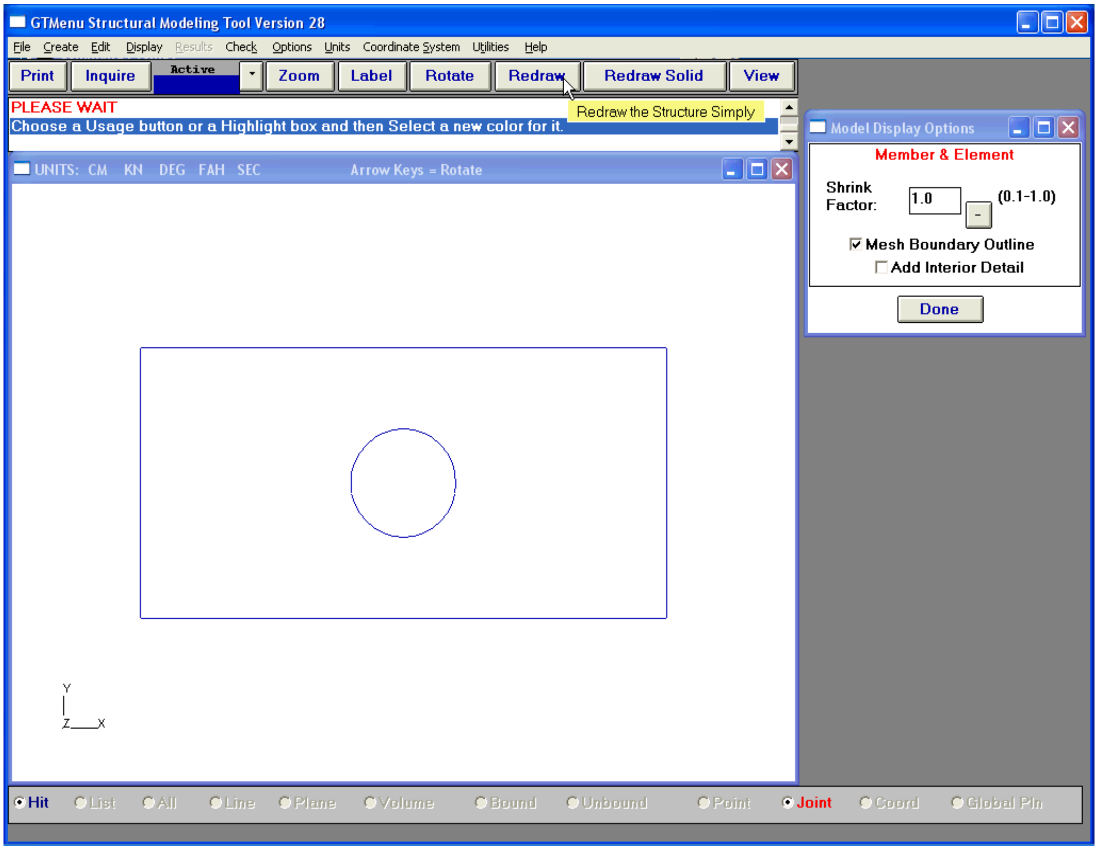
This Create Joints option allows you to specify straight lines between joints or points in your model and then create joints on those lines by projecting coordinates or spacing along a global axis onto those lines. An example of specifying the coordinates along the Y axis is shown below:



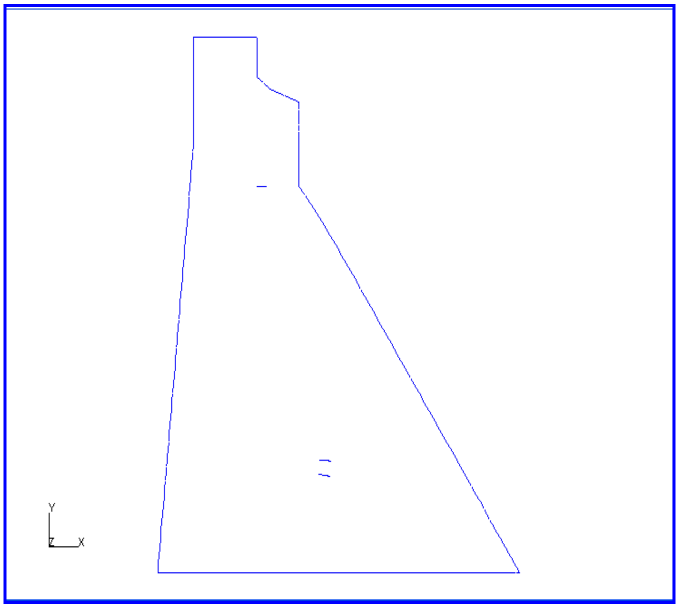
The joints created by projecting the coordinates on the Y-Axis onto the lines are shown below:



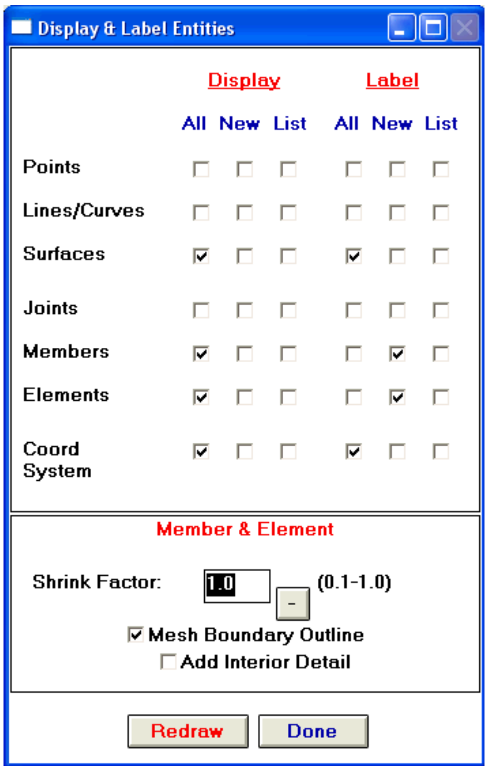
2. A new option has been added to display a finite element mesh boundary outline. This will display edges which have only one element incident upon them. The mesh boundary outline option is useful for displaying modeling with interior boundaries such as a plate with a hole. An example is shown below:

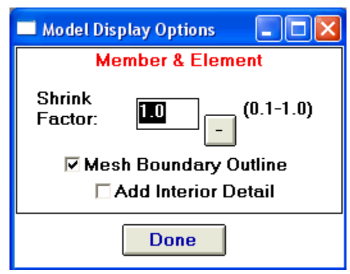


This option is also useful for detecting portions of the model which are not connected correctly. For instance, if an interior edge appears where there should not be one then you have not connected your model correctly. An example of a model with interior edges shown indicating that regions of the model are not connected correctly is shown:



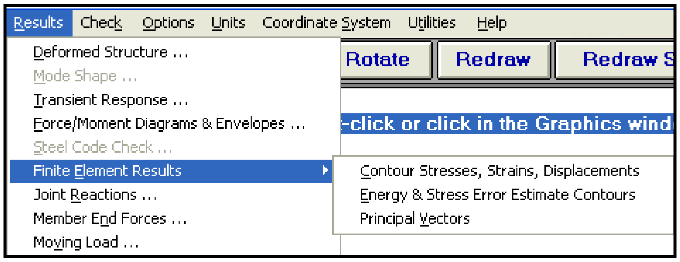
3. The Mesh Boundary Outline option is available by selecting Options from the Menu Bar and then selecting Model Options. The Mesh Boundary option is also available by selecting the Label button from the Button Bar and selecting Display and Label Settings. Examples of these two dialogs are shown below:





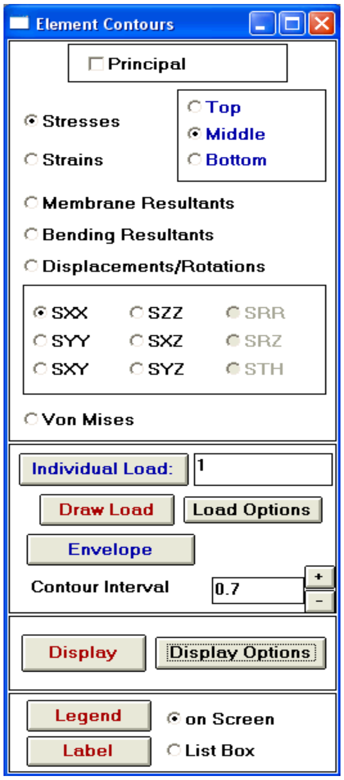
The Add Interior Detail option shown in the above dialogs is useful for meshes which contain three dimensional solids. This option will display edges where there are three or five or more elements connected. Also note in the display Label Settings dialog that the Shrink option is now available under this dialog as well as the Display Model dialog.

4. The Results pulldown has been enhanced and revised as shown below:

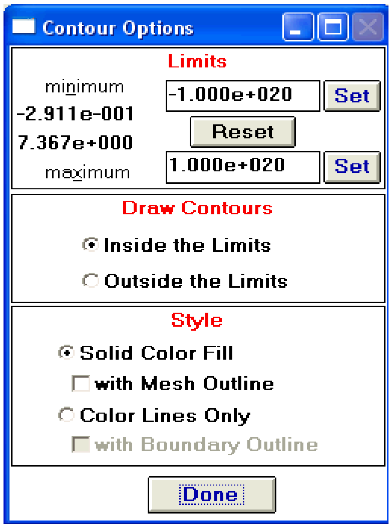


Finite Element Contours option has been renamed to Finite Element Results since the finite element results now includes Principal Vectors which are described later in this section. Also, a new option has been added to label Member End Forces which is also later described. Furthermore, new options have been added to the contouring of stresses, strains, and displacements.

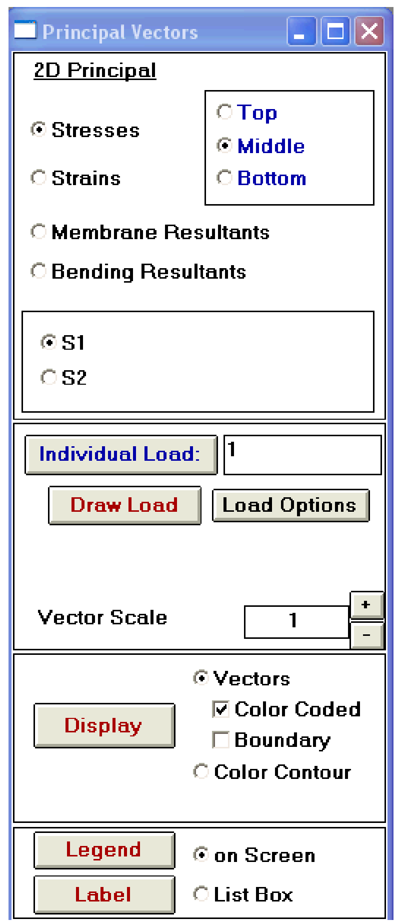
5. A new Display Options button has been added to the Element Contours dialog in GTMenu. The Contour Options dialog will allow you to set Minimum and Maximum values for contouring. Subsequent contouring will then be performed approximately within or outside of these limits depending which option is selected. You may also choose if you want the element (mesh) boundaries drawn and if you want solid (color-filled) or line contours drawn. An example of the new Element Contours dialog is shown below:



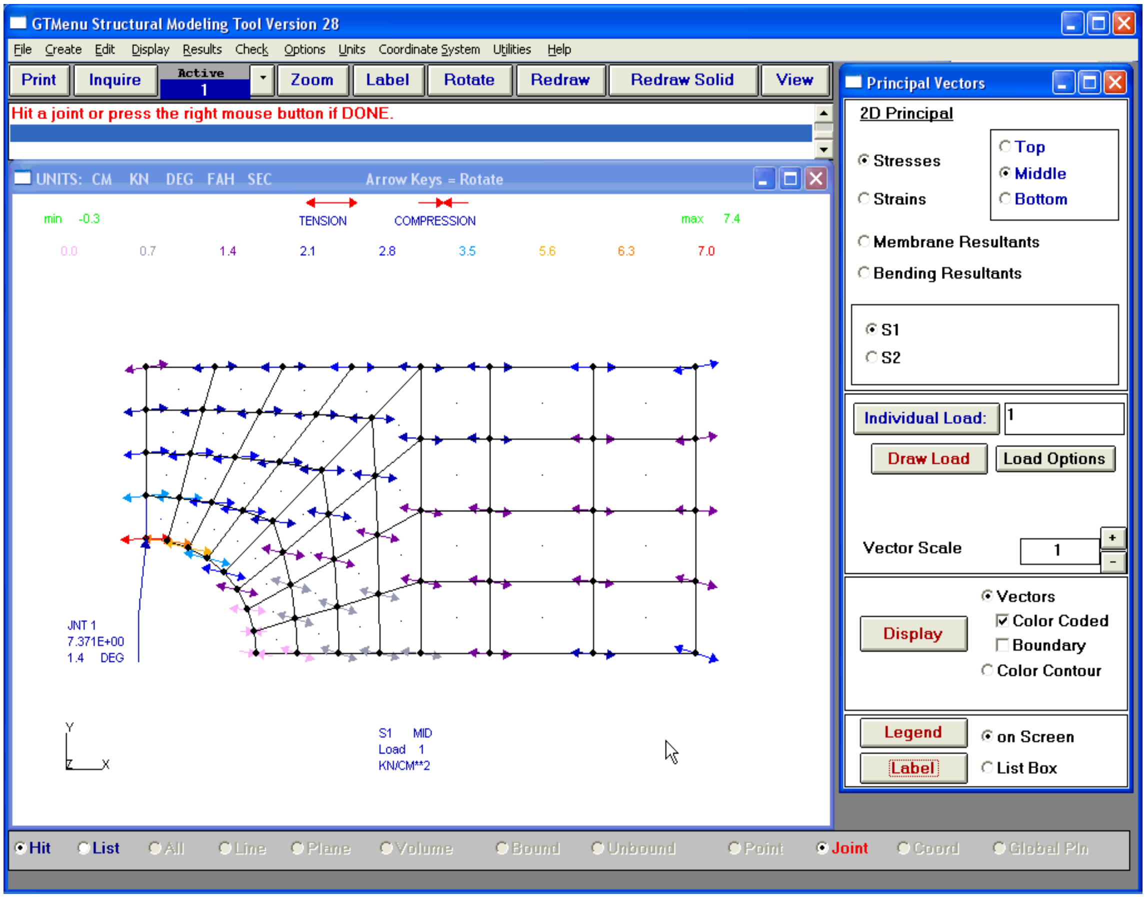
The new Display Contour Options dialog is shown below:



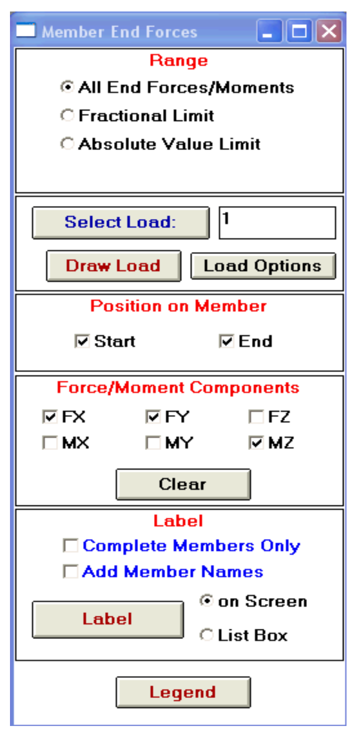
6. GTMenu can now draw principal stress, strain, and resultant vectors for two-dimensional finite elements. The vectors are drawn as scaled color coded vectors by default. The vectors are oriented in the direction of the principal component being drawn. The magnitude and angle of the vectors may also be labeled. The new Principal Vectors dialog is shown:



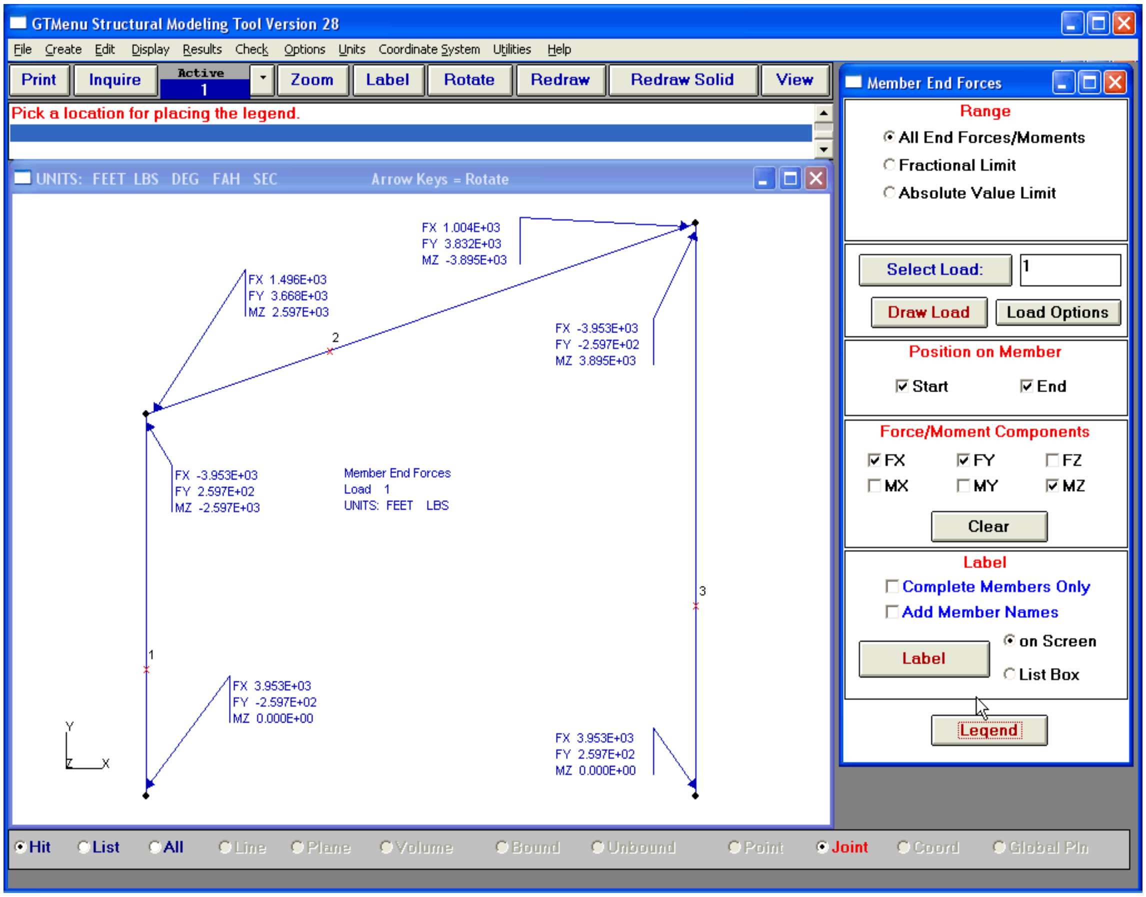
An example of a display illustrating the principal vectors and labeling is shown below:



7. A new Results option has been added to label Member End Forces. The Member End Forces may now be displayed on-screen, or they may be sent to a list box. The forces may be labeled at the start and/or end of the members. You may also choose the force and moment components to label. The new Member End Forces dialog is shown:

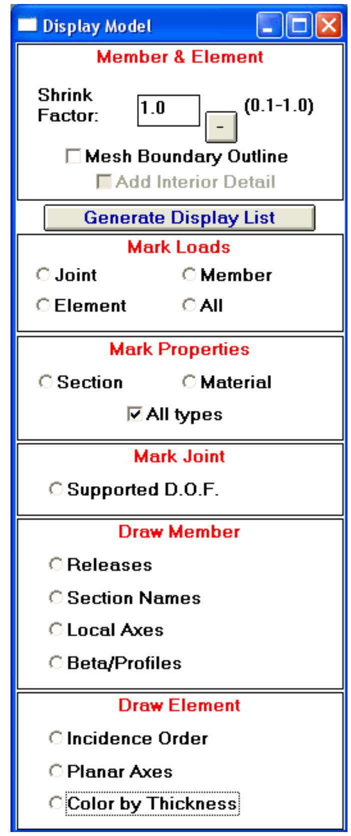


An example of labeling Member End Forces is shown in the figure below:

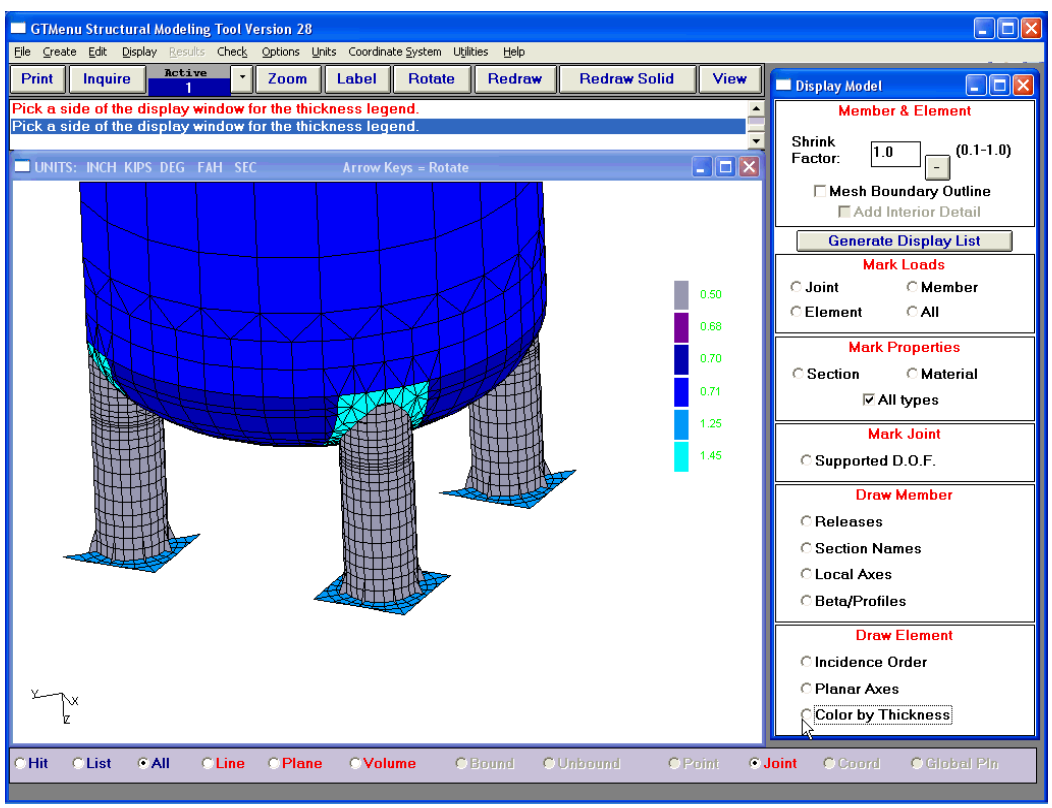


8. An option to Draw Supports has been added to the Create Supports dialog as shown below:

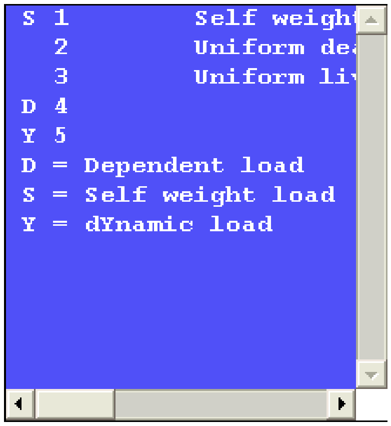
9. A Color by Thickness option has been added for 2D finite element meshes under the Display Model dialog. This feature is useful for checking models which have different thicknesses assigned to finite elements. The modified Display Model is shown with the new Color by Thickness option shown at the bottom of the dialog:



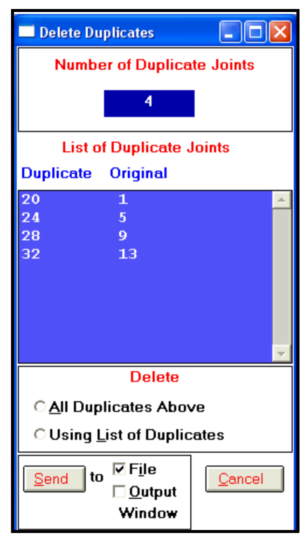
An example of the display created using the Color by Thickness option is shown below:



10. All load names that are currently used are now listed and classified to help in choosing the name for a new load as shown in the figure below:



11. When a duplicate joint, member, or element is found, the original name is now displayed in addition to the duplicate name to aid in the location of the duplicates and in deciding which duplicates to remove. An example of the output from Check Duplicate Joints is shown:

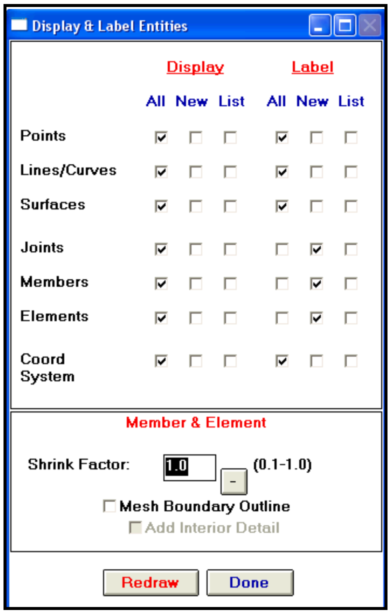


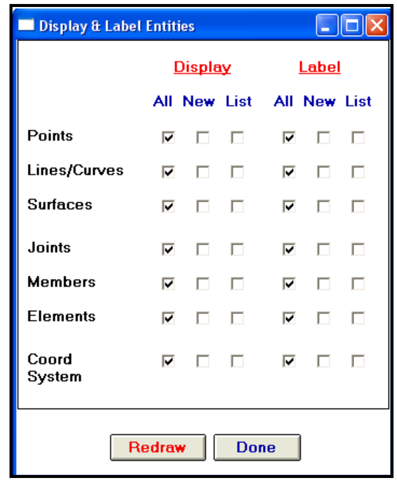
In addition you may now send the results of the Check Duplicates to the GTSTRUDL Output Window or to a file using the options at the bottom of the Delete Duplicates dialog.

12. Label and Display settings (Label button on the Button Bar, Display and Label Settings option) are now saved between GTMenu executions within a GTSTRUDL session. Thus, if you end GTMenu and then return, the label and display settings are retained.

The default Label settings have been changed for Joints, Members, and Elements. In Version 28, the default Label settings have been set to “New”. The “New” option is for a joint, member, or element that has been newly created but not stored. Once a joint, member, or element is stored and the Graphics Windows redrawn, the labels will no longer be shown.

The default Label settings for Versions 28 and 27 are shown below:





Version 28

Version 27

13. The GTMenu input file generator can now be accessed from command mode, by the following command:

GTMenu GENERATE INPUT FILE

General form:

GTMenu GENERATE INPUT FILE ('filename')

where

'filename' = 1 to 256 character filename for the input file. If none is specified 'GTMenuInputFile.gti' is assumed.

When writing a GTSTRUDL input file from GTMenu, if Points exist, the line "GTMenu POINT COORDINATES" is now output followed by names and coordinate values, analogous to "JOINT COORDINATES". In addition, "GTMenu Line Incidences" are also output for Construction Lines as shown on the next page.

From command mode, the Point and Line input is read and appended to any currently existing Points and Lines, ignoring Points and Lines having duplicate IDs.

```
UNITS  CM    KN    DEG   FAH
GTMenu POINT COORDINATES
   'P1'           5.0000000E+00  0.0000000E+00  0.0000000E+00
   'P2'           1.0000000E+01  0.0000000E+00  0.0000000E+00
   'P3'           0.0000000E+00  5.0000000E+00  0.0000000E+00
   'P4'           5.0000000E+00  5.0000000E+00  0.0000000E+00
   'P5'           2.0000000E+00  0.0000000E+00  0.0000000E+00
   'P6'           1.9615706E+00  3.9018065E-01  0.0000000E+00
   'P7'           1.8477591E+00  7.6536691E-01  0.0000000E+00
   'P8'           1.6629393E+00  1.1111405E+00  0.0000000E+00
   'P9'           1.4142135E+00  1.4142135E+00  0.0000000E+00
   'P10'          1.1111405E+00  1.6629394E+00  0.0000000E+00
   'P11'          7.6536685E-01  1.8477591E+00  0.0000000E+00
   'P12'          3.9018074E-01  1.9615705E+00  0.0000000E+00
   'P13'          1.5099580E-07  2.0000000E+00  0.0000000E+00
$$
GTMenu LINE INCIDENCES
   'C1'      'P13'     'P3'
   'C2'      'P3'      'P4'
   'C3'      'P4'      'P1'
   'C4'      'P1'      'P5'
   'C5'      'P1'      'P2'
   'C6'      'P9'      'P4'
   'C7'      'P5'      'P6'      'P7'      'P8'      'P9'
   'C8'      'P9'      'P10'     'P11'     'P12'     'P13'
$$
```

More information on the GTMenu Point Coordinate and Line Incidence commands may be found in Section 5.4.9.

14. When GENERATING an INPUT FILE from GTMenu.

    MATERIAL STEEL
    MATERIAL ALUMINUM
    and MATERIAL CONCRETE

    are now used when applicable instead of listing CONSTANTS.

15. Excessive comment lines (lines starting with $$) have been removed from the input file. Other comment lines which make the input file more readable have been changed into blank lines.

16. When GTMenu Generates an Input File, JOINT COORDINATES and MEMBER INCIDENCES now generate only one initial spacing comment line, and it is a blank line instead of a '$$'. In addition, the MEMBER INCIDENCES command is no longer preceded by a UNITS specification.

17. In the Input File generated by GTMenu, adjacent joints with the same spring constants are now grouped together under the JOINT RELEASES.

18. In the Input File generated by GTMenu, ELEMENT INCIDENCES are now written with the element name (ID) up to 8 incidences on a single line.

19. Interspersed comments ($$) have been removed from the ELEMENT INCIDENCES and ELEMENT PROPERTIES.

20. When generating an Input File, a Beta of 0.0 is not listed in the file since that is the default value.

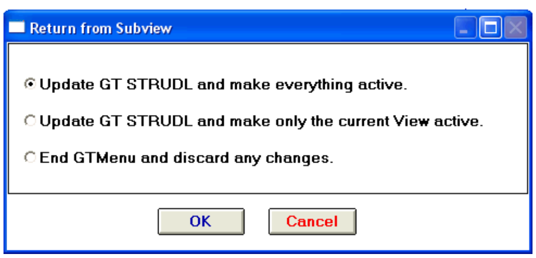
21. When generating an Input File, if material constants have not been specified, the Input Generator inserts a comment warning that material constants are missing.

22. The New View Name dialog has been moved so it does not cover part of the structure when creating new views.

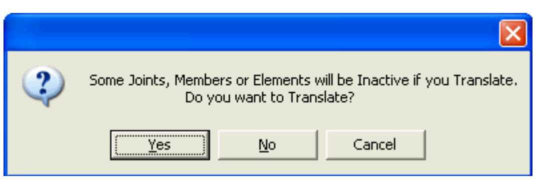
23. The contour legend is redrawn when a contour is refreshed.

24. The prompt asking if the user would like to update the VIEW database when leaving the View Menu has been removed.

25. If you “End GTMenu” while in a view other than Views 1 to 4, the following dialog will now appear:



This dialog makes it easier to return to the GTSTRUDL Output Window with your model changes and the entire structure active while in a Subview (all views other than Views 1 to 4). The above display replaces the Version 27 dialog shown below:



26. When Creating Members or Elements Using a Grid, a larger dot is now drawn for a grid point to make the grid points more visible.

27. New graphics Hotkeys have been added. The “LJ”, “LM”, and “LE” Hotkeys will label joints, members, or elements. The “LM” and “LE” Hotkeys will both label members and elements. A dialog will open which will allow you to identify the joints or members and elements to label with the default set to ALL. The new Hotkey, “LA”, will label all joints,

members, and elements.

28. Four other new Hotkeys are the “A”, “W”, “<“, and “>” keys. The “A” Hotkey will redraw the Graphics Window. The “W” Hotkey will initiate windowing while the “<“ and “>” Hotkeys will rotate the graphics display about the screen z-axis (the axis normal to the screen) counterclockwise and clockwise respectively.

    The HotKey HELP documentation has been updated to describe these new Hotkeys. To access the Hotkey Help documentation, select Help from the Menu Bar and then select Hotkey Help to see all the available Hotkeys.

29. A list in GTMenu, which may be Sent to a File or the GTSTRUDL Output (Command Window), is now preceded by a line describing how the list was produced.

30. Line/curve labels (U, V, W, etc.) have been repositioned to make them more readable.

31. Twenty-one sections are now used to compute a member diagram/envelope display regardless of Section specifications while 11 sections are used to draw the deformed shape between joints.

32. The boundary nodes of a Superelement can be listed in GTMenu in an INQUIRE box by specifying the name of the Superelement in list mode.

33. A preview line pointing to the nearest section location has been added to the box which is used to position displacement output on a deformed-between-joints display.

34. Table section longnames are displayed in Display Model and are listed with an INQUIRE.

35. The Material, Prismatic and Pipe Property Group dialogs have been revised. The Material and Prismatic Property Group dialogs now have an option to bring up the calculator. All of the Property Group dialogs (Pipe, Prismatic and Table) will now show the longname profile description.

36. A scroll bar has been added to allow viewing long load titles and dependent load descriptions.

37. Previously, when using the GTMenu feature “Print Preview and Edit”, the user was required to changed the name of the automatically generated Scope Editor file (cap00x.ssc) with the “Save As” selection, or the file would be deleted and any changes lost.

    Now, if the automatically generated file is “Saved” without a name change, it will no longer be deleted. Also, the default file name has been changed to “Temp00x.SSC” to further emphasize the temporary nature of the file.

38. When performing a transient animation in GTMenu, if there is only one dynamic load, it is now selected automatically.

39. The warning message indicating that section displacements are not available for truss members will no longer be given when displaying a Deformed Structure in GTMenu using the “Deform between joints” option.

40. When a joint is selected, it is now marked with a capital "X" instead of a lower-case "x" in order to make the selection more visible.

2.7 GTTable

1. GTTABLE commands are now available within GTSTRUDL. You no longer need to run a separate GTTABLE process in order to create Tables. The output from the commands will no longer produce overflows of the output fields (*********** printed) when executed inside of GTSTRUDL.

2.8 Nonlinear

1. Pushover analysis now contains a new switchable displacement control option. When displacement control is enabled and a conventional pushover analysis load increment encounters convergence difficulties, typically indicating a collapse condition, the pushover analysis procedure switches to displacement control, whereby selectable joint displacement degrees of freedom are monitored and controlled. The displacement control analysis procedure stops when the maximum of the selectable control displacements reaches a specified magnitude.

2. The primary steel stress-strain model, due to Balan, Popov, et.al., used in plastic hinge

analysis has been replace by the simpler, more widely known model described by Priestley, Seible, and Calvi in the text Seismic Design and Retrofit of Bridges (John Wiley & Sons, 1996).

3. The element stress output from the LIST CABLE ANALYSIS RESULTS and the LIST STRESSES command has been expanded for the IPCABLE element to include the cable axial force (FA) in addition to the cable axial stress. An example of the new output from the LIST STRESSES command is shown below. The new output from the LIST CABLE ANALYSIS RESULTS is similar.

```
{   60} > LIST STRESSES ELEMENTS 1 2 3

   ***************************
   *RESULTS OF LATEST ANALYSES*
   ***************************

    PROBLEM - CA051     TITLE - ST4796 -- Implement the LIST CABLE LENGTH command

    ACTIVE UNITS  M    N    RAD  DEGF SEC
=====================================================================================
  LOADING - 1             w
=====================================================================================

   ELEMENT STRESSES

 ELEMENT

 1        NODE  1                 Sxx     1166058.       Force       75.22939
          NODE  3                 Sxx     993526.1       Force       64.09834
          NODE  2                 Sxx     1079792.       Force       69.66386
 2        NODE  3                 Sxx     993650.3       Force       64.10635
          NODE  5                 Sxx     864930.1       Force       55.80183
          NODE  4                 Sxx     929290.2       Force       59.95409
 3        NODE  5                 Sxx     864983.8       Force       55.80530
          NODE  7                 Sxx     776710.9       Force       50.11028
          NODE  6                 Sxx     820847.3       Force       52.95779
 4        NODE  7                 Sxx     776637.4       Force       50.10554
          NODE  9                 Sxx     724438.7       Force       46.73789
```

4. The LIST CABLE LENGTHS command has been added for the output of the computed unstressed length of IPCABLE elements. The unstressed length includes the effect of the latest cable prestress analysis. The results also include the sum total of the lengths of the elements specified in the command list. The following is an example of the LIST CABLE LENGTHS command and the resulting output:

```
{   58} > LIST CABLE LENGTH ELEMENTS 4 TO 7
```

```
******************************
* RESULTS OF LATEST ANALYSIS *
******************************



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


Cable Element Unstressed Lengths
================================

Element                        Length
-------                      ----------

4                              2.15029
5                              2.02449
6                              2.02433
7                              2.15063
                             ==========

Total Length =                 8.34974
```

5. Nonlinear dynamic analysis now supports the assembly of proportional and non-proportional stiffness and mass damping matrices from all sources that provide for the specification of stiffness and mass damping factors, including the CONSTANTS, JOINT RELEASES, and INERTIA OF JOINTS WEIGHT/MASS commands. In previous versions, the damping factors from the CONSTANTS command were not supported.

6. Bandwidth reduction is now executed automatically (by default) for nonlinear dynamic analysis.

2.9 Offshore

1. Dramatic efficiency improvements have been made to the reading and processing of wave load data for fatigue analysis. When the number of wave member loads exceeds 200,000, which is typical for most jacket structure models, efficiency improvements of up to 90% have been observed. The new procedures that provide these efficiency improvements are executed by new options in the READ WAVE LOADS FOR FATIGUE and ASSEMBLE commands as follows:

   READ WAVE LOADS FOR FATIGUE NEW
   ASSEMBLE FOR FATIGUE

The new options are described in detail in Volume 8 of the GTSTRUDL Reference Manual.

2. Offshore punching shear output has been modified to print the largest stress actual/allowable ratio and also the largest geometry check actual/allowable ratio. Previous punching shear output only printed the largest actual/allowable ratio. This resulted in not knowing the stress level of the member when the geometry actual/allowable ratio was the largest ratio. The new output allows the user to inspect the member stress level and also the geometry ratio. Two columns of ratios are now printed under the ACTUAL/ALLOWABLE heading. The first column is the largest stress actual/allowable ratio and the second column is the largest geometry check actual/allowable ratio.

3. Two new output options have been added to the punching shear code check command. These options allow a user to print a summary of the punching shear code check results. The summarize results option prints the largest actual/allowable ratios. This command is documented in Section 4.4 of Volume 8 of the User Reference Manual.

2.10 Steel Design

1. Three new cross-sections have been added to the LRFD3 code. The new cross-sections are Channel, Tee, and Plate Girder. You may code check or design based on axial and bending effects in these cross-sections. One new parameter has been added for the channel cross-section and 10 new parameters for the plate girder cross-section. The new Parameters are discussed in the Table LRFD3.1-1.

2. Double angle cross-sections under bi-axial bending can now be selected and code checked based on the LRFD3 code. Previously, double angle cross-sections could be designed and code checked based on tension or compression force only. Now you can code check or design double angles based on the axial force and bending moments. LRFD3 code check for double angles considers members under tension force , compression force considering flexural-torsional buckling, modified column slenderness, check or design number of connectors, major and minor axis bending check, major and minor axis shear check, and combined axial and bending check. Three new parameters have been added for the double angle cross-sections code check based on the LRFD3 code. Parameters are discussed in the Table LRFD3.1-1.

3. Single angle cross-sections under bi-axial bending now can be code checked based on the LRFD3 code. Previously single angle cross-sections could be code checked based on axial force only. Now, you can code check or design single angles based on axial force and bi-

axial bending. LRFD3 code check for single angles considers members under tension force, compression force considering flexural-torsional buckling, major and minor axis bending check, major and minor axis shear check, and combined axial and bending check. One new parameter has been added for the single angle cross-sections code check based on the LRFD3 code. Parameters are discussed in the Table LRFD3.1-1.

4. 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 ASD9-E code are described in Section 5.2.10 of this Release Guide.

5. The LIST CODE CHECK RESULTS command has been modified to display members that failed member constraint checks. Now, if a member fails a member constraint check but passes the code check, the GTMenu code check graphical display of results and the code check results datasheet will mark the member as a failed member.

An example of the modified LIST CODE CHECK RESULTS output is shown below:

```
{   96} > LIST  CODE  CHECK  RESULTS

============================================================
Results of the latest CHECK or SELECT commands
   Length Units: FEET
      Job ID: MS625
   Job title: Ex2, Check Member Constraints Failure
============================================================

Member   P/F   Section Load      Provision Act/Allow   Provision Act/Allow   Profile
-------- --- --------- --------  --------- ---------   --------- ---------   --------
1         F     12.000 1         H1-1 COM      0.747     B7 COMP     0.407   W21x62
          Failed Member Constraints Check
2         F     12.000 1         H1-1 COM      0.747     B7 COMP     0.407   W21x62
          Failed Member Constraints Check
3         F      7.500 1         H1-3 COM      0.883     B7 COMP     0.508   W21x62
          Failed Member Constraints Check
```

6. Automatic K-factor computation output has been enhanced to display the support status at the ends of the column. Also, when no column exists at the end of a column line, an information message is output.

7. CW (C_w, warping constant) in the steel design NF83 code has always been computed during the code check. CW computation has been modified to check the table database to see if CW

exists in the table database. If CW exists in the table database, it is now used in the NF83 equations. If CW does not exist in the table database, then it is computed.

8. A warning message limit has been added to the following error message for the ASD9 code:

```
**** ERROR_MSA9S7 -- DENOMINATOR   (1 - fa/Fez) FOR MAJOR AXIS BENDING STRESS RATIO IN COMBINED
                                   STRESSES EQUATION (EQ. H1-1) IS LESS THAN OR EQUAL TO ZERO
                                   FOR MEMBER 1 LOADING 1 SECTION 0.00 ACTUAL STRESS fa IS
                                   GREATER THAN EULER STRESS Fez. Z AXIS ELASTIC BUCKLING
                                   FAILURE.
```

To suppress the above warning message, use the OUTPUT WARNING MESSAGE LIMIT command.

9. Joint, member, and load list options of the steel design and connection design commands are now checked for non-existing joints, members, or loads. If a user accidently specifies joint, member, or load names that do not exist, an error message will be output and a list of user specified joints, members, or loads that do not exist will also be output.

10. The following example illustrates the error message and output for a case when a user specified non-existing members for the parameter FYLD.

```
{    92} > UNITS  INCH  KIPS
{    93} > PARAMETER
{    94} >   CODE      ASD9   ALL MEMBERS
{    95} >   FYLD      40.0   MEMBERS  1 TO 5, 20 TO 24, 30, 33, 38


**** ERROR_IOPJML -- Following specified members have not been defined
                     (member incidences have not been specified):
                     20         21         22         23         24         30
                     33         38
```

11. User specified values for yield and tensile strength parameters (FYLD, FTS, Py, Fy, Fu, etc.) are now checked for a range of possible valid values. There are times when a user specifies yield or tensile strength values (parameter FYLD, FTS, Py, Fy, Fu, etc.) in the wrong units or accidently mistypes the parameter values which may result in an incorrect code check. If the user specified values for yield or tensile strength are smaller than 20.0 ksi (. 24.82 MPa) or larger than 200.0 ksi (. 8963.18 MPa), a warning message is issued. Keep in mind that the warning message does not prevent you from performing code check. Examples of the warning message for the ASD9 and 00BS5950 codes are shown below:

```
{    84} > UNITS  POUNDS  FEET
{    85} > PARAMETERS
{    86} >   CODE   ASD9    MEMBER  1
```

```
{   87} >   FYLD   36.0    MEMBER  1

**** WARNING_MSPARC -- Specified value for the parameter FYLD     is equal to     36.0
                       Active Units are   LB    FEET
                       Please check the units or the specified value for a possible ERROR.


{   88} >
{   89} >
{   90} > UNITS  N  M
{   91} > PARAMETERS
{   92} >   CODE   00BS5950   MEMBER  2
{   93} >   Py     235.0      MEMBER  2

**** WARNING_MSPARC -- Specified value for the parameter Py       is equal to     235.
                       Active Units are   N     M
                       Please check the units or the specified value for a possible ERROR.
```

2.11 Steel Tables

1. The following new tables have been added based on the AISC LRFD 3rd Edition:

   a. W-LRFD3 W shapes
   b. M/S/HPL3 M, S, and HP shapes
   c. C-LRFD3 Channels, C and MC shapes
   d. WT-LRFD3 Tee, WT, MT, and ST shapes
   e. 2L-EQ-L3 Equal leg Double Angles
   f. 2L-LL-L3 Long legs back-to-back Double Angles
   g. 2L-SL-L3 Short legs back-to-back Double Angles
   h. 2LALL-L3 All Double Angles. This includes equal leg double angles and long and short legs back-to-back shapes.

2.12 Stiffness Analysis

1. The new STIFFNESS ANALYSIS command option, STIFFNESS ANALYSIS GTHCS BLOCK SIZE, executes the new Georgia Tech Hyper-column solver, which allows for the efficient solution of large structures.

   An example of the efficiency of the new static solver versus the existing solver is shown below:

   Number of Joints = 14,698 (6 DOF) = 88,188 total DOF
   Band = 468
   Time to solve using existing solver =4,242 sec

Time to solve using new GTHCS solver =2,840 sec

Another example illustrating the solution of a very large problem that could not be solved in previous versions of GTSTRUDL is shown below:

Number of Joints = 39,765 (6 DOF) = 238,590 total DOF
Number of Members = 3136
Number of Elements = 43,776
Band = 1,179
Time to solve using existing solver = N/A unable to solve due to running out of virtual memory
Time to solve using new GTHCS solver = 38,778 sec

This page intentionally left blank.

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 Buckling Analysis

1. Linear buckling analysis no longer aborts when joint constraints are in the model. (GPRF 2003.10)

2. Linear buckling analysis no longer aborts when dynamic loads are present in the list of active loads. (GPRF 2004.1)

3.2 Dynamic Analysis

1. Both consistent and lumped masses are now computed correctly for plane frame members having variable properties with three or more segments and where only the AX, IY or IZ section properties are specified. (GPRF 99.17)

2. The RESPONSE SPECTRUM LOAD command no longer aborts when the specified response spectrum file is not found. (GPRF 2003.9)

3. Dynamic transient analysis now computes correct displacements, velocities, and accelerations for condensed degrees of freedom when the external file solver is used.
(No GPRF issued as this was a prerelease feature in Version 27)

4. Modal time history analysis using the PERFORM TRANSIENT ANALYSIS, PERFORM MODE SUPERPOSITION ANALYSIS, and DYNAMIC ANALYSIS MODE SUPERPOSITION, correctly computes absolute accelerations when the USE EXTERNAL FILE SOLVER is active and there are two or more active transient loads having SUPPORT ACCELERATION components.
(No GPRF issued as this was a prerelease feature in Version 27)

3.3 Finite Elements

1. The results from the LIST PRINCIPAL STRESSES command are now correct for loading combinations for the IPQL series (IPQL, IPQL2,..., etc.) elements. (GPRF 91.44 and 91.45)

2. If a loading containing Joint Temperatures is later changed and the Joint Temperatures deleted, the joints which contain the deleted Joint Temperatures are no longer changed to planar element (PLAN-EL) joints. Furthermore, other characteristics associated with the joints are no longer incorrectly changed either. (GPRF 2001.7)

3. If a loading containing Joint Temperatures is later deleted and a new loading is created with the same name (ID), the original Joint Temperature loadings will no longer be present in the new loading. Issuing the PRINT LOADING DATA command will print out values of 0.0 for all of the joints which originally had Joint Temperatures unless the new loading also contains Joint Temperatures. In that case, the new values specified for the Joint Temperatures will appear in the PRINT LOADING DATA output. (GPRF 2001.8)

4. Local Surface Forces on 3D solid finite elements (IPLS, IPQS, IPSL, IPSQ, TRIP, and WEDGE15) now produce correct results if the normal to the front face of the element points into the element. Previously, the sign of the forces was incorrect and the loads were being applied in a direction opposite to what was expected and opposite to the direction displayed in GTMenu. Global surface forces on these elements were not affected by this problem. (GPRF 2004.10)

3.4 General

1. LIST SECTION DISPLACEMENT results are now correct when a member has one of the following combination of releases and shear center eccentricities:
   a. Start Moment Z member release and EZ shear center eccentricity
   b. Start Moment Y member release and EY shear center eccentricity

   (GPRF 2003.12)

2. The Self Weight Command will now be recognized correctly when it immediately follows a Member Property command with Variable properties specified using segments. (GPRF 2004.11)

3. The Variable Member Property and Variable Member Dimension commands will now function correctly if a Group is used to specify the list of members with variable properties or variable dimensions in tabular format. (GPRF 2004.12)

3.5 GTMenu

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

1. In GTMenu, the Hit mode is no longer lost by opening the Label or Shrinkage settings dialogs. In some instances, the Mode Bar would show that Hit mode was set but the user would have to close the current dialog or select Hit again on the Mode Bar.

2. Random characters are no longer displayed for the load name in the Display Load menu when no static loads are defined.

3. In a model which contains a mixture of 3D solid and 2D finite elements, the first element in the current view is now checked to determine whether the contouring mode is for solid or two-dimensional elements.

4. An abort in GTMenu when many members or elements are given without incidences has been corrected.

5. GTMenu will no longer display a mode number as -1 in the Legend when displaying dynamic modes.

6. The input file generated when using the TRANS3D solid element is now correct and the "With Nodes" specification for the TRANS3D is now written on the line with the element type.

7. Requesting reaction values to be labeled or displayed at a free joint will no longer cause an abort.

8. Results are no longer labeled incorrectly under the following conditions:
   Results (reactions, joint displacements,...etc.) have been displayed.
      The load is labeled.
      Results are then labeled.

9. Edit Thickness from the Edit Element dialog no longer goes into an infinite loop following a right-click Edit of an element's thickness.

10. In some instances during contouring finite element results, a triangular area would not contain color-filled contours. The missing triangles in a color-filled contour have been corrected.

11. The color scale used for color contours with a smaller-than-default step size has been corrected.

12. Previously, GTSTRUDL would occasionally abort following a deformed shape animation involving a cable element. This has been corrected.

13. Multiple joints at the same location can now have their contour values labeled separately. Multiple joints at the same location may occur in models containing master and slave joints or joint ties.

14. The following displays are corrected in GTMenu for the special case of the Beta Angle, that is, for when eccentricities cause a member to become vertical that wasn't already vertical or if a member is vertical and the eccentricity makes the member non-vertical:
    Local Axes (Display Model)
    Cross section with Beta (Display Model)
    Redraw Solid for table sections
    Force or moment diagram or envelop (Results).

15. The display of finite element incidence order, and planar axes is now ignored for cable elements and nonlinear springs.

16. Bringing-up the Inquire list box using the Button Bar or by HotKeys no longer disables the Mode Bar.

17. Deleted joints that have unspecified coordinates are no longer listed upon entering GTMenu.

18. An abort will no longer occur when entering GTMenu with a model which contains Superelement and Group definitions.

19. A Display → Moving Load abort has been corrected, where a model with no joint displacements would cause animation of the deformed structure with a moving load to abort.

20. Redraw Solid performed on a deformed display no longer cause a subsequent Redraw to use deformed colors on an undeformed display. Now, the colors are reset for the undeformed display.

21. Variable Property Group descriptions have been revised when the Property Groups are displayed. Previously, extraneous characters could appear and the word “variable” was abbreviated.

22. An abort will no longer occur when sending Inquire Output to a file having a long pathname.

23. Occasionally when checking for duplicate elements, an element would be erroneously indicated as being a duplicate element. This has been corrected.

24. Creating Elements will no longer abort when using Create Members and Elements Using Grid.

25. Member properties for members with variable properties are now correct after the model has been brought into GTMenu. This problem occurred only in Version 27. The error could be determined by looking at the Member Properties dialog in GTMenu or by issuing the PRINT MEMBER PROPERTIES command after exiting GTMenu with changes. The input file created by GTMenu would also contain incorrect member properties.

26. The Input File created in GTMenu will no longer after an erroneous continuation mark (-) on the last line of a list of Member Constraints. This occurred only if the last line contained exactly five members.

27. The number of property groups allowed by Redraw Solid has been increased from 1000 to 10000. Although 1000 property groups seems like a large number of property groups, offshore models with variable member properties could exceed the 1000 limit and cause unpredictable results.

28. Variable pressures on a finite element mesh no longer generate incorrect element surface forces with some of the forces being in the opposite direction of the applied pressure.

29. An abort will no longer occur when creating finite elements using the Create Finite Elements Using Grid option.

30. The precision of the data written to this View file has been increased. Previously, a problem could occur when a previously created View file was read and some of the views did not display due to a lack of precision (not enough significant digits) of the information in the View file.

31. An abort in GTMenu has been corrected for models which contain IPCABLE elements with fewer than six nodes.

32. A range of loads can now be selected correctly in GTMenu for force and moment envelopes.

33. The CALCULATE RESULTANT command will no longer produce incorrect results under the following conditions;

    a. Stiffness analysis is performed prior to entering GTMenu

    b. GTMenu is entered and the user makes changes to the model such as creating a new view, exits GTMenu, and then issues the CALCULATE RESULTANT command or

       GTMenu is entered and the user "Saves" the model while in GTMenu. The model is subsequently restored and the CALCULATE RESULTANT command is given.

    This error is due to GTMenu destroying the rotation matrix that transforms the element forces from the planar coordinate system to the global coordinate system.

34. Previously, if a structure had eccentricities, GTSTRUDL might abort when GTMenu wrote an input file while alphabetizing the joints. This has been fixed.

3.6 Nonlinear Analysis

1. Nonlinear static analysis no longer aborts for multiple loading conditions when linear elastic connection member releases are present.(GPRF 2003.5)

2. Nonlinear analysis has been corrected and now produces correct results when the following conditions exist simultaneously:

    a. Tension/compression-only is the only nonlinear effect specified for a plane frame or space frame member.

    b. Compression strain is computed for such a compression-only frame member in the final equilibrium iteration of a convergent nonlinear analysis, when tension was

detected in an earlier equilibrium iteration prior to convergence; or vice versa for a tension-only member.
(GPRF 2003.12)

3. The NONLINEAR ANALYSIS command no longer aborts when nonlinear spring elements are present in the model and all joints are fully fixed supports. (GPRF 2004.3)

4. The computation of nonlinear static analysis reaction components corresponding to the fixed degrees of freedom in released support joints has been corrected for the case when joint loads are applied to those fixed degrees of freedom. (GPRF 2004.6)

5. Nonlinear dynamic analysis now produces consistently correct results for the case where the sign of the axial strain in tension/compression only, linear geometry frame members reverses between time points. (No GPRF issued as this was a prerelease feature in Version 27 and remains a prerelease in Version 28)

6. Nonlinear analysis, including pushover analysis, no longer aborts if tension/compression only and material nonlinearity (plastic hinge, nonlinear member end connection, plastic segment) are mixed in a frame member without also specifying nonlinear geometry.
(No GPRF issued as this was a prerelease feature in Version 27 and remains a prerelease in Version 28)

7. The second in a sequence of consecutive nonlinear dynamic analyses no longer aborts when member loads are present in the initial stress loading specified in the DYNAMIC PARAMETERS command.
(No GPRF issued as this was a prerelease feature in Version 27 and remains a prerelease in Version 28)

8. Nonlinear dynamic analysis now produces correct results for the case where the sign of the axial strain in tension/compression only, and linear geometry frame members reverses between time points. (No GPRF issued as this was a prerelease feature in Version 27 and remains a prerelease in Version 28)

3.7 Offshore

1. The PILE ANALYSIS USING SUPERELEMENT abort has been corrected. (GPRF 2004.9)

2. The LIST FATIGUE TRANSFER FUNCTIONS command will no longer abort if given without any of the directions, members, and positive data options, following one or more COMPUTE FATIGUE commands. (GPRF 2004.13)

3.8 Reinforced Concrete

1. The diameter of circular steel cage reinforcement is now reported correctly for the CP110 and BS8110 concrete design for round columns using the PRINT REINFORCEMENT command. (GPRF 2003.6)

2. An abort in Proportioning columns designed using the ACI codes will no longer occur during the capacity check phase. (GPRF 2003.7)

3.9 Rigid Bodies

1. The deletion of a rigid body no longer inactivates a slave joint to which only one member or finite element is incident. (GPRF 99.14)

2. The process of deleting rigid bodies no longer incorrectly inactivates slave joints on which only one other member or finite element is incident. (GPRF 2004.4)

3. Static analysis no longer aborts when member/element loads are applied to rigid bodies. A warning message flags this condition and the analysis proceeds normally, ignoring the member loads applied to the rigid bodies. (GPRF 2004.5)

4. GTSTRUDL analyses (stiffness, dynamic, etc.) no longer abort or indicate the presence of instabilities if the model contains slave joints that have multiple master joints. Data consistency checking now terminates an analysis when this condition is detected. (GPRF 2004.8)

3.10 Scope Environment

1. The LABEL BETA command now correctly rotates the cross-section shapes according to the position of the member after member eccentricities have been applied. (GPRF 92.11)

2. A Scope Environment contour of a rotated window on a 3-D structure now correctly calculates which elements should be averaged prior to drawing the contour.
(No GPRF was issued)

This page intentionally left blank.

CHAPTER 4

KNOWN DEFICIENCIES

This chapter describes known problems or deficiencies in Version 28. 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 <u>7 INC 1 TO 8 INC 1</u>

   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 → 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)

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 28 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 LRFD3 Tables
  - 5.2.3 BS5950 Steel Design Code and Parameters
  - 5.2.4 Steel Design by Indian Standard Code IS800
  - 5.2.5 IS800 Tables
  - 5.2.6 Steel Deflection Check and Design
  - 5.2.7 Brazilian Table
  - 5.2.8 ACI Code 318-99
  - 5.2.9 Rectangular and Circular Concrete Cross Section Tables
  - 5.2.10 ASD9-E Code

- 5.3 Analysis Prerelease Features
  - 5.3.1 Calculate Error Estimate Command
  - 5.3.2 Output of Response Spectrum Results
  - 5.3.3 Form Static Load Command
  - 5.3.4 Form UBC97 Load Command
  - 5.3.5 Form IS1893 Load Command

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 LRFD3 Code

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

I shapes	Subjected to bi-axial bending and axial forces
Channels	Subjected to bi-axial bending and axial forces
Single Angles	Subjected to bi-axial bending and axial forces
Double Angles	Subjected to bi-axial bending and axial forces
Tees	Subjected to bi-axial bending and axial forces
Round HSS (Pipes)	Subjected to bi-axial bending, axial, and torsional forces
Rectangular and Square HSS (Structural Tubes)	Subjected to bi-axial bending, axial, and torsional forces
Plate Girders	Subjected to bi-axial bending and axial forces

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.

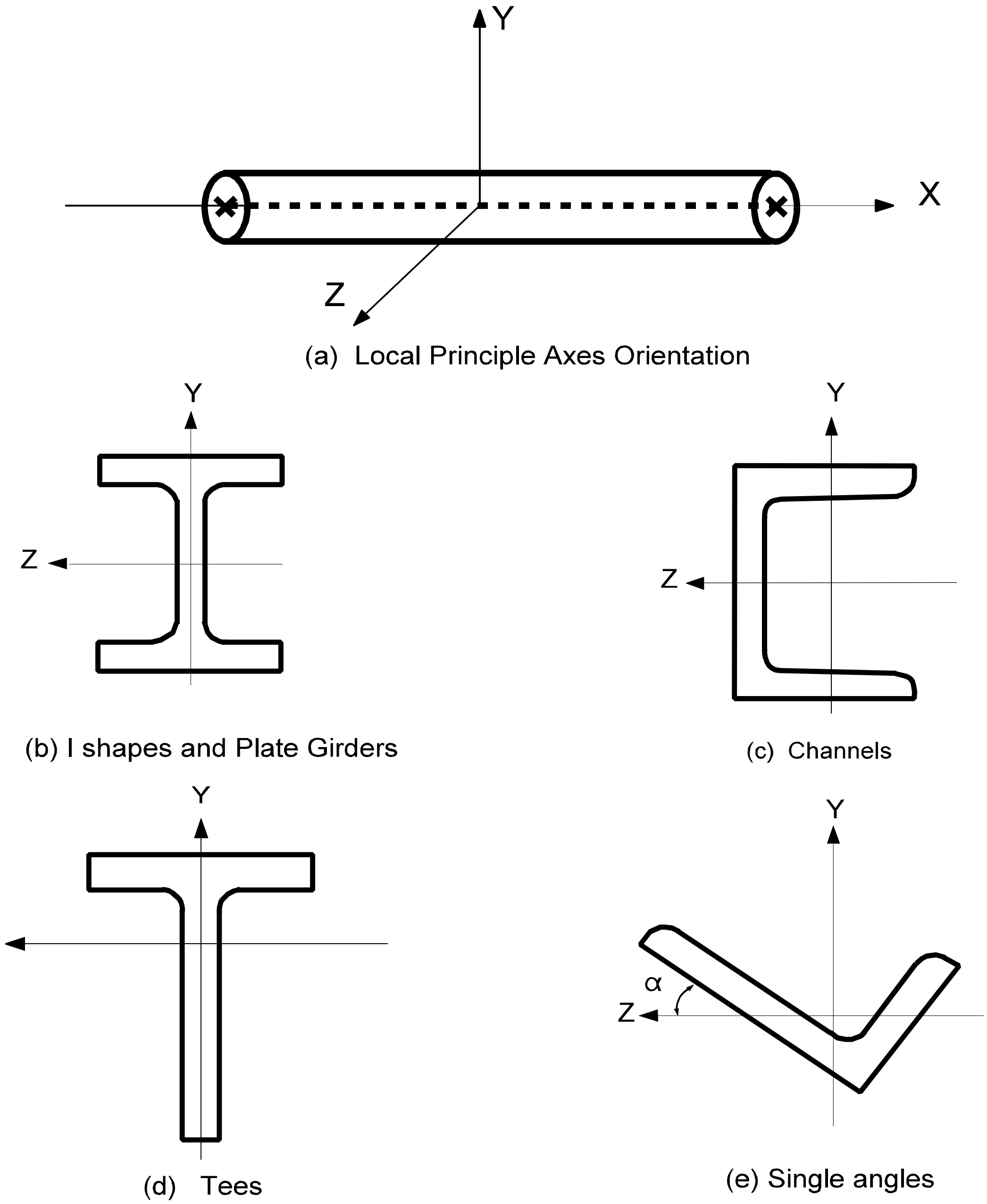
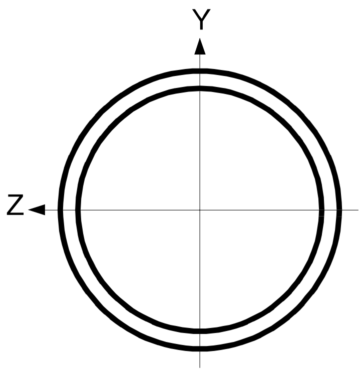
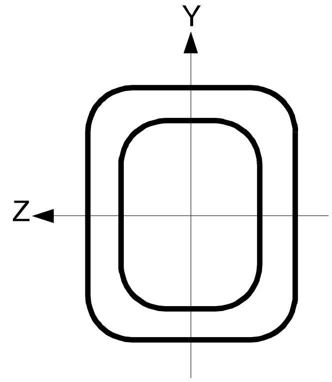


Figure LRFD3.1-1 Local Axes for Design with LRFD3

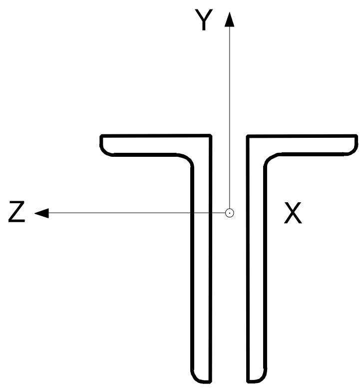


(f) Pipes

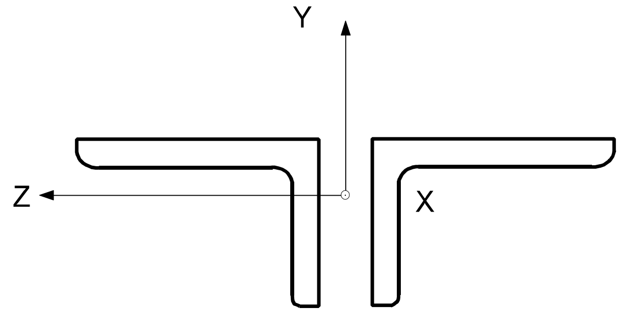


(g) Structural tubing

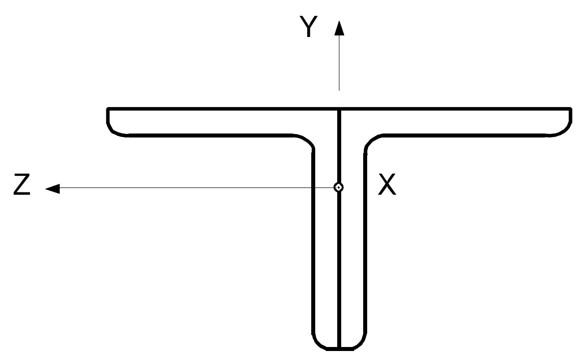
Figure LRFD3.1-1 Local Axes for Design with LRFD3 (continued)



(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, 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, and plate girders). The designer is reminded to check the torsional stresses for open cross-sections (I shape, channels, single angles, double angles, tees, 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

Section	Title
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 $L_b \leq L_r$
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
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

<u>Section</u>	<u>Title</u>
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

<u>Section</u>	<u>Title</u>
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

<u>Section</u>	<u>Title</u>
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 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	Title
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 except single and double angle (tension or compression axial strengths only). 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 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.
4.	Sections LRFD3.4	Describes the subsections in the Section LRFD3.4.
5.	Section LRFD3.4.1	Defines the symbols used in the LRFD3 code provisions.
6.	Section LRFD3.4.2	Contains detailed discussion of the code provisions and the equations applicable to the I shape cross-sections subjected to axial and bending forces.
7.	Section LRFD3.4.3	Contains detailed discussion of the code provisions and the equations applicable to the single angle cross-sections subjected to axial force only.
8.	Section LRFD3.4.4	Contains detailed discussion of the code provisions and the equations applicable to the double angle cross-sections subjected to axial force only.

Section	Title	
9.	Section LRFD3.4.5	Contains detailed discussion of the code provisions and the equations applicable to the round HSS (pipe) cross-sections subjected to axial, bending, and torsional forces.
10.	Section LRFD3.4.6	Contains detailed discussion of the code provisions and the equations applicable to the rectangular and square HSS (structural tube) cross-sections subjected to axial, bending, and torsional forces.
11.	Section LRFD3.4.7	Contains detailed discussion of the code provision and the equations applicable to the plate girder cross-sections subjected to axial and bending forces.
12.	Section LRFD3.4.8	Contains detailed discussion of the code provision and the equations applicable to the channel cross-sections subjected to the axial and bending forces.
13.	Section LRFD3.4.9	Contains detailed discussion of the code provision and the equations applicable to the tee cross-sections subjected to the axial and bending forces.

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). LRFD3 code is applicable to the following cross-sections: 1. I shape, Channel, Tee, Double Angle, and Plate Girder profiles subjected to axial and bending. 2. Round HSS (Pipe), Rectangular and Square HSS (Structural Tube) profiles subjected to axial, bending, and torsional forces. 3. Single Angle profile subjected to axial force only. See Sections LRFD3.2, LRFD3.3, and LRFD3.4 for a more detailed description.
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 (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 plate 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 (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 (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 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
K-Factors (continued)		
SDSWAYZ	YES	Indicates the presence or absence of sidesway about the local Z axis. YES=sidesway permitted. NO=sidesway prevented.
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).
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.

Table LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Buckling Length (continued)		
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 (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 (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 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 code check. When the SELECT MEMBER (design) is requested, the user specified value is used unless more connectors are required. If the

Table LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	Default Value	Meaning
Double Angle Parameters (continued)		
		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 physical 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 (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 (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.
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

Table LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Plate Girder Parameters (continued)		
		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.
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

Table LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Plate Girder Parameters (continued)		
		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.
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.

Table LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
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.
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 an explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces.

Table LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Output Processing and System Parameters (continued)		
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.

Table LRFD3.1-2

GTSTRUDL AISC Codes*

Code Name	Parameter Table	Application
LRFD3	LRFD3.1-1 Volume 2C	Checks compliance of I shapes, channels, tees, and double angles (subject to bi-axial bending and axial force), round HSS (Pipes), rectangular and square HSS (structural tubes) (subjected to bi-axial bending, axial, and torsional forces), single and double angles (subjected to axial forces only) shape profiles to the 1999 AISC LRFD, Third Edition, Specification (96).
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.
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.

* 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**
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.

This page intentionally left blank.

5.2.2 GTSTRUDL LRFD3 Profile Tables

Table LRFD3.1-3

GTSTRUDL Profile Tables for the Design based on the LRFD3 Code

Profile Shapes	**Reference**
I shapes	See Table D-1 in Appendix D for list of applicable table names for I shapes, W, S, M, HP shapes, wide flange shapes, universal beam shapes, universal column shapes, etc.
Single Angles	See Table D-2 in Appendix D for list of single angle table names applicable to LRFD3 code.
Double Angles	See Table D-3 in Appendix D for list of double angle table names applicable to LRFD3 code.
Round HSS	See Table D-4 in Appendix D for list of round HSS (pipe, circular hollow section) table names applicable to LRFD3 code.
Rectangular HSS	See Table D-5 in Appendix D for list of rectangular and square HSS (structural tube, rectangular and square hollow section) table names applicable to LRFD3 code.
Channels	See Table D-6 in Appendix D for a list of channel table names applicable to LRFD3 codes.
Tees	See Table D-7 in Appendix D for a list of tee table names applicable to LRFD3 codes.
Plate Girders	See Table D-9 in Appendix D 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.

5.2.3 GTSTRUDL BS5950 Steel Design Code and Parameters

00BS5950 Code
British Standard
BS 5950-1:2000

00BS5950.1 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)	Subject 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 primarily 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.

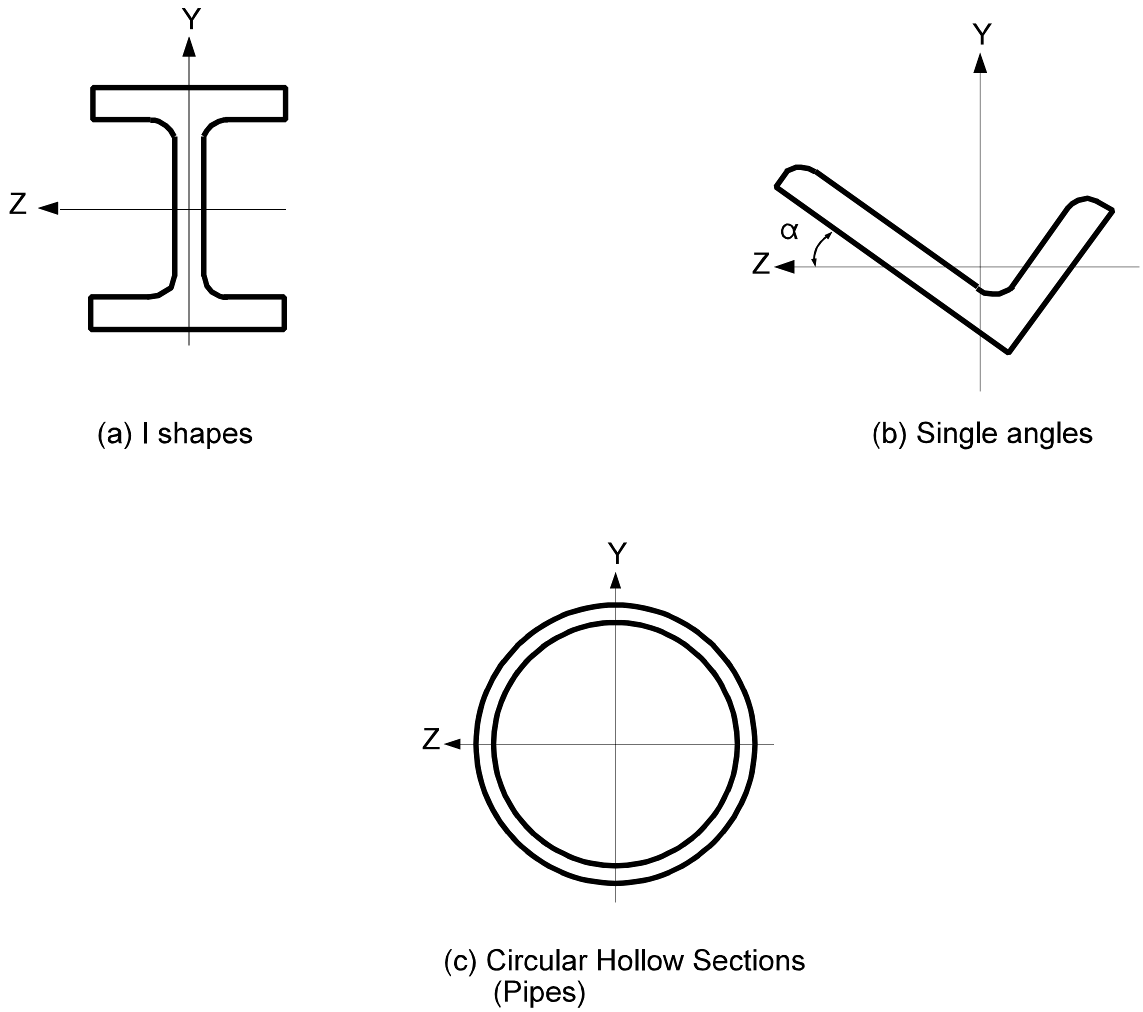


Figure 00BS5950.1-1 Local Axes for Design with 00BS5950

The sections of the BS 5950-1:2000 specifications (95) which 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

Section	Title
4.3.6.6	Equivalent uniform moment factor m_{LT}
	Table 18. Equivalent uniform moment factor m_{LT} for lateral-torsional buckling
4.3.6.9	Ratio β_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

Section	Title
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
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 Table 00BS5950.1-2 and 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 00BS5950.1-3 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 the a/d ratio where 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 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 00BS5950.1-4 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 00BS5950.1-5 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 00BS5950.1-5 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 00BS5950.1-6 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 00BS5950.1-6 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**
		<u>Equivalent Uniform Moment Factors</u> (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 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 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 for the explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties.

Table 00BS5950.1-2

GTSTRUDL British Standard Codes

Code Name	Parameter Table	Application
00BS5950	00BS5950.1-1 Volume 2C	Checks compliance of I shapes and Circular Hollow Sections (Pipes) (subjected to bi-axial bending and axial force), and also Single Angles (subjected to axial forces only) profiles to the BS 5950-1:2000, Part 1 specification (95), adopted May, 2001.
BS5950	BS5950 Volume 2A	Checks compliance of I, Single angle, Channel, Tee, Double angle, Solid round bar, Pipe, Solid square and Rectangular bar, and Structural tubing shape profiles to the BS 5950: Part 1: 1990 specification (79), adopted July 31, 1990.
BS449	BS449 Volume 2A	Checks compliance of I, Single angle, Channel, Tee, Pipe, and Structural tubing shape profiles to the British Standard 449, part 2 Metric Units, Specifications for the Use of Structural Steel in Building, British Standard Institution, October 1969, with amendments through July 1975.

Table 00BS5950.1-3

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

Profile Shapes	**Reference**
I shapes	See Table D-1 in Appendix D 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 Table D-2 in Appendix D for list of single angle Table names applicable to 00BS5950 code.
Circular Hollow Sections	See Table D-4 in Appendix D for list of Circular Hollow Section (Pipe, Round HSS) Table names applicable to 00BS5950 code.

Table 00BS5950.1-4

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 (N/mm^2)		
	$t \leq 16$	$16< t \leq 40$	$40< t \leq 63$	$63< t \leq 80$	$80< t \leq 100$	$100< t \leq 150$	$150< t \leq 200$	$200< t \leq 250$	$t \leq 100$	$100< t \leq 150$	$150< t \leq 250$
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 00BS5950.1-4 (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 (N/mm^2)		
	$t \le 16$	$16 < t \le 40$	$40 < t \le 63$	$63 < t \le 80$	$80 < t \le 100$	$100 < t \le 150$	$150 < t \le 200$	$200 < t \le 250$	$t \le 100$	$100 < t \le 150$	$150 < t \le 250$
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 00BS5950.1-5

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 00BS5950.1-6

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).

5.2.4 GTSTRUDL Indian Standard Design Code IS800

A new steel design code named IS800 has been added. This code is based on the Indian Standard, IS:800-1984, Code of Practice for General Construction in Steel, Second Revision. Applicable cross-sections for the IS800 code as follows:

I-shapes	Solid Round Bars
Channels	Pipes
Single Angles	Solid Square Bars
Tees	Solid Rectangular
Double Angles	Structural Tubes

Table IS800

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 Section IS800 for a more detailed description.
TBLNAM	ISBEAMS	Identifies the table of profiles to be used during selection. See Table 2.1-2a 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’).

Table IS800 (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 2.1-3 in Volume 2A 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.

Table IS800 (continued)

IS800 Code Parameters

Parameter Name	**Default Value**	**Meaning**
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. See the COLUMN/BEAM LINE command (Section 2.3 of Volume 2A). 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.
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 (continued)

IS800 Code Parameters

Parameter Name	**Default Value**	**Meaning**
K-Factors (continued)		
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).
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.

Table IS800 (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 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 IS800 (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.
CMY	Computed	Coefficient which modifies Y axis bending stress in interaction equation (IS:800-1984 Second Ed., Section 7).
CMZ	Computed	Coefficient which modifies Z axis bending stress in interaction equation (IS:800-1984 Second Ed., Section 7).
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	89.0(N-mm)	Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero.
MZMIN	89.0(N-mm)	Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero.

Table IS800 (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.
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.

Table IS800 (continued)

IS800 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Output Processing and System Parameters (continued)		
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.

5.2.5 GTSTRUDL Profile Tables for the Design based on the IS800 Code

The profile tables for design based on the IS800 Code are shown below:

Table 2.1-2l

GTSTRUDL Profile Tables for the Design based on the IS800 Code*

Table Name	Reference
Angle shape Tables	
ISEQANGL	Equal single leg angle cross-sections from Tables 5.1 and 5.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision.
ISUNANGL	Unequal single leg angle cross-sections from Tables 6.1 and 6.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision.
ANGLES	Equal and unequal single leg angles from 1969 AISC ASD Seventh Edition (16).
ANGLESM	Equal and unequal single leg angles from AISC Metric “ANGLES, Equal legs and unequal legs” table (83).
EQANGLE	Equal single leg angles from 1969 AISC ASD Seventh Edition (16).
ULANGLE	Unequal single leg angles from 1969 AISC ASD Seventh Edition (16).
USANGLE	Unequal single leg angles from 1969 AISC ASD Seventh Edition (16).
EQDBLANG	Double angles with equal legs back-to-back from 1969 AISC ASD Seventh Edition (16).
LLDBLANG	Double angles with long legs back-to-back from 1969 AISC ASD Seventh Edition (16).

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

Table 2.1-2l (continued)
GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

Table Name	**Reference**

Angle shape Tables (continued)

SLDBLANG	Double angles with short legs back-to-back from 1969 AISC ASD Seventh Edition (16).
EQDBANGM	Equal leg double angle profiles from AISC Metric "DOUBLE ANGLES, Two equal leg angles, Properties of sections" table (83).
LLDBANGM	Unequal leg, long leg back-to-back, double angle profiles from AISC Metric "DOUBLE ANGLES, Two unequal leg angles, Properties of sections, Long legs back-to-back" table (83).
SLDBANGM	Unequal leg, short leg back-to-back, double angle profiles from AISC Metric "DOUBLE ANGLES, Two equal leg angles, Properties of sections, Short legs back-to-back" table (83).
BSEQANGL	Equal leg single angle profiles from British "EQUAL ANGLES" table (82).
BSEQDBAN	Equal leg double angle profiles from British "COMPOUND EQUAL ANGLES LEGS BACK TO BACK" table (82).
BSLLDBAN	Unequal long legs back-to-back double angle profiles from British "COMPOUND UNEQUAL ANGLES LONG LEGS BACK TO BACK" table (82).
BSSLDBAN	Unequal short legs back-to-back double angle profiles from British "COMPOUND UNEQUAL ANGLES SHORT LEGS BACK TO BACK" table (82).

BAR shape Tables

BARS	Round and square solid bar shapes.
RBAR	Rectangular solid bar shapes.

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

Table 2.1-2l (continued)
GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

Table Name	**Reference**
Channel shape Tables	
ISCHAN	Channel sections (sloping flange channels) from Table 4.1 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision.
ISCHAMP	Channel sections (parallel flange channels) from Table 4.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision.
CHANNEL9	Channel shapes from 1989 AISC ASD Ninth Edition (72).
CHANNELM	Channel shape profiles from AISC Metric “CHANNELS AMERICAN STANDARD” table (83).
CHANNELS	Channel shapes from 1978 AISC ASD Eighth Edition (33).
BSCHAN	Channel shape profiles from British “CHANNELS” table (82).
I shape Tables	
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.
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.

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

Table 2.1-2l (continued)

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

Table Name	**Reference**
	I shape Tables (continued)
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 Appendix C of Volume 2A for Table description and profile names.

Table 2.1-2l (continued)

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

Table Name	**Reference**
I shape Tables (continued)	
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).
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".
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.

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

Table 2.1-2l (continued)

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

Table Name	Reference
Pipe shape Tables	
PIPES9	Pipe shapes from 1989 AISC ASD Ninth Edition (72).
PIPESM	Pipe shapes from AISC Metric “PIPE” table (83).
AISCPIPE	Pipe shapes from 1978 AISC ASD Eighth Edition (33).
SSPIPE	Pipe shapes from ANSI B36.19-1976.
WSPIPE	Pipe shapes from ANSI B36.10-1979
CIHOLLOW	British Circular Hollow profiles from 1996 BS 5950 Section Properties, 4th Edition (82).
Tee shape Tables	
TEES9	Tee shapes from 1989 AISC ASD Ninth Edition (72).
TEESM	Tee shape profiles from AISC Metric “STRUCTURAL TEES, Cut from W shapes” table (83).
TEES	Tee shapes from 1978 AISC ASD Eighth Edition (33).
TEEUBEAM	Tee shape profiles from British “STRUCTURAL TEES CUT FROM UNIVERSAL BEAMS” table (82).
TEEUCOLU	Tee shape profiles from British “STRUCTURAL TEES CUT FROM UNIVERSAL COLUMNS” table (82).

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

Table 2.1-2l (continued)

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

Table Name	**Reference**
	Tube shape Tables (continued)
TUBES9	Structural Tubing shapes from 1989 AISC ASD Ninth Edition (72).
TUBESM	Structural Tubing shapes from AISC Metric “STRUCTURAL TUBING Square and Rectangular” table (83).
TUBE80	Structural Tubing shapes from 1978 AISC ASD Eighth Edition (33).
AISCTUBE	Structural Tubing shapes from 1969 AISC ASD Seventh Edition (16).
REHOLLOW	British Rectangular Hollow profiles from 1996 BS 5950 Section Properties, 4th Edition (82).
SQHOLLOW	British Square Hollow profiles from 1996 BS 5950 Section Properties, 4th Edition (82).

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

This page intentionally left blank.

5.2.6 Steel Deflection Check and Design

Deflection check and design is now available for steel design. Deflection check and design for steel members can be performed with or without a stress code check or select. Deflection check or design is available for all steel design codes (i.e. ASD9, LRFD2, BS5950,CAN97, EC3, IS800, etc.). For physical members, only a deflection check is valid at the present time. Deflection design has not been implemented at this time for physical members or when the start and end joints are specified by the parameters ‘DefStaJT’ and ‘DefEndJT’.

Deflection check or design is based on the member or physical member chord deflection (see LIST SECTION DISPLACEMENTS command, Section 2.1.14.6 of the Volume 1). Chord deflection is the displacement of the member or physical member relative to a line between member or physical member deflected end points in the member or physical member reference frame (i.e. the member’s start and end deflection are always equal to 0.0).

Only the default value for the TRACE parameter is valid for deflection check. If a value other than 4 (default value) has been specified for the parameter TRACE, output from CHECK or SELECT will not have deflection result information. Note that the WITH AXIAL, BEAM, or GENERAL options of the SELECT command are not valid for deflection design. These options are not supported for the SELECT command.

Nine new parameters are available for the deflection check or design. The new parameters are described below:

New Deflection Check Parameters

Parameter Name	Default Value	Meaning
DefCheck	NO	Parameter to request the deflection check or design. Deflection check or design is based on the member or physical member chord deflection (see LIST SECTION DISPLACEMENTS command, Section 2.1.14.6 of Volume 1). Chord deflection is the displacement of the member or physical member relative to a line between member or physical member deflected end points in the member or physical member reference frame (i.e. the member’s start and end deflection are always equal to 0.0). Values of ‘NO’, ‘YES’, ‘YES-Y’, ‘YES-Z’,

'ONLY', 'ONLY-Y', and 'ONLY-Z' are valid for this parameter and explanations of these parameters are as follows:

NO	=	No deflection check or design.
YES	=	Perform deflection code check (CHECK MEMBER command) or design a member to satisfy deflection requirement (SELECT MEMBER command). This option performs a deflection check or design for local Y and Z axis and also performs code check or design based on the user specified code. When this option is used, the design code name must be specified also.
YES-Y	=	Perform deflection check or design for local Y axis deflection. Also, perform a code check or design based on the user specified code. When this option is used, the design code name must be specified also.
YES-Z	=	Perform deflection check or design for local Z axis deflection. Also, perform a code check or design based on the user specified code. When this option is used, the design code name must be specified also.
ONLY	=	Perform only the deflection code check (CHECK MEMBER command) or design of a member to satisfy deflection requirement (SELECT MEMBER command). This option performs a deflection check for local Y and Z axis. This option does not perform a code check or code design but does perform a deflection check or deflection design.
ONLY-Y	=	Perform only the deflection check or design for local Y axis deflection. This option does not perform a code check or code design but does perform a deflection check or deflection design.

		ONLY-Z = Perform only the deflection check or design for local Z axis deflection. This option does not perform a code check or code design but does perform a deflection check or deflection design.
DefLoads	NO	Parameter to specify the loadings that you want to be used for the deflection check. A value of YES for a list of loads defines the loads to be used for the deflection check. If this parameter has not been specified, the active loads are used for the deflection check.
DefLimit	360	Parameter to specify the deflection limitation. Member length (chord length) is divided by the value of this parameter which defines the deflection limitation (L/360). This value is used for both local Y and Z direction deflection check.
DefLim-Y	Computed	Parameter to specify the local Y direction deflection limitation. This parameter is used for the local Y direction deflection check. Member length (chord length) is divided by the value of this parameter which defines the Y direction deflection limitation (L/360). If not specified, the value for parameter 'DefLimit' is used.
DefLim-Z	Computed	Parameter to specify the local Z direction deflection limitation. This parameter is used for the local Z direction deflection check. Member length (chord length) is divided by the value of this parameter which defines the Z direction deflection limitation (L/360). If not specified, the value for parameter 'DefLimit' is used.
DefPhys	NO	Parameter to define the members that you want to be deflection checked based on the member's physical length. The specified members must be part of a physical member. A value of YES indicates that the deflection check should be based on the physical member chord. Deflection design is not available for physical members at this time.

DefStaJT	Computed	This parameter can be used to specify the start joint and parameter 'DefEndJT' can be used to specify the end joint of the chord that the member is lying on. The deflection is checked based on the chord through the joints specified by the parameters 'DefStaJT' and 'DefEndJT'. Start joint name followed by a list of members should be specified for this parameter (ex: DefStaJT JNT1 Members 'M1' 'M2' 'M3'). Deflection design is not available for this option at this time.
DefEndJT	Computed	This parameter can be used to specify the end joint and parameter 'DefStaJT' can be used to specify the start joint of the chord that the member is lying on. The deflection is checked based on the chord through the joints specified by the parameters 'DefStaJT' and 'DefEndJT'. End joint name followed by a list of members should be specified for this parameter (ex: DefEndJT JNT4 Members 'M1' 'M2' 'M3'). Deflection design is not available for this option at this time.
DefNuSec	9	Parameter to specify the number of sections to be used for the computation of the member deflection. The default of 9 sections is used to compute the section deflections. The deflection check is performed for the location that has the largest deflection.

Example:

The example illustrates the usage of several of the new parameters needed in order to perform a deflection check. The user should note that both factored and unfactored loading combinations are required since deflection checks are usually performed on unfactored (service) loads.

```
$  Factored load combinations for code check

LOADING  COMBINATION  'A'  'SW + Live Load'  -
        COMBINE   'SW'  1.0   'LL'  1.0
LOADING  COMBINATION  'B'  '0.75(SW + Live Load + Wind Load from Right)'
-
        COMBINE   'SW'  0.75   'LL'  0.75   'WLRX'  0.75
```

```
        LOADING  COMBINATION  'C'  '0.75(SW + Live Load + Wind Load from Left)'
-
              COMBINE   'SW'  0.75   'LL'  0.75   'WLLX'  0.75
        LOADING  COMBINATION  'D'  '0.75(SW + Live Load + Wind Load from Back)'
-
             COMBINE   'SW'  0.75   'LL'  0.75   'WLBZ'  0.75
        LOADING  COMBINATION  'E'  '0.75(SW + Live Load + Wind Load from Front)'
-
             COMBINE   'SW'  0.75   'LL'  0.75   'WLFZ'  0.75

        $  Service load combinations for deflection check

        LOADING  COMBINATION  'Def-F'  'SW + Live Load + Wind Load from Right'
-
            COMBINE   'SW'  1.0   'LL'  1.0   'WLRX'  1.0
        LOADING  COMBINATION  'Def-G'  'SW + Live Load + Wind Load from Left'  -
            COMBINE   'SW'  1.0   'LL'  1.0   'WLLX'  1.0
        LOADING  COMBINATION  'Def-H'  'SW + Live Load + Wind Load from Back'
-
            COMBINE   'SW'  1.0   'LL'  1.0   'WLBZ'  1.0

        LOADING  COMBINATION  'Def-I'  'SW + Live Load + Wind Load from Front'  -
            COMBINE   'SW'  1.0   'LL'  1.0   'WLFZ'  1.0
....
....
....
        PARAMETERS
            CODE          ASD9         ALL MEMBERS
            STEELGRD      A572-G50     ALL MEMBERS
            DefCheck      YES-Y        MEMBERS 5 TO 8          $ Check Y direction deflection
            DefLoads      YES          LOADS  'A' 'Def-F'  'Def-G'  -
                                          'Def-H'  'Def-I'         $ Deflection loads

        $  Activate loads for code check

        LOAD  LIST  'A'  'B'  'C'  'D'  'E'

        CHECK  ALL  MEMBERS
```

This page intentionally left blank.

5.2.7 Brazilian Table

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

The above tables are from Brazilian Standard, Perfil I estrutural de aco soldado por arco eletrico - Especificacao, ABNT, NBR 5884:2000. Brazilian Standard, Structural profile type I welded steel joint by electric arc - Specification, ABNT, NBR 5884:2000.

The profiles which are available in the above tables are shown on the pages which follow.

Table CS

Profile Names

CS150x25	CS300x122	CS450x154	CS550x290	CS650x409
CS150x29	CS300x131	CS450x165	CS550x345	CS650x425
CS150x31	CS300x138	CS450x175	CS550x358	CS650x437
CS150x37	CS300x149	CS450x188	CS550x368	CS650x468
CS150x45	CS350x89	CS450x198	CS550x395	CS650x484
CS200x29	CS350x93	CS450x209	CS550x407	CS650x496
CS200x34	CS350x108	CS450x216	CS550x417	CS650x525
CS200x39	CS350x112	CS450x227	CS550x441	CS650x593
CS200x41	CS350x119	CS450x236	CS550x498	CS700x389
CS200x50	CS350x128	CS450x280	CS600x250	CS700x426
CS200x61	CS350x135	CS450x291	CS600x281	CS700x441
CS250x43	CS350x144	CS450x321	CS600x294	CS700x458
CS250x49	CS350x153	CS450x331	CS600x305	CS700x471
CS250x52	CS350x161	CS500x172	CS600x318	CS700x505
CS250x63	CS350x175	CS500x195	CS600x332	CS700x522
CS250x66	CS350x182	CS500x207	CS600x377	CS700x535
CS250x76	CS350x216	CS500x221	CS600x391	CS700x567
CS250x79	CS400x106	CS500x233	CS600x402	CS700x640
CS250x84	CS400x128	CS500x253	CS600x432	CS750x417
CS250x90	CS400x137	CS500x263	CS600x446	CS750x457
CS250x95	CS400x146	CS500x312	CS600x456	CS750x473
CS250x108	CS400x155	CS500x324	CS600x483	CS750x492
CS300x62	CS400x165	CS500x333	CS600x546	CS750x506
CS300x76	CS400x176	CS500x369	CS650x305	CS750x542
CS300x92	CS400x185	CS500x378	CS650x319	CS750x560
CS300x95	CS400x201	CS550x228	CS650x330	CS750x574
CS300x102	CS400x209	CS550x257	CS650x345	CS750x608
CS300x109	CS400x248	CS550x269	CS650x361	CS750x687
CS300x115	CS450x144	CS550x279	CS650x395	

Table CVS

Profile Names

CVS150X15	CVS300X55	CVS450X177	CVS550X361	CVS700X327
CVS150X18	CVS300X66	CVS450X188	CVS550X370	CVS700X342
CVS150X20	CVS300X80	CVS450X206	CVS600X156	CVS750X284
CVS150X22	CVS300X83	CVS450X216	CVS600X190	CVS750X301
CVS150X24	CVS300X94	CVS500X123	CVS600X210	CVS750X334
CVS200X21	CVS300X100	CVS500X134	CVS600X226	CVS750X350
CVS200X24	CVS300X113	CVS500X150	CVS600X239	CVS800X288
CVS200X28	CVS350X73	CVS500X162	CVS600X278	CVS800X310
CVS200X27	CVS350X87	CVS500X180	CVS600X292	CVS800X328
CVS200X30	CVS350X98	CVS500X194	CVS600X328	CVS800X365
CVS200X36	CVS350X105	CVS500X204	CVS600X339	CVS800X382
CVS200X38	CVS350X118	CVS500X217	CVS600X369	CVS850X336
CVS200X46	CVS350X128	CVS500X238	CVS650X211	CVS850X355
CVS250X30	CVS350X136	CVS500X250	CVS650X234	CVS850X396
CVS250X33	CVS400X82	CVS500X259	CVS650X252	CVS850X414
CVS250X40	CVS400X87	CVS500X281	CVS650X266	CVS900X342
CVS250X47	CVS400X103	CVS500X317	CVS650X282	CVS900X362
CVS250X56	CVS400X116	CVS550X184	CVS650X310	CVS900X402
CVS250X64	CVS400X125	CVS550X204	CVS650X326	CVS900X422
CVS250X72	CVS400X140	CVS550X220	CVS650X351	CVS950X368
CVS300X47	CVS400X152	CVS550X232	CVS650X366	CVS950X389
CVS300X57	CVS400X162	CVS550X245	CVS650X413	CVS950X433
CVS300X67	CVS450X116	CVS500X270	CVS650X461	CVS950X454
CVS300X70	CVS450X130	CVS550X283	CVS700X214	CVS1000X394
CVS300X79	CVS450X141	CVS550X293	CVS700X232	CVS1000X416
CVS300X85	CVS450X156	CVS550X319	CVS700X278	CVS1000X464
CVS300X95	CVS450X168	CVS550X329	CVS700X293	CVS1000X486

Table VS

Profile Names

VS150x15	VS350x28	VS550x88	VS900x142	VS1400x478
VS150x18	VS350x33B	VS550x100	VS900x159	VS1500x270
VS150x20	VS350x36	VS600x81	VS900x177	VS1500x293
VS150x19	VS350x30B	VS600x95	VS900x191	VS1500x319
VS150x21	VS350x35	VS600x111	VS950x127	VS1500x339
VS200x19	VS350x39	VS600x125	VS950x146	VS1500x388
VS200x22	VS350x38	VS600x140	VS950x162	VS1500x434
VS200x25	VS350x42	VS600x152	VS950x180	VS1500x488
VS200x20	VS350x51	VS650x84	VS950x194	VS1600x328
VS200x23	VS400x28	VS650x98	VS1000x140	VS1600x348
VS200x26	VS400x32	VS650x114	VS1000x161	VS1600x398
VS250x21	VS400x35	VS650x128	VS1000x180	VS1600x444
VS250x24	VS400x30	VS650x143	VS1000x201	VS1600x498
VS250x27	VS400x34	VS650x155	VS1000x217	VS1700x338
VS250x23	VS400x38	VS700x105	VS1100x159	VS1700x358
VS250x26	VS400x32B	VS700x122	VS1100x180	VS1700x408
VS250x30	VS400x37	VS700x137	VS1100x199	VS1700x454
VS250x25	VS400x41	VS700x154	VS1100x219	VS1700x507
VS250x29	VS400x39	VS700x166	VS1100x235	VS1800x348
VS250x32	VS400x44	VS750x108	VS1200x200	VS1800x368
VS300x23	VS400x53	VS750x125	VS1200x221	VS1800x418
VS300x26	VS450x51	VS750x140	VS1200x244	VS1800x464
VS300x28	VS450x60	VS750x157	VS1200x262	VS1800x517
VS300x25	VS450x71	VS750x170	VS1200x307	VS1800x465
VS300x28B	VS450x80	VS800x111	VS1300x237	VS1800x511
VS300x31	VS450x59	VS800x129	VS1300x258	VS1800x564
VS300x27	VS450x70	VS800x143	VS1300x281	VS1900x429
VS300x31B	VS450x83	VS800x160	VS1300x299	VS1900x478
VS300x34	VS450x95	VS800x173	VS1300x344	VS1900x524
VS300x33	VS500x61	VS850x120	VS1400x260	VS1900x577
VS300x37	VS500x73	VS850x139	VS1400x283	VS2000x461
VS300x46	VS500x86	VS850x155	VS1400x309	VS2000x515
VS350x26	VS500x97	VS850x174	VS1400x329	VS2000x566
VS350x30	VS550x64	VS850x188	VS1400x378	VS2000x624
VS350x33	VS550x75	VS900x124	VS1400x424	

5.2.8 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.

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

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

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.

5.2.9 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	CIR24
CIR14	CIR26
CIR16	CIR28
CIR18	CIR30
CIR20	CIR32
CIR22	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	REC20X12	REC22X12	REC24X12	REC26X12	REC28X12
REC18X14	REC20X14	REC22X14	REC24X14	REC26X14	REC28X14

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

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

5.2.10 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:

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.

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

Parameter Name	**Default Value**	**Meaning**
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.
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.

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

Parameter Name	Default Value	Meaning
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. See the COLUMN/BEAM LINE command (Section 2.3 of Volume 2A). 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.
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.

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

Parameter Name	Default Value	Meaning
K-Factors (continued)		
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.
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).
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).

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

Parameter Name	**Default Value**	**Meaning**
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.
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.

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

Parameter Name	**Default Value**	**Meaning**
Flexural - Torsional Buckling (continued)		
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.

Table ASD9-E.1-1 (continued)

ASD9-E 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.
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 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 indicating 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 indicating 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 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.)

AISC Tables (American Institute of Steel Construction) (continued)

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

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

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

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:

CALCULATE ERROR (ESTIMATE) (BASED ON) -

$$
* \left\{ \begin{array}{l} \text{ENERGY (NORM)} \\ \text{MAX DIFFERENCE} \\ \text{DIFFERENCE FROM AVERAGE} \\ \text{PERCENT MAX DIFFERENCE} \\ \text{PERCENT DIFFERENCE FROM AVERAGE} \\ \text{NORMALIZED PERCENT MAX DIFFERENCE} \\ \text{NORMALIZED PERCENT DIFFERENCE FROM AVERAGE} \end{array} \right\} -
$$

$$
\text{(AT)} \; ^{*}\left\{ \begin{array}{l} \text{TOP} \\ \text{MIDDLE} \\ \text{BOTTOM} \end{array} \right\} \text{(SURFACES) (FOR)} \left\{ \begin{array}{l} \rightarrow \text{ALL} \\ \\ \text{ELEMENT list} \end{array} \right\}
$$

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:

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

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,

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

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

$$\left| Value_{Max} - Value_{Min} \right|$$

Difference from Average Method

$$MAX\ (\left| Value_{Max} - Value_{Avg} \right|, \left| Value_{Min} - Value_{Avg} \right|)$$

Percent Maximum Difference Method

$$\left| \frac{Value_{Max} - Value_{Min}}{Value_{Avg}} \right| \times 100\%$$

Percent Difference from Average Method

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

Normalized Percent Maximum Difference

$$\left| \frac{Value_{Max} - Value_{Min}}{Value_{VectorMax}} \right| \times 100\%$$

Normalized Percent Difference from Average Method

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

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 Output of Response Spectrum Results

The LIST RESPONSE SPECTRUM command has been modified to include the output of BASE/STORY shear output. The section is numbered below as it is currently numbered in Volume 3 of the GTSTRUDL User Reference Manual.

2.4.6.7 Output of Response Spectrum Results

General form:

```
                                      * ⎧ PARTICIPATION   (FACTORS)   ⎫
                                        ⎪ SPECTRAL  DISPLACEMENTS     ⎪
                                        ⎪ SPECTRAL  VELOCITIES        ⎪
LIST  RESPONSE  SPECTRUM  (             ⎨ SPECTRAL  ACCELERATIONS     ⎬)   -
                                        ⎪ (MODAL)  COEFFICIENTS       ⎪
                                        ⎪ ⎧BASE ⎫                     ⎪
                                        ⎪ ⎨     ⎬   (SHEAR)           ⎪
                                        ⎩ ⎩STORY⎭                     ⎭

                                      * ⎧ DISPLACEMENTS ⎫
                                        ⎪ VELOCITIES    ⎪
                                        ⎨ ACCELERATIONS ⎬ -
                                        ⎪ FORCES        ⎪
                                        ⎪ LOADS         ⎪
                                        ⎩ REACTIONS     ⎭

                                    * ⎧ ⎧ RMS       ⎫ ⎫
                                      ⎪ ⎪ ABS       ⎪ ⎪
                                      ⎪ ⎪ PRMS      ⎪ ⎪
((MODAL) COMBINATIONS                 ⎨ ⎨ CQC       ⎬ ⎬ ) (list)
                                      ⎪ ⎪ (NRC)TPM  ⎪ ⎪
                                      ⎪ ⎪ (NRC)GRP  ⎪ ⎪
                                      ⎪ ⎩ (NRC)DSM  ⎭ ⎪
                                      ⎩ ALL           ⎭
```

Elements:

list = see Section 2.4.6.5

Explanation:

This command is used to output results associated with active RESPONSE SPECTRA loading conditions (Section 2.4.4.6). The requested results must have been previously computed via the DYNAMIC ANALYSIS command (Section 2.4.5.4), the split solver commands (Section 2.4.5.5) or the COMPUTE RESPONSE SPECTRA command (Section 2.4.5.9).

Participation factors, spectral displacements, velocites, accelerations and modal coefficients are defined in Section 2.4.2.5. The participation factor output via the LIST DYNAMIC PART FACTOR command is a normalized value expressed as a percentage, while the value output via this command is the actual factor as computed in Equation 5-6 in Section 2.4.2.5.

The BASE/STORY SHEAR option lists the global X, Y, and Z components of the total response spectrum inertia forces computed for each active mode and summed over the joints specified in the list. Modal combinations are also computed and listed if specified. These results are computed and listed completely for each active response spectrum loading condition.

Finite element forces, generalized stresses and strains can be computed but cannot be output via the LIST RESPONSE SPECTRA command. However, these finite element results may be copied into static loading conditions via the CREATE PSEUDO STATIC LOADING command (Section 2.4.7) and then output via the LIST command (Section 2.1.14.4). Member section forces and stresses can be listed via a similar approach by using the CREATE PSEUDO STATIC LOAD and LIST SECTION commands (Section 2.1.14.6).

Other commands which affect LIST RESPONSE SPECTRA are as follows:

OUTPUT MODAL CONTRIBUTIONS (Section 2.4.6.3)

OUTPUT DECIMAL (Section 2.1.14.3)

OUTPUT FIELD (Section 2.1.14.3)

Examples:

(1) LIST RESPONSE SPECTRA DISPL MODAL COMBINATIONS ALL. This command outputs previously computed peak displacements for all joints and all of the modal combination techniques.

(2) COMPUTE RESPONSE SPECTRA STRESSES FORCES MODAL -
COMB ALL MEM 1 TO 10 OUTPUT MODAL CONTRIBUTIONS ON
LIST RESPONSE SPECTRA FORCES MODE COMB RMS CQC -
MEM 1 TO 10

The output generated by the LIST RESPONSE SPECTRA command consists of the modal contributions to member end forces and the RMS and CQC combinations of those contributions. Non-zero results will exist only for members 1 to 10.

(3) LIST RESPONSE SPECTRA PARTICIPATION FACTORS SPECTRAL -
DISPLACEMENTS MODAL COEFFICENT

This command lists the participation factors, spectral displacements and the modal coefficients associated with all active response spectra loadings.

(4) LIST RESPONSE SPECTRUM STORY SHEARS -
MODE COMBINATION RMS CQC JOINTS 101 TO 140

This command lists the total response spectrum inertia forces, including the RMS and CQC modal combinations, summed over joints 101 to 140. If joints 101 through 140 represent all joints above the fifth floor of a structure, then the listed results may be interpreted as the modal and total story shear applied above the fifth floor.

This page intentionally left blank.

5.3.3 FORM STATIC LOAD Command -- Automatic Generation of Static Equivalent Earthquake Loads

General form:

$$\underline{\text{FOR}}\text{M}\ \ \underline{\text{STATI}}\text{C}\ \ (\underline{\text{EAR}}\text{THQUAKE})\ \ \underline{\text{LOAD}}\ \begin{Bmatrix} 'a_{sl}' \\ i_{sl} \end{Bmatrix}\ ('\text{title}_{sl}')\ \ -$$

$$\underline{\text{FRO}}\text{M}\ \begin{Bmatrix} \underline{\text{MAS}}\text{S}\ \ [\underline{\text{X}}]\ v_x\ \ [\underline{\text{Y}}]\ v_y\ \ [\underline{\text{Z}}]\ v_z \\ \begin{Bmatrix} \rightarrow \underline{\text{RMS}} \\ \underline{\text{CQC}} \\ \underline{\text{SUM}} \end{Bmatrix}\ (\underline{\text{OF}}\ \underline{\text{RES}}\text{PONSE}\ \underline{\text{SPE}}\text{CTRUM})\ \underline{\text{LOA}}\text{D}\ \begin{Bmatrix} 'a_{RS}' \\ i_{RS} \end{Bmatrix}\ (\underline{\text{FAC}}\text{TOR}\ v_{RS}) \end{Bmatrix}$$

Elements:

$'a_{sl}'/i_{sl}$	=	alphanumeric or integer name for the generated static earthquake load. This name must be unique among all current loading names and is limited to eight characters or digits.
$'\text{title}_{sl}'$	=	optional static load title of up to 64 characters in length.
v_x	=	scaling factor for the mass load in the global X direction. v_x is taken as 0.0 by default.
v_y	=	scaling factor for the mass load in the global Y direction. v_y is taken as 0.0 by default.
v_z	=	scaling factor for the mass load in the global Z direction. v_z is taken as 0.0 by default.
$'a_{RS}'/ i_{RS}$	=	name of the response spectrum load to be used for the calculation of static earthquake load $'a_{sl}'/i_{sl}$.
v_{RS}	=	scaling factor to be applied to the response spectrum static earthquake load $'a_{sl}'/i_{sl}$.

Explanation:

The FORM STATIC LOAD command is used to compute an independent loading condition consisting of a static joint load representation of either the structural mass or a response spectrum load. The mass and response spectrum load options are described in greater detail as follows:

$\underline{\text{MAS}}\text{S}\ [\ \underline{\text{X}}\ ]\ v_x\ [\ \underline{\text{Y}}\ ]\ v_y\ [\ \underline{\text{Z}}\ ]\ v_z$

The MASS option generates an independent loading condition containing joint loads which are statically equivalent to any factored combination of the structural mass in the global X, Y, and Z directions. Because the global direction scaling factors v_x, v_y, and v_z are taken as 0.0 by default, it is necessary to specify a non-zero value for the appropriate scaling factor if joint load components are to be computed for a particular global direction.

The MASS-equivalent static joint load vector is computed by the following equation:

$$\{F_{EM}\} = [\,M\,]\,\{v_{XYZ}\}\,g \qquad \text{Eq. 5.3.3-1}$$

where,

$\{F_{EM}\}$	=	MASS-equivalent joint load vector,
$[M]$	=	system global mass matrix,
$\{v_{XYZ}\}$	=	vector of global direction scaling factors v_x, v_y, and v_z, arranged in the appropriate joint degree-of-freedom locations,
g	=	acceleration due to gravity, taken as 386.0886 inches/-second^2 by default.

According to Equation 5.3.3-1, it is necessary that the structural mass has been defined, and that, as a minimum, the PERFORM ASSEMBLY FOR DYNAMICS command (Section 2.4.5.5.1, Volume 3, GTSTRUDL User Reference Manual) has been executed prior the execution of this option.

The calculation of a MASS-equivalent static load conforms to the NEHRP guidelines for the calculation of the *uniform pattern* lateral load distribution described in Section 3.3.3.2.C of <u>NEHRP Guidelines for the Seismic Rehabilitation of Buildings (FEMA Publication 273)</u>.

$$\left\{\begin{array}{l} \rightarrow\ \underline{\text{RMS}} \\ \quad\ \underline{\text{CQC}} \\ \quad\ \underline{\text{SUM}} \end{array}\right\} (\underline{\text{OF}}\ \ \underline{\text{RES}}\text{PONSE}\ \ \underline{\text{SPE}}\text{CTRUM})\ \ \underline{\text{LOA}}\text{D} \left\{\begin{array}{c} 'a_{RS}' \\ \\ i_{RS} \end{array}\right\} (\underline{\text{FAC}}\text{TOR}\ \ v_{RS})$$

The RESPONSE SPECTRUM LOAD option generates an independent loading condition consisting of joint loads that represent a measure of the total base shear computed for the response spectrum load ‘a_{RS}’/i_{RS}.

The additional RMS, CQC, and SUM options provide for the selection of the modal combination method to be used for the computation of the global joint loads from the modal joint load components. RMS and CQC indicate the Root Mean Square and Complete Quadratic Combination methods, respectively, as described in Section 2.4.2.5, Volume 3 of the GTSTRUDL User Reference Manual. The SUM option indicates a direct algebraic summation of the modal joint load components.

The equivalent response spectrum static joint loading in the i^{th} active mode is computed by the following equation:

$$\{ f_{RS}\}_i = - v_{RS}\, \Gamma_i\, S_{ai}\, [\, M\,]\, \{\Phi_i \} \qquad \text{Eq. 5.3.3-2}$$

where,

$\{f_{RS}\}_i$	=	response spectrum static joint load vector for the i^{th} mode,
v_{RS}	=	scaling factor as defined above,
[M]	=	the global system mass matrix,
Γ_i	=	the response spectrum participation factor for the i^{th} mode,
S_{ai}	=	the response spectrum spectral acceleration for the i_{th} mode,
$\{\Phi_i \}$	=	mode shape displacement vector for the i^{th} mode.

The total response spectrum static joint load vector is computed by combining the $\{f_{RS}\}_i$ for each active mode using the selected RMS, CQC, or SUM procedure.

Because the values for Γ_i and S_{ai} are determined from the direction and response spectrum data of response spectrum load 'a_{RS}'/i_{RS}, a response spectrum analysis for this load must have been performed prior to the execution of FORM STATIC LOAD RESPONSE SPECTRUM option. However, a COMPUTE RESPONSE SPECTRUM command execution subsequent to the response spectrum analysis is not required.

The calculation of a response spectrum static load conforms to the NEHRP guidelines for the calculation of the *modal pattern* lateral load distribution using a Response Spectrum Analysis as described in Section 3.3.3.2.C of NEHRP Guidelines for the Seismic Rehabilitation of Buildings (FEMA Publication 273).

The independent loading conditions generated by the FORM STATIC LOAD command are conventional independent static loading conditions, and as such, may be used and manipulated in the same manner as independent loads defined by other means.

Errors:

The following messages indicate error or warning conditions that can occur during the execution of the FORM STATIC LOAD command:

```
**** ERROR_STGELL -- System mass matrix does not exist.
                     SCAN MODE entered.
```

This message indicates that the mass matrix had not been assembled prior to the execution of the FORM STATIC LOAD command. SCAN MODE is set and may be removed by giving the SCAN OFF command. The minimum requirement for the MASS option is that the PERFORM ASSEMBLY FOR DYNAMICS command must be executed.

```
**** ERROR_STGELL --  Specified response spectrum loading 1-G.2 does not
                      exist.  SCAN MODE entered.
```

This error message indicates that the specified response spectrum load has not been defined. SCAN MODE is set and may be removed by giving the SCAN OFF command.

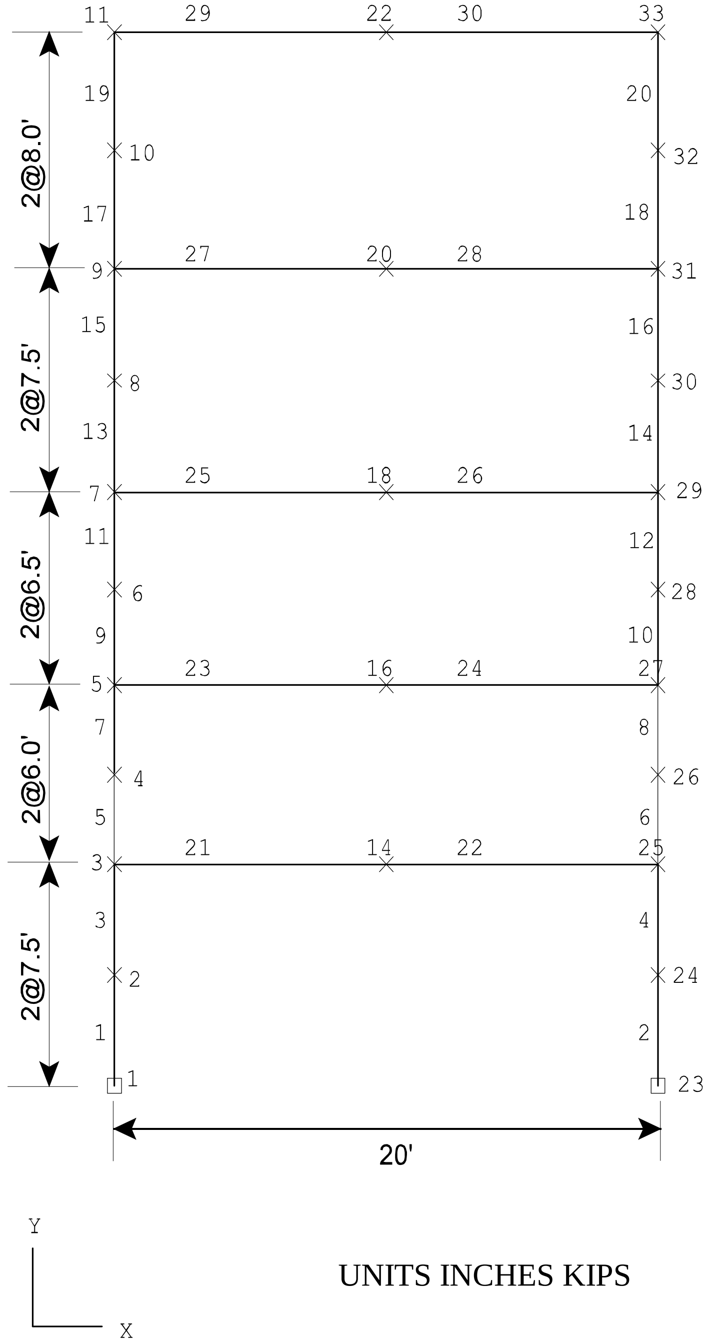
```
**** ERROR_STGELL -- Results do not exist for response spectrum loading
                     1-G.2.  Response spectrum analysis has not yet been
                     run or the specified loading is not a response
                     spectrum load.  SCAN MODE entered.
```

This error message indicates that while the specified response spectrum load is valid, the required response spectrum analysis for this load has not yet been executed. SCAN MODE is set and may be removed by giving the SCAN OFF command.

Example:

Figure 5.3.3-1 shows the plane frame structure of example SEL-1 which illustrates the use of the FORM STATIC LOAD command to create two static lateral loads based on the structural mass and on a response spectrum load. Note that the structure model includes mid-member joints to insure that the effects of fundamental member modes are not overlooked in the response spectrum analysis. The effects of such modes may arise due to the presence of the added joint inertia at joint 14.

The complete command input for this example is shown in Figure 5.3.3-2.



UNITS INCHES KIPS

Density = 0.00615

AX 33.6	IZ 2000.0	Members 1 to 4
AX 33.6	IZ 1000.0	Members 5 to 20
AX 240.0	IZ 18000.0	Members 21 to 30

Figure 5.3.3-1 Example SEL-1, Geometry, Structural, and Material Properties

```
STRUDL 'SEL-1' 'Example of static earthquake load generation'
$
$ 4-story plane frame for static earthquake load generation.
$ Geometry.
$
PRINT GEN OFF
UNITS FEET
GEN 11 JOINTS ID 1 1 X 0.0 Y DIFF 0.0 2 AT 7.5 2 AT 6.0 2 AT 6.5 2 AT 7.5 2 AT 8.0
REPEAT 2 TIMES ID 11 X 20.0
TYPE PLANE FRAME
GEN 10 MEMB ID 1 2 FROM 1 1 TO 2 1
REPEAT 1 TIME ID 1 FROM 22 TO 22
GEN 2 MEMB ID 21 1 FROM 3 11 TO 14 11
REPEAT 4 TIMES ID 2 FROM 2 TO 2

DELETIONS
JOINTS 12 13 TO 21 BY 2
ADDITIONS

STATUS SUPPORTS 1 23

$
$ Structural and material properties.
$
UNITS INCHES KIPS
MEMBER PROP PRISMATIC
1 TO 4 AX 33.6 IZ 2000.0
5 TO 20 AX 33.6 IZ 1000.0
21 TO 30 AX 240 IZ 18000

CONSTANTS
E 3000000. ALL
DENSITY 0.00615

$
$ Lumped mass plus added joint masses.
$
INERTIA OF JOINTS LUMPED
INERTIA OF JOINTS MASS
 14 TRANSL X 2.0 Y 2.0

$
$ For the static response spectrum load generation, use a
$ constant 1-g acceleration spectrum.
$
```

Figure 5.3.3-2 Command Listing for Example SEL-1

```
UNITS INCHES SECONDS
STORE RESPONSE SPECTRA ACCEL LIN VS NAT FREQ LIN 'ONE-G'
DAMPING 0.05 FACTOR 386.0886
  1.0 0.0   1.0 10000.0

RESPONSE SPECTRA LOAD '1-G.1'
SUPPORT ACCELERATIONS
TRANS X FILE 'ONE-G'
END OF RESPONSE SPECTRA LOADING

DAMPING RATIOS 0.05 100

$
$ Perform eigenvalue analysis and response spectrum analysis.
$

UNITS CYCLES SECS
EIGENPROBLEM PARAMETERS
  PRINT MAX
END

ASSEMBLE FOR DYNAMICS
PERFORM EIGENVALUE ANALYSIS
LIST DYNAMIC PARTICIPATION FACTORS

LOAD LIST '1-G.1'
PERF RESPONSE SPECTRUM ANALYSIS

$
$ Generate the static earthquake loads and print the joint load contents.
$
FORM STATIC EARTHQUAKE LOAD 'ERS1-G.1'  -
    'Equivalent STATIC EARTHQUAKE load 1-G.1, RS load 1-G.1'  -
    FROM RMS OF RESPONSE SPECTRUM LOAD '1-G.1'
FORM STATIC EARTHQUAKE LOAD 'EM1-G.1'   -
    'Equivalent STATIC EARTHQUAKE load 1-G.1 from total mass'  FROM MASS X 1.0

PRINT APPLIED JOINT LOADS
$

FINISH
```

Figure 5.3.3-2 Command Listing for Example SEL-1 (Continued)

Figure 5.3.3-3 contains the text output from the PRINT APPLIED JOINT LOADS command, showing the joint load contents of the loads generated by the FORM STATIC EARTHQUAKE LOAD examples.

```
{   78} > $
{   79} > $ Generate the static earthquake loads and print the joint load contents.
{   80} > $
{   81} > FORM STATIC EARTHQUAKE LOAD 'ERS1-G.1' 'Equivalent STATIC EARTHQUAKE load 1-G.1, RS load 1-G.1' -
{   82} >_  FROM RMS OF RESPONSE SPECTRUM LOAD '1-G.1'

Time to create equivalent static earthquake load =    0.00 Seconds

{   83} > FORM STATIC EARTHQUAKE LOAD 'EM1-G.1' 'Equivalent STATIC EARTHQUAKE load 1-G.1 from total mass' -
{   84} >_  FROM MASS X 1.0

Time to create equivalent static earthquake load =    0.00 Seconds

{   85} >
{   86} > PRINT APPLIED JOINT LOADS

****************************************
*  PROBLEM DATA FROM INTERNAL STORAGE  *
****************************************

JOB ID - SEL-1        JOB TITLE - Example of static earthquake load generation

ACTIVE UNITS -  LENGTH        WEIGHT        ANGLE       TEMPERATURE      TIME
                  INCH          KIP           CYC           DEGF          SEC

         ********** LOADING DATA **********

LOADING - ERS1-G.1     Equivalent STATIC EARTHQUAKE load 1-G.1, RS load 1-G.1              STATUS - ACTIVE

JOINT LOADS-------------------------------------------------------------------------------/

JOINT     STEP   FORCE X          Y           Z      MOMENT X          Y           Z

1                    0.000       0.000       0.000         0.000       0.000       0.000
2                    9.713       0.189       0.000         0.000       0.000       0.000
3                  104.472       3.722       0.000         0.000       0.000       0.017
4                   10.172       0.239       0.000         0.000       0.000       0.000
5                  151.872       4.293       0.000         0.000       0.000       0.006
6                   14.246       0.373       0.000         0.000       0.000       0.000
7                  192.343       5.818       0.000         0.000       0.000       0.006
8                   19.907       0.514       0.000         0.000       0.000       0.001
9                  240.328       6.715       0.000         0.000       0.000       0.006
10                  26.654       0.624       0.000         0.000       0.000       0.001
11                 288.704       6.211       0.000         0.000       0.000       0.004
14                 608.142       0.000       0.000         0.000       0.000       0.018
16                 279.441       0.000       0.000         0.000       0.000       0.005
18                 350.463       0.000       0.000         0.000       0.000       0.005
20                 433.694       0.000       0.000         0.000       0.000       0.004
22                 546.897       0.000       0.000         0.000       0.000       0.002
23                   0.000       0.000       0.000         0.000       0.000       0.000
24                   9.713       0.189       0.000         0.000       0.000       0.000
```

Figure 5.3.3-3 PRINT APPLIED JOINT LOAD Results for Example SEL-1

25	104.472	3.722	0.000	0.000	0.000	0.017
26	10.172	0.239	0.000	0.000	0.000	0.000
27	151.872	4.293	0.000	0.000	0.000	0.006
28	14.246	0.373	0.000	0.000	0.000	0.000
29	192.343	5.818	0.000	0.000	0.000	0.006
30	19.907	0.514	0.000	0.000	0.000	0.001
31	240.328	6.715	0.000	0.000	0.000	0.006
32	26.654	0.624	0.000	0.000	0.000	0.001
33	288.704	6.211	0.000	0.000	0.000	0.004

LOADING - EM1-G.1 Equivalent STATIC EARTHQUAKE load 1-G.1 from total mass STATUS - ACTIVE

JOINT LOADS--/

JOINT	STEP	FORCE X	Y	Z	MOMENT X	Y	Z
1		0.000	0.000	0.000	0.000	0.000	0.000
2		18.598	0.000	0.000	0.000	0.000	0.000
3		193.858	0.000	0.000	0.000	0.000	0.000
4		14.878	0.000	0.000	0.000	0.000	0.000
5		192.618	0.000	0.000	0.000	0.000	0.000
6		16.118	0.000	0.000	0.000	0.000	0.000
7		194.478	0.000	0.000	0.000	0.000	0.000
8		18.598	0.000	0.000	0.000	0.000	0.000
9		196.338	0.000	0.000	0.000	0.000	0.000
10		19.837	0.000	0.000	0.000	0.000	0.000
11		187.039	0.000	0.000	0.000	0.000	0.000
14		1126.417	0.000	0.000	0.000	0.000	0.000
16		354.240	0.000	0.000	0.000	0.000	0.000
18		354.240	0.000	0.000	0.000	0.000	0.000
20		354.240	0.000	0.000	0.000	0.000	0.000
22		354.240	0.000	0.000	0.000	0.000	0.000
23		0.000	0.000	0.000	0.000	0.000	0.000
24		18.598	0.000	0.000	0.000	0.000	0.000
25		193.858	0.000	0.000	0.000	0.000	0.000
26		14.878	0.000	0.000	0.000	0.000	0.000
27		192.618	0.000	0.000	0.000	0.000	0.000
28		16.118	0.000	0.000	0.000	0.000	0.000
29		194.478	0.000	0.000	0.000	0.000	0.000
30		18.598	0.000	0.000	0.000	0.000	0.000
31		196.338	0.000	0.000	0.000	0.000	0.000
32		19.837	0.000	0.000	0.000	0.000	0.000
33		187.039	0.000	0.000	0.000	0.000	0.000

```
****************************************
*  END OF DATA FROM  INTERNAL STORAGE  *
****************************************
```

Figure 5.3.3-3 PRINT APPLIED JOINT LOAD Results for Example SEL-1 (Continued)

5.3.4 FORM UBC97 LOAD Command -- Automatic Generation of Seismic Loads According to 1997 UBC

General form:

$$\underline{\text{FOR}}\text{M}\ \ \underline{\text{UBC97}}\ \ (\underline{\text{STAT}}\text{IC})\ \ (\underline{\text{SEI}}\text{SMIC})\ \ \underline{\text{LOA}}\text{D} \begin{Bmatrix} '\text{a}_{sl}' \\ \text{i}_{sl} \end{Bmatrix} \ ('\text{title}_{sl}')\ \ -$$

$$\underline{\text{DIR}}\text{ECTION} \begin{Bmatrix} \rightarrow \underline{\text{X}} \\ \underline{\text{Y}} \\ \underline{\text{Z}} \end{Bmatrix}\ \ \underline{\text{HEI}}\text{GHT}\ \ (\underline{\text{DIR}}\text{ECTION}) \begin{Bmatrix} \underline{\text{X}} \\ \rightarrow \underline{\text{Y}} \\ \underline{\text{Z}} \end{Bmatrix}\ \ \underline{\text{WEI}}\text{GHT}\ \ (\underline{\text{LOA}}\text{D}) \begin{Bmatrix} '\text{a}_{w}' \\ \text{i}_{w} \end{Bmatrix}\ \ -$$

$$\underline{\text{ZONE}} \begin{Bmatrix} \underline{1} \\ \underline{\text{2A}} \\ \underline{\text{2B}} \\ \underline{3} \\ \underline{4} \\ \underline{\text{Z}}\ \text{v}_{Z} \end{Bmatrix}\ {}^{*}\begin{Bmatrix} \underline{\text{SOI}}\text{L} \begin{Bmatrix} \underline{\text{SA}} \\ \underline{\text{SB}} \\ \underline{\text{SC}} \\ \underline{\text{SD}} \\ \underline{\text{SE}} \\ \underline{\text{SF}} \end{Bmatrix} \\ \underline{\text{CA}}\ \ \text{v}_{CA} \\ \underline{\text{CV}}\ \ \text{v}_{CV} \\ \underline{\text{NA}}\ \ \text{v}_{NA} \\ \underline{\text{NV}}\ \ \text{v}_{NV} \end{Bmatrix}\ \ \underline{\text{OCC}}\text{UPANCY}\ (\underline{\text{CAT}}\text{EGORY}) \begin{Bmatrix} \underline{\text{ESS}}\text{ENTIAL}\ \ (\underline{\text{FAC}}\text{ILITIES}) \\ \underline{\text{HAZ}}\text{ARDOUS}\ \ (\underline{\text{FAC}}\text{ILITIES}) \\ \underline{\text{SPE}}\text{CIAL}\ \ (\underline{\text{STR}}\text{UCTURES}) \\ \underline{\text{STAN}}\text{DARD}\ \ (\underline{\text{STR}}\text{UCTURES}) \\ \underline{\text{MISC}}\text{ELLANEOUS}\ \ (\underline{\text{STR}}\text{UCTURES}) \\ \underline{\text{I}}\ \ \text{v}_{I} \end{Bmatrix}\ \ -$$

$$\underline{\text{CT}} \begin{Bmatrix} \underline{\text{SMR}} \\ \underline{\text{RCMR}} \\ \underline{\text{OTHER}} \\ \text{v}_{CT} \end{Bmatrix}\ \ \underline{\text{R}}\ \ \text{v}_{R}\ \ \left((\underline{\text{WIT}}\text{H})\ \underline{\text{TOR}}\text{SION} \begin{Bmatrix} \rightarrow \underline{\text{PLU}}\text{S} \\ \underline{\text{MIN}}\text{US} \end{Bmatrix} \right)\ \ (\underline{\text{FLO}}\text{OR}\ \ (\underline{\text{TOL}}\text{ERANCE})\ \ \text{v}_{TOL})$$

Elements:

$'a_{sl}'/i_{sl}$ = alphanumeric or integer name for the generated UBC 1997 static seismic load. This name must be unique among all current loading names and is limited to eight characters or digits.

$'title_{sl}'$ = optional static load title of up to 64 characters in length.

$'a_{w}'/i_{w}$ = alphanumeric or integer name of the independent loading that is used for the calculation of the weight distribution of the structure.

v_Z = decimal value for the UBC 1997 seismic zone factor Z. This specified value supersedes the calculated value based on the seismic zone specified by the ZONE option.

v_{CA} = decimal value for the UBC 1997 seismic coefficient C_a. This specified value supersedes the calculated value based on the seismic zone and soil profiles specified by the ZONE and SOIL options.

v_{CV} = decimal value for the UBC 1997 seismic coefficient C_v. This specified value supersedes the calculated value based on the seismic zone and soil profiles specified by the ZONE and SOIL options.

v_{NA} = decimal value for the UBC 1997 near-source factor N_a used in the calculation of C_a when seismic zone 4 is selected using the ZONE option. The default value is taken as 1.0.

v_{NV} = decimal value for the UBC 1997 near-source factor N_v used in the calculation of C_v when seismic zone 4 is selected using the ZONE option. The default value is taken as 1.0.

v_I = decimal value for the UBC 1997 importance factor I, to be used instead of the calculated value based on the selection of one of the OCCUPANCE CATAGORY options: ESSENTIAL, HAZARDOUS, SPECIAL, STANDARD, or MISCELLANEOUS.

v_{CT} = decimal value for the UBC 1997 numerical coefficient C_t, to be used instead of the calculated value based on the selection of one of the CT options: SMR, RCMR, or OTHER.

v_R = decimal value for the UBC 1997 numerical coefficient R.

v_{TOL} = All active joints within this specified HEIGHT DIRECTION tolerance of one another are assumed to define the geometry of a single floor. The input of this value assumes length units. The default value for v_{TOL} is taken as 6.0 inches (15.24 cm).

Explanation:

The FORM UBC97 LOAD command is used to compute an independent loading condition consisting only of static joint loads in accordance with the provisions of Sections 1630.2, 1630.5 and 1630.6 of the 1997 Uniform Building Code, Vol.2 [ref 97]. A very important aspect regarding the execution of this command is that all load computations are performed only on the currently active joints.

The options used to define this loading condition are described as follows:

$$\underline{\text{DIR}}\text{ECTION} \left\{ \begin{array}{r} \rightarrow \underline{\text{X}} \\ \underline{\text{Y}} \\ \underline{\text{Z}} \end{array} \right\}$$

The DIRECTION option is used to specify the global coordinate direction of computed joint load components contained in the loading condition. The global X direction is the default.

$$\underline{\text{HEI}}\text{GHT} \quad (\underline{\text{DIR}}\text{ECTION}) \left\{ \begin{array}{r} \underline{\text{X}} \\ \rightarrow \underline{\text{Y}} \\ \underline{\text{Z}} \end{array} \right\}$$

The HEIGHT DIRECTION option specifies the global coordinate direction that defines the elevation coordinate for the structure. For example, HEIGHT DIRECTION Y specifies that the height of the building structure and the building floor elevations are defined with respect to the global Y axis. It is assumed that elevations are measured from 0 at the joint having the least HEIGHT DIRECTION joint coordinate value to the full height of the structure at the joint having the largest HEIGHT DIRECTION joint coordinate value.

$$\underline{\text{WEI}}\text{GHT} \ \ (\underline{\text{LOA}}\text{D}) \left\{ \begin{array}{c} 'a_w' \\ i_w \end{array} \right\}$$

The required WEIGHT LOAD command option is used to identify the static, independent loading from which the total weight of the building model is computed. The total weight is computed as the sum of the absolute values of all translation load components (FORCE X, FORCE Y, and FORCE Z). The specified loading must have been defined previous to the time that the FORM UBC97 command is given.

$$\underline{\text{ZON}}\text{E} \begin{Bmatrix} \underline{1} \\ \underline{2A} \\ \underline{2B} \\ \underline{3} \\ \underline{4} \\ \underline{Z}\ \ v_Z \end{Bmatrix}$$

This optional command is used to specify data about the relevant earthquake zone for the load calculations. Use the ZONE option to select zone 1, 2A, 2B, 3, 4, or to directly specify a value v_Z for the seismic zone factor Z. One of the ZONE options 1 through 4 must be selected if one of the SOIL options SA through SF is also selected as described below.

$$* \begin{Bmatrix} \underline{\text{SOIL}} \begin{Bmatrix} \underline{\text{SA}} \\ \underline{\text{SB}} \\ \underline{\text{SC}} \\ \underline{\text{SD}} \\ \underline{\text{SE}} \\ \underline{\text{SF}} \end{Bmatrix} \\ \\ \underline{\text{CA}}\ \ v_{CA} \\ \underline{\text{CV}}\ \ v_{CV} \\ \underline{\text{NA}}\ \ v_{NA} \\ \underline{\text{NV}}\ \ v_{NV} \end{Bmatrix}$$

These options are used to specify data about the seismic zone soil conditions. The SOIL option is used to specify the soil type: SA = S_A, SB = S_B, SC = S_C, SD = S_D, SE = S_E, and SF = S_F. The selected SOIL specification – SA through SF – is used in combination of the selected seismic ZONE specification – 1 through 4 – to calculate the values for seismic coefficients C_a and C_v according to 1997 UBC Tables 16-Q and 16-R. Any values specified using the CA and CV parameters supersede the values computed from the ZONE and SOIL specifications. Values for CA and CV must be specified if the ZONE and/or SOIL specifications are not given. The values for NA and NV are taken as 1.0 unless otherwise specified. These values are the values for the near-source fractors N_a and N_v respectively, which are used in the calculation of C_a and C_v respectively when the ZONE 4 specification above is given.

$$\underline{\text{OCC}}\text{UPANCY}\ (\underline{\text{CAT}}\text{EGORY})\ \left\{\begin{array}{l} \underline{\text{ESS}}\text{ENTIAL}\ \ (\underline{\text{FAC}}\text{ILITIES}) \\ \underline{\text{HAZ}}\text{ARDOUS}\ \ (\underline{\text{FAC}}\text{ILITIES}) \\ \underline{\text{SPE}}\text{CIAL}\ \ (\underline{\text{STR}}\text{UCTURES}) \\ \underline{\text{STAN}}\text{DARD}\ \ (\underline{\text{STR}}\text{UCTURES}) \\ \underline{\text{MISC}}\text{ELLANEOUS}\ \ (\underline{\text{STR}}\text{UCTURES}) \\ \underline{\text{I}}\ v_I \end{array}\right\}$$

The seismic importance factor I is calculated based on the OCCUPANCY CATAGORY specified using this required option. The value I is calculated automatically by specifying one of the categories ESSENTIAL, HAZARDOUS, SPECIAL, STANDARD, or MISCELLANEOUS. A value for I can also be specified directly by giving the value v_I.

$$\underline{\text{CT}}\ \left\{\begin{array}{l} \underline{\text{SMR}} \\ \underline{\text{RCMR}} \\ \underline{\text{OTH}}\text{ER} \\ v_{CT} \end{array}\right\}$$

The numerical coefficient C_t used for the calculation of the structure period according the 1997 UBC Section 1630.2.2, Method A is calculated using the specifications of this required option. The value for C_t is calculated automatically by specifying one of the categories SMR, RCMR, or OTHER. SMR stands for Steel Moment-Resting frames, RCMR stands for Reinforced Concrete Moment-Resting frames and eccentrically braced frames, and OTHER stands for all OTHER buildings. The value C_t can also be specified directly by giving a value for v_{CT}.

$\underline{\text{R}}\ v_R$

This required option is used to specify the over-strength/ductility factor R according to 1997 UBC Tables 16-N or 16-P.

(WITH) TORSION $\left\{\begin{array}{l}\rightarrow \text{PLUS} \\ \text{MINUS}\end{array}\right\}$

The optional WITH TORSION specification is used to specify that the calculation of the UBC97 joint loads shall include the effects of the rigid diaphram, torsional mass displacement described in 1997 UBC Sections 1630.6 and 1630.7. The default PLUS option indicates that each floor mass distribution for the load computation shall reflect the shift from the floor center of mass in a positive global direction perpendicular to the global load-application direction specified by the DIRECTION option. The MINUS option indicates that each floor mass distribution shall reflect the shift from the floor center of mass in a negative global direction perpendicular to the global load-application direction.

FLOOR (TOLERANCE) v_{TOL}

The optional FLOOR TOLERANCE specification is used to specify the elevation neighborhood within which groups of joints comprise a particular floor for the calculation of the UBC97 load. All joints whose elevation is within the specified tolerance are assumed to comprise a floor. The value for v_{TOL} must reflect active length units and is taken as 6 inches by default.

Because the UBC97 load computations are performed only for the active joints, any joints that shall not be considered as part of any floor can be inactivated prior to issuing the FORM UBC97 command and then re-activated following the command:

```
INACTIVE JOINTS...
FORM UBC97 LOAD...
ACTIVE JOINTS ALL
```

Errors:

The following messages indicate warning conditions that can occur during the execution of the FORM UBC97 LOAD command:

1. The following message is produced if the specified weight loading does not exist.

```
**** WARNING_STUBC9 -- Specified WEIGHT loading DL1      does not
                       exist.  Command ignored.
```

2. One or more of the following three messages are produced if seismic zone and soil profile data are not correctly specified:

```
**** WARNING_STUBC9 -- Value for seismic coefficient Cv incorrectly
                       specified.  Command ignored.

**** WARNING_STUBC9 -- Value for seismic coefficient Ca incorrectly
                       specified.  Command ignored.

**** WARNING_STUBC9 -- Value for seismic zone factor Z incorrectly
                       specified.  Command ignored.
```

The following is an example of a FORM UBC97 command that will produce these warning messages:

```
FORM UBC97 LOAD 'TESTUBCZ' DIR X WEIGHT LOAD 'DL' -
     SOIL SC CT SMR OCC I 1.5 R 5.5
```

While the SOIL SC specification is given, the ZONE/Z specification is missing, resulting in insufficient data for the calculation of the seismic coefficients C_a and C_v.

3. The following message is produced if the required occupancy data are incorrectly specified.

```
**** WARNING_STUBC9 -- Value for importance factor I incorrectly
                       specified.  Command ignored.
```

The following is an example of a FORM UBC97 command that will produce this warning message:

```
FORM UBC97 LOAD 'TESTUBCZ' DIR X WEIGHT LOAD 'DL' -
   CA 0.24 CV 0.32 CT SMR R 5.5
```

The required OCCUPANCY option is not given.

4. This warning message is given if a value for the over-strength/ductility factor R is not specified using the required R command option:

```
**** WARNING_STUBC9 -- Value for R factor incorrectly specified.
                       Command ignored.
```

5. The following warning message is given if the value for the numerical coefficient C_t is not correctly specified using the required CT command option:

```
**** WARNING_STUBC9 -- Value for period coefficient Ct incorrectly
                       specified.  Command ignored.
```

Example:

The following UBC97 command example correctly defines a 1997 UBC static lateral load having the name UBCX:

```
FORM UBC97 LOAD 'UBCX' DIR X WEIGHT LOAD 'DL' -
     SOIL SC ZONE 2B  CT SMR OCC I 1.5 R 5.5
```

The load is calculated using the following 1997 UBC parameter values:

C_t	=	0.035
Z	=	0.2
I	=	1.5
R	=	5.5
C_a	=	0.24
C_v	=	0.32
N_a	=	1.0
N_v	=	1.0

No torsion effects are included in the computation of loading UBCX.

The following example is the same as the previous one, with the exception that the CV 0.35 parameter is added following the ZONE 2B option. The TORSION option is also added:

```
FORM UBC97 LOAD 'UBCXNEW' DIR X WEIGHT LOAD 'DL' -
     SOIL SC ZONE 2B CV 0.35 CT SMR OCC I 1.5 R 5.5
     TORSION MINUS
```

The 1997 UBC parameter values that are used to calculate loading UBCXNEW are the same as those that are used to calculate load UBCX in the previous example, with the exception that C_v = 0.35 rather than C_v = 0.32, by virtue of the fact that this is the value that is directly specified for C_v using the CV 0.35 parameter specification. The specified value for C_v takes precedence over the value that is computed according to the 1997 UBC provisions. Loading UBCXNEW also reflects a negative global Z offset of the floor masses with respect to each floor center of mass.

This page intentionally left blank.

5.3.5 FORM IS1893 LOAD Command - Automatic Generation of Static Seismic Loads According to IS 1893

General Form:

$$\underline{\text{FOR}}\text{M}\ \ \underline{\text{IS1893}}\ \ (\underline{\text{STAT}}\text{IC})\ \ (\underline{\text{SEI}}\text{SMIC})\ \ \underline{\text{LOAD}}\ \ \left\{\begin{matrix}'a_{sl}'\\ i_{sl}\end{matrix}\right\}\ \ ('\text{title}_{sl}')\ \ -$$

$$\underline{\text{DIR}}\text{ECTION}\ \left\{\begin{matrix}\rightarrow \underline{\text{X}}\\ \underline{\text{Y}}\\ \underline{\text{Z}}\end{matrix}\right\}\ \ \underline{\text{HEI}}\text{GHT}\ \ \underline{\text{DIR}}\text{ECTION}\ \left\{\begin{matrix}\underline{\text{X}}\\ \rightarrow \underline{\text{Y}}\\ \underline{\text{Z}}\end{matrix}\right\}\ \ \underline{\text{WEI}}\text{GHT}\ \ (\underline{\text{LOAD}})\ \left\{\begin{matrix}'a_{W}'\\ i_{W}\end{matrix}\right\}\ \ -$$

$$\underline{\text{ZON}}\text{E}\ \left\{\begin{matrix}\underline{\text{II}}\\ \underline{\text{III}}\\ \underline{\text{IV}}\\ \underline{\text{V}}\\ \underline{\text{Z}}\ v_Z\end{matrix}\right\}\ \ \underline{\text{SOI}}\text{L}\ \ (\underline{\text{TYP}}\text{E})\ \left\{\begin{matrix}\underline{\text{I}}\\ \underline{\text{II}}\\ \underline{\text{III}}\end{matrix}\right\}\ \ \underline{\text{IMP}}\text{ORTANCE}\ \ (\underline{\text{FAC}}\text{TOR})\ v_I\ \ -$$

$$\underline{\text{PER}}\text{IOD}\ \ (\underline{\text{TYP}}\text{E})\ \left\{\begin{matrix}\underline{\text{SMR}}\\ \underline{\text{RCMR}}\\ \underline{\text{OTH}}\text{ER}\\ v_T\end{matrix}\right\}\ \ \underline{\text{R}}\ v_R\ \ \ (\underline{\text{FLO}}\text{OR}\ \ (\underline{\text{TOL}}\text{ERANCE})\ \ v_{TOL})$$

Elements:

$'a_{s1}'/i_{s1}$ = alphanumeric or integer name for the generated IS 1893 static seismic load. This name must be unique among all current loading names and is limited to eight characters or digits.

$'\text{title}_{s1}'$ = optional static load title of up to 64 characters in length.

$'a_W'/i_W$ = alphanumeric or integer name of the independent loading that is used for the calculation of the weight distribution of the structure.

v_Z = decimal value for the IS 1893 seismic zone factor Z. This specified value supersedes the calculated value based on the seismic zone specified by the ZONE option.

v_I = decimal value for the IS 1893 importance factor I.

v_T = decimal value for the building fundamental period in seconds to be used for the calculation of the average response acceleration coefficient S_a/g.

v_R = decimal value for the IS 1893 response reduction factor R.

v_{TOL} = all joints within this specified HEIGHT DIRECTION tolerance of one another are assumed to define the geometry of a single floor. The input of this value assumes length units. The default value for v_{TOL} is taken as 6.0 inches (15.24 cm).

Explanation:

The FORM IS1893 LOAD command is used to compute an independent loading condition consisting of static joint loads only in accordance with the provisions of Sections 6 and 7, Indian Standard IS 1893 (Part I) : 2.2. A very important aspect regarding the execution of this command is that all computations are performed only on the currently active joints.

The options used to define this loading condition are described as follows:

$$\underline{\text{DIR}}\text{ECTION} \left\{ \begin{array}{l} \rightarrow \underline{\text{X}} \\ \quad\ \underline{\text{Y}} \\ \quad\ \underline{\text{Z}} \end{array} \right\}$$

The DIRECTION option is used to specify the global coordinate direction of the computed joint load components contained in the loading condition. The global X direction is the default.

$$\underline{\text{HEI}}\text{GHT}\ \ \underline{\text{DIR}}\text{ECTION} \left\{ \begin{array}{l} \quad\ \underline{\text{X}} \\ \rightarrow \underline{\text{Y}} \\ \quad\ \underline{\text{Z}} \end{array} \right\}$$

The HEIGHT DIRECTION option specifies the global coordinate direction that defines the elevation coordinate for the structure. For example, HEIGHT DIRECTION Y specifies that the height of the building structure and the building floor elevations are defined with respect to the global Y axis. It is assumed that elevations are defined with respect to the global Y axis. It is assumed that elevations are measured from 0 at the joint having the least HEIGHT DIRECTION joint coordinate value to the full height of the structure at the joint having the largest HEIGHT DIRECTION joint coordinate value.

$$\underline{\text{WEI}}\text{GHT}\ (\underline{\text{LOA}}\text{D})\ \left\{ \begin{array}{l} \text{'}a_w\text{'} \\ i_w \end{array} \right\}$$

The required WEIGHT LOAD command option is used to identify the static, independent loading from which the total weight of the building model is computed. The total weight is computed as the sum of the absolute values of all translation load components (FORCE X, FORCE Y, and FORCE Z). The specified loading must have been defined previous to the time that the FORM IS1893 LOAD command is given.

$$\underline{\text{ZON}}\text{E}\ \left\{ \begin{array}{l} \underline{\text{II}} \\ \underline{\text{III}} \\ \underline{\text{IV}} \\ \underline{\text{V}} \\ \underline{\text{Z}}\ v_z \end{array} \right\}$$

This required command is used to specify data about the relevant earthquake zone for the load calculations. Use the ZONE option to select zone II, III, IV, V or to directly specify a value v_z for the seismic zone factor Z.

$$\underline{\text{SOI}}\text{L}\ (\underline{\text{TYP}}\text{E})\ \left\{ \begin{array}{l} \underline{\text{I}} \\ \underline{\text{II}} \\ \underline{\text{III}} \end{array} \right\}$$

These options are used to specify data about the seismic zone soil conditions for the calculation of the average response acceleration coefficient S_a/g.

IMPORTANCE (FACTOR) v_I

This required command is used to specify the value for the seismic importance factor I.

$$\text{PERIOD (TYPE)} \begin{Bmatrix} \text{SMR} \\ \text{RCMR} \\ \text{OTHER} \\ v_T \end{Bmatrix}$$

The required PERIOD command is used to select the equation for the calculation of the approximate building natural period T_a. The SMR, RCMR and OTHER options are used to select an empirical equation to compute the building natural period, where SMT stands for Steel Moment-Resisting frames, RCMR stands for Reinforced Concrete Moment-Resisting frames and eccentrically braced frames, and OTHER stands for all OTHER buildings. The period in seconds also can be specified directly by specifying a decimal value v_T.

R v_R

This required option is used to specify the value for the response reduction factor.

FLOOR (TOLERANCE) v_{TOL}

The option FLOOR TOLERANCE specification is used to specify the elevation neighborhood within which groups of joints comprise a particular floor for the calculations of the IS1893 load. All joints whose elevation is within the specified tolerance are assumed to comprise a floor. The value for v_{TOL} must reflect active length units and is taken as 6 inches by default.

Because the IS1893 load computations are performed only for the active joints, any joints that shall not be considered as part of any floor can be inactivated prior to issuing the FORM IS1893 command and then re-activated following the command:

```
INACTIVE JOINTS ...
FORM IS1893 LOAD ...
ACTIVE JOINTS ALL
```

Errors:

The following messages indicate warning conditions that can occur during the execution of the FORM IS1893 LOAD command.

1. The following message is produced if the specified weight loading does not exist.

```
****  WARNING_STIS93 –  Specified  WEIGHT  loading DL1 does not
                        exist.  Command ignored.
```

2. The following message is produced if seismic zone data are not correctly specified:

```
****  WARNING_STIS93 –  Specified  WEIGHT  loading DL1 does not
                        exist.  Command ignored.
```

The following is an example of a FORM IS1893 command that will produce this warning message:

```
FORM IS1893 LOAD 'TESTISX' DIRE X WEIGHT LOAD 'DL'  -
       SOIL II PERIOD TYPE SMR IMP 1.5 R 4.0
```

The ZONE option is not specified in the command.

3. The following message is produced if the soil type is not correctly specified:

```
****  WARNING_STIS93 –  Soil type not correctly specified.  Command
                        ignored.
```

The following is an example of a FORM IS1893 command that will produce this warning message:

```
FORM IS1893 LOAD 'TESTISX' DIR X WEIGHT LOAD 'DL' ZONE  -
       III PERIOD TYPE SMR IMP 1.5 R 4.0
```

The SOIL TYPE option is not specified in the command.

4. The following message is produced if the importance factor I is incorrectly specified.

```
****  WARNING_STIS93 –  Specified  WEIGHT  loading DL1 does not
                        exist.  Command ignored.
```

The following is an example of a FORM IS1893 command that will produce this warning message:

```
FORM IS1893 LOAD ‘RESTISX’ DIR X WEIGHT LOAD ‘DL’ ZONE  -
       III SOIL TYPE III PERIOD TYPE SMR R 4.0
```

The required value of the importance factor I is not specified.

5. This warning message is given if a value for the response reduction factor R is not specified using the required R command option:

```
****  WARNING_STIS93 –  R factor incorrectly specified.  Command
                        ignored.
```

6. The following warning message is given if the PERIOD option is not specified.

```
****  WARNING_STIS93 –  R factor incorrectly specified.  Command
                        ignored.
```

Example

The following FORM IS1893 command example correctly defines an IS 1893 static lateral load having the name IS1893X:

```
FORM IS1893 LOAD ‘IS1893X’ DIR X WEIGHT LOAD ‘DL’  -
       SOIL TYPE II ZONE V PERIOD 1.00  -
       IMPORT 1.5  R  4.0
```

The period is directly specified as 1.0 seconds.

5.3.6 Element Properties Command for Nonlinear Hysteretic Spring Element

The Element Properties command for the hysteretic version of the nonlinear spring element is shown below and is numbered as it will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.5.3.5 The ELEMENT PROPERTIES Command for the NLS4PH Element

The complete syntax of the ELEMENT PROPERTIES command is described in Section 2.3.5.2, Volume 3 of the GTSTRUDL User Reference Manual. The properties for the NLS4PH element, a four-parameter, hysteretic version of the nonlinear spring element (type NLS), are described by the following additional ELEMENT PROPERTIES command syntax elements:

General form:

<u>ELE</u>MENT <u>PROP</u>ERTIES
list (NLS4PH specs)
.
.
.
list (NLS4PH specs)

where,

NLS4PH specs = <u>TYP</u>E 'NLS4PH' (force specs) (moment specs) (orientation specs)

$$\text{force specs} = \underline{\text{FOR}}\text{CE} \; {}^{*}\begin{Bmatrix} \underline{\text{X}}\ [\,\underline{\text{UX0}}\,]\ v_{ux0}\ [\,\underline{\text{FX0}}\,]\ v_{fx0}\ [\,\underline{\text{UXM}}\text{AX}\,]\ v_{uxm}\ [\,\underline{\text{FXM}}\text{AX}\,]\ v_{fxm} \\ \underline{\text{Y}}\ [\,\underline{\text{UY0}}\,]\ v_{uy0}\ [\,\underline{\text{FY0}}\,]\ v_{fy0}\ [\,\underline{\text{UYM}}\text{AX}\,]\ v_{uym}\ [\,\underline{\text{FYM}}\text{AX}\,]\ v_{fym} \\ \underline{\text{Z}}\ [\,\underline{\text{UZ0}}\,]\ v_{uz0}\ [\,\underline{\text{FZ0}}\,]\ v_{fz0}\ [\,\underline{\text{UZM}}\text{AX}\,]\ v_{uzm}\ [\,\underline{\text{FZM}}\text{AX}\,]\ v_{fzm} \end{Bmatrix}$$

$$\text{moment specs} = \underline{\text{MOM}}\text{ENT} \; {}^{*}\begin{Bmatrix} \underline{\text{X}}\ [\,\underline{\text{THX0}}\,]\ v_{thx0}\ [\,\underline{\text{MX0}}\,]\ v_{mx0}\ [\,\underline{\text{THXM}}\text{AX}\,]\ v_{thxm}\ [\,\underline{\text{MXM}}\text{AX}\,]\ v_{mxm} \\ \underline{\text{Y}}\ [\,\underline{\text{THY0}}\,]\ v_{thy0}\ [\,\underline{\text{MY0}}\,]\ v_{my0}\ [\,\underline{\text{THYM}}\text{AX}\,]\ v_{thym}\ [\,\underline{\text{MYM}}\text{AX}\,]\ v_{mym} \\ \underline{\text{Z}}\ [\,\underline{\text{THZ0}}\,]\ v_{thz0}\ [\,\underline{\text{MZ0}}\,]\ v_{mz0}\ [\,\underline{\text{THZM}}\text{AX}\,]\ v_{thzm}\ [\,\underline{\text{MZM}}\text{AX}\,]\ v_{mzm} \end{Bmatrix}$$

$$\text{orientation specs} = {}^{*}\begin{Bmatrix} \underline{\text{TH1}}\ v_1 \\ \underline{\text{TH2}}\ v_2 \\ \underline{\text{TH3}}\ v_3 \end{Bmatrix}$$

Elements:

v_{ux0} = displacement in the spring X direction at curve point 0,
v_{fx0} = spring X force corresponding to v_{ux0},
v_{uxm} = displacement in the spring X direction at the point of maximum displacement on the curve,
v_{fxm} = spring X force corresponding to v_{uxm},
v_{uy0} = displacement in the spring Y direction at curve point 0,
v_{fy0} = spring Y force corresponding to v_{uy0},
v_{uym} = displacement in the spring Y direction at the point of maximum displacement on the curve,
v_{fym} = spring Y force corresponding to v_{uym},
v_{uz0} = displacement in the spring Z direction at curve point 0,
v_{fz0} = spring Z force corresponding to v_{uz0},
v_{uzm} = displacement in the spring Z direction at the point of maximum displacement on the curve,
v_{fzm} = spring Z force corresponding to v_{uzm},
v_{thx0} = rotation about the spring X axis at curve point 0,
v_{mx0} = spring X axis moment corresponding to v_{thx0},
v_{thxm} = rotation about the spring X axis at the point of maximum rotation on the curve,
v_{mxm} = spring X axis moment corresponding to v_{thxm},
v_{thy0} = rotation about the spring Y axis at curve point 0,
v_{my0} = spring Y moment corresponding to v_{thy0},
v_{thym} = rotation about the spring Y axis at the point of maximum rotation on the curve,
v_{mym} = spring Y axis moment corresponding to v_{thym},
v_{thz0} = rotation about the spring Z axis at curve point 0,
v_{mz0} = spring Z axis moment corresponding to v_{thz0},
v_{thzm} = rotation about the spring Z axis at the point of maximum rotation on the curve,
v_{mzm} = spring Z axis moment corresponding to v_{thzm},
v_1, v_2, v_3
= values in current angle units of the rotation angles θ_1, θ_2, and θ_3 respectively, as shown in Figure 2.1.7-1 (Section 2.1.7.1, Volume 1), that define the rotated orientation of the spring axes with respect to the global coordinate system.

All displacement, rotation, force, and moment values assume the appropriate active length, angular, and force units.

Explanation:

The NLS4PH element is a special version of the nonlinear spring element (Section 2.5.3, Volume 3 of the GTSTRUDL User Reference Manual), where the stiffness properties are described by bilinear, symmetric, hysteretic, force-displacement and moment-rotation curves. A symmetric curve is defined by a total of three data points: the first point at the 0, 0 origin of the curve is assumed; the second and third points are defined by the specified data values.

The specified data points, for example v_{uy0}, v_{fy0}, v_{uym}, and v_{fym}, must always be positive, and the slope of the first line segment connecting point 0,0 and the point defined by the first pair of data values (e.g. v_{uy0} and v_{fy0}) must be greater than the slope of the second line segment connecting the points defined by the first pair (v_{uy0} and v_{fy0}) and the second pair (v_{uym} and v_{fym}) of values. In addition, the slope of the second line segment must be greater than or equal to zero. The symmetric negative portion of the force-displacement or moment-rotation curve is defined by taking the negative of the specified pairs of data values.

Figure 2.5.3.5-1 illustrates a force-displacement curve specified by the four data point values v_{uy0}, v_{fy0}, v_{uym}, and v_{fym}. The initial curve template and the assumed hysteretic behavior are shown.

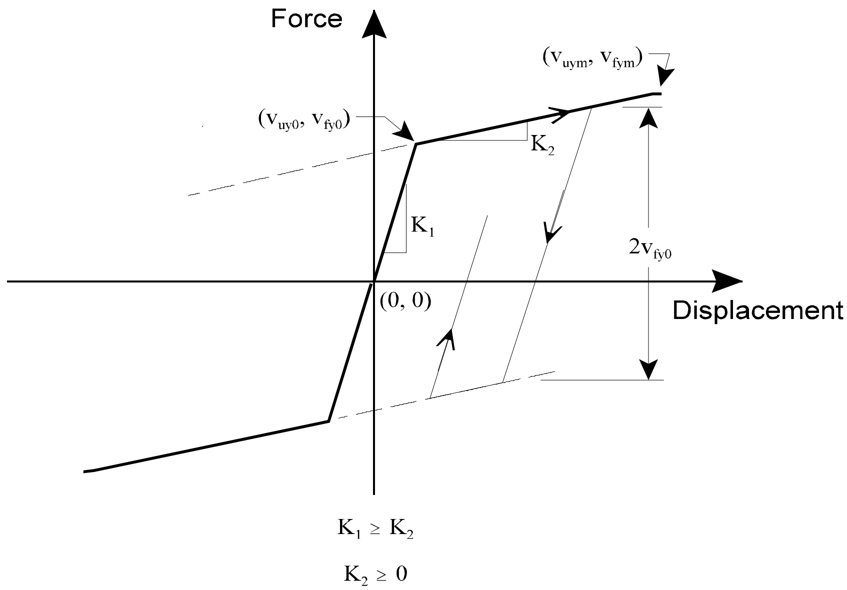


Figure 2.5.3.5-1 NLS4PH Force-Displacement Spring Curve with Hysteretic Behavior

The active degrees-of-freedom of the NLS4PH element are defined by the selection of the force/moment components for which the data point values are specified. A total of six degrees-of-freedom can be specified, and all degrees of freedom are uncoupled.

The default orientation of the degrees-of-freedom of the NLS4PH element is the global coordinate system. FORCE X means global force X, FORCE Y, global force Y, etc. This orientation can be rotated with respect to the global coordinate system by specifying values v_1, v_2, and v_3 for the angles TH1, TH2, and TH3. The sign convention for these angles is identical to that of the local release orientation angles of the JOINT RELEASE command in Section 2.1.7.1, Volume 1 of the GTSTRUDL User Reference Manual.

The NLS4PH element is applicable for both nonlinear static and dynamic analyses.

Modifications:

In CHANGES mode, existing data are replaced with the newly specified data. Unspecified data is left unchanged.

In DELETIONS mode, only the list of elements need to be specified (see Section 2.3.5.5). All properties data pertaining to the specified elements are deleted.

Example:

The following example illustrates the specification of NLS4PH properties for element 8. The specified degrees-of-freedom are force X and Moment Z. The local orientation of the spring is defined by the TH2 value of -30.0° (a negative left-hand rotation about the global Y axis).

```
UNITS DEGREES KIPS INCHES
ELEMENT PROPERTIES
    8 TYPE 'NLS4PH'  -
      FORCE X UX0 0.5 FX0 20.0 UXM 3.0 FXM 22.0  -
      MOM   Z THZ0 1.2 MZ0 650.0 THZMAX 10.0 MZMAX 1500.0 TH2 -30.0
```

Error Messages:

The following messages indicate error conditions that can occur because of the incorrect specification of the element properties data for the NLS4PH element:

```
{   63} > UNITS KIPS INCHES
{   64} > ELEMENT PROPERTIES TYPE 'NLS4PH'

**** ERROR_XYPROP – Command syntax incorrect for NLS4PH element.
                    Scan mode entered.

{   65} >   7 FORCE X UX0 0.5 FX0 20.0 UXM 3.0 FXM 22.0
CI-w-cmdmiss, ERROR: Command name missing from line.
```

These messages are general and are produced when any incorrect syntax is used in the ELEMENT PROPERTIES command for the NLS4PH element. In this case, the TYPE ‘NLS4PH’ specification in the ELEMENT PROPERTIES command header is not permitted. Scan mode is entered, precluding the execution of any time-consuming and resource-expensive operation such as analysis or the output of analysis results.

```
{   70} > ELEMENT PROPERTIES
{   71} >   7 TYPE 'NLS4PH' FORCE X UX0 0.5 FX0 -10.0 UXM 3.0 FXM
22.0

 **** ERROR_STNS4P -- All NLS4PH element property values must be
>= 0.
                      Scan mode entered.
```

This message also is given when the specified data do not conform to the requirements described above. In this example, a negative value is specified for FX0; it must be greater than 0.0. Scan mode is also entered as a result of this error condition.

```
{   87} > ELEMENT PROPERTIES
{   88} >   8 TYPE 'NLS4PH' -
{   89) >_  MOM Z THZ0 1.2 MZ0 650.0 THZMAX 10.0 MZMAX 1500.0 TH2
-30.0

 **** ERROR_STNS4P -- Properties already specified for NLS4PH
element 8
                      Scan mode entered.
```

This message is given when the NLS4PH properties for a specified element previously have been given. Scan mode is entered. CHANGES mode is required to modify existing NLS4PH properties.

5.3.7 Nonlinear Dynamic Analysis

Sections 2.5.6.1 and 2.5.6.2 below describe the commands used to execute a nonlinear dynamic analysis. Section 2.5.6.1 describes extensions to the optional DYNAMIC PARAMETERS command (Section 2.4.5.3) which are used to set nonlinear dynamic analysis control parameters. Section 2.5.6.2 describes the DYNAMIC ANALYSIS NONLINEAR command.

Section 2.5.6.3 provides an example problem illustrating a complete nonlinear dynamic analysis execution, including the use of the DYNAMIC PARAMETERS extensions and the DYNAMIC ANALYSIS NONLINEAR command. The sections are numbered as they will appear when they are added to Volume 3 of the GTSTRUDL User Reference Manual.

2.5.6.1 Extensions to the DYNAMIC PARAMETERS Command

General form:

$$\begin{array}{l}
\underline{\text{DYN}}\text{AMIC } \underline{\text{PARAM}}\text{ETERS} \\
\quad \underline{\text{BLO}}\text{CK } (\underline{\text{SIZ}}\text{E})\ i_{BS} \\
\quad \underline{\text{UPD}}\text{ATE } (\underline{\text{STI}}\text{FFNESS } \underline{\text{EV}}\text{ERY})\ i_U\ (\underline{\text{TIM}}\text{E } \underline{\text{STE}}\text{PS}) \\
\quad \underline{\text{MAX}}\text{IMUM } (\underline{\text{NUM}}\text{BER } \underline{\text{OF}}\ \underline{\text{EQ}}\text{UILIBRIUM } \underline{\text{CYC}}\text{LES})\ i_{MAX} \\
\quad \underline{\text{CONV}}\text{ERGENCE } (\underline{\text{TOL}}\text{ERANCE})\ \underline{\text{ENER}}\text{GY}\ v_{TOL} \\
\quad \underline{\text{INIT}}\text{IAL } (\underline{\text{STR}}\text{ESS } \underline{\text{LOA}}\text{D}) \left\{ \begin{array}{c} \rightarrow \underline{\text{OFF}} \\ i_{ISL} \\ 'a_{ISL}' \end{array} \right\} \\
\quad \underline{\text{RES}}\text{ULTS } (\underline{\text{FIL}}\text{E } \underline{\text{NAM}}\text{E})\ '\text{fn}' \\
\underline{\text{END}}\ (\underline{\text{O}}\text{F } \underline{\text{DYN}}\text{AMIC } \underline{\text{PAR}}\text{AMETERS})
\end{array}$$

Elements:

i_{BS} = size of equation blocks in double words if the out-of-core equation solver is to be used,

i_U = time step interval at which complete updating of the effective global tangent stiffness matrix is to occur,

i_{MAX} = maximum number of equilibrium correction iterations permitted within each time increment, (default i_{MAX} = 15),

v_{TOL} = a decimal number much less than 1.0, representing the incremental energy convergence tolerance, (default v_{TOL} = .001),

i_{ISL}/'a_{ISL}' = integer or alphanumeric id of the initial stress loading,

'fn' = alphanumeric string to be used in the construction of the dynamic analysis results file names. The length of the alphanumeric string is limited to 24 characters.

Explanation:

The DYNAMIC PARAMETERS command includes the following options which are used to control a nonlinear dynamic analysis:

BLOCK SIZE i_{BS}

By default, GTSTRUDL attempts to perform the nonlinear dynamic analysis solution entirely within the virtual memory (RAM plus page file) of the computer. If this is not possible or practical, the BLOCK SIZE parameter may be specified, indicating that an out-of-core solution procedure is to be used. The out-of-core solution process divides the equations of motion into blocks of maximum double word size i_{BS}, which are stored in scratch files. The blocks are transferred back and forth between the scratch files and virtual memory during the solution process, and the scratch files are deleted from the system when the solution is complete. The block transfer process results in longer solution times, and while the blocked solution process requires less virtual memory, sufficient hard disk space is needed for the storage of the blocks. If a value of 0 is given for i_{BS}, the solution procedure is reset to the in-core method.

UPDATE STIFFNESS EVERY i_U TIME STEPS

The nonlinear dynamic analysis is a modified Newton-Raphson process whereby the an effective dynamic stiffness matrix is updated, i.e. re-computed, only at the beginning of a time step and only at the time step interval specified by i_U. If this option is not specified or a value of 0 is specified for i_U, the effective dynamic stiffness matrix is computed at the beginning of the first time step and remains constant thereafter.

MAXIMUM NUMBER OF EQUILIBRIUM CYCLES i_{MAX}

Equilibrium correction cycles are executed for every time step. This option is used to specify the maximum permissible number of these cycles in any given time step. If a value for i_{MAX} is not specified, 15 is assumed. If the maximum number of cycles is reached and convergence has not been achieved, then the entire analysis is terminated.

CONVERGENCE TOLERANCE ENERGY v_{TOL}

The CONVERGENCE TOLERANCE ENERGY option specifies the value for the convergence tolerance used in the incremental energy convergence check. If not specified, a convergence tolerance value of 0.001 is assumed.

INITIAL STRESS LOAD OFF i_{ISL}/'a_{ISL}'

This option identifies a loading condition, i_{ISL}/'a_{ISL}', from which an initial displacement and stress state is established for the subsequent nonlinear dynamic analysis. The analysis starts with the structure in this state, including the applied loading that produced it. The initial stress load induces no acceleration at the start of the analysis, and is carried throughout the entire analysis.

The initial stress load is typically a self-weight load applied to the structure, for which a static stiffness or nonlinear analysis has been previously executed. The initial stress load also may be the prestress loading from the previous prestress analysis of a cable structure (see Section 2.6.3, Volume 3, GTSTRUDL User Reference Manual).

The INITIAL STRESS LOAD OFF command is used to switch off the inclusion of an initial stress load for a nonlinear dynamic analysis in the case where an initial stress load was specified for a previous nonlinear dynamic analysis.

RESULTS FILE NAME 'fn'

Nonlinear dynamic analysis uses the external file solver procedure described in Section 2.4.5.3 (Section 5.10 of this Release Guide), whereby the results of the dynamic analysis – displacements, velocities, accelerations, member forces, etc. – are permanently stored at the end of each time step into external save files rather than in virtual memory. The RESULTS FILE NAME option specifies an alphanumeric string to be used as a file name prefix in the creation of the results file names. If this option is not given, then the file name prefix string 'fn' is taken as the problem id given in the STRUDL command (Section 2.1.2.3, Volume 1, GTSTRUDL User Reference Manual) or the CHANGE ID command (Section 2.1.2.5, Volume 1, GTSTRUDL User Reference Manual). If a problem id is not specified in either of these two commands, then ‘fn’ is taken as ‘DyJob’.

It is recommended that the RESULTS FILE NAME command be given only once in a given job, regardless of the number of nonlinear dynamic analyses performed for different transient loading conditions. Giving a new RESULTS FILE NAME command prior to each nonlinear dynamic analysis for a new transient loading is permitted; however, doing so will make it cumbersome later to access the results from the different loading conditions. Prior to accessing the results from a particular transient loading condition, it will be necessary to re-issue the RESULTS FILE NAME command with the value of ‘fn’ initially used to create the results for that loading condition.

Modifications:

The DYNAMIC PARAMETERS command functions identically in ADDITIONS, CHANGES, and DELETIONS modes. To change the value of a dynamic parameter, simply re-specify the new value, regardless of input mode.

2.5.6.2 DYNAMIC ANALYSIS NONLINEAR Command

General form:

$$\underline{\text{DYN}}\text{AMIC } \underline{\text{ANA}}\text{LYSIS } \underline{\text{NONL}}\text{INEAR} \quad \left\{ \begin{array}{l} \underline{\text{BETA}}\ v_b \\ \underline{\text{NJP}}\ i_{NJP} \end{array} \right\}^*$$

Elements:

v_b = decimal value of β in the Newmark-β formulation of direct integration. The default value is 0.25, corresponding to the constant average acceleration method.

i_{NJP} = integer number of joints per partition.

Explanation:

The DYNAMIC ANALYSIS NONLINEAR command is used to execute a nonlinear dynamic analysis, using the Newmark-β formulation as described in Section 2.4.2.4.2 of Volume 3, GTSTRUDL User Reference Manual.

The BETA option is used to change the value of β in the Newmark-β formulation from the default value of 0.25, which corresponds to the unconditionally stable constant average acceleration method.

The NJP option is used to specify the number of joints per partition, which has only a minimal effect on the efficiency of the nonlinear dynamic analysis procedure. The default NJP value is 36/JF, where JF = the number of degrees-of-freedom associated with the type of member or element specified in the last TYPE command (Section 2.1.5.2.2, Volume 1, GTSTRUDL User Reference Manual, Section 2.3.4, Volume 3, GTSTRUDL User Reference Manual).

Nonlinear dynamic analysis supports all available TRANSIENT LOADING options.

The nonlinear dynamic analysis calculates and stores nodal displacements, velocities, accelerations, and element nodal forces for nonlinear geometric frame and truss members, tension/compression-only members, friction damper members, nonlinear spring elements, NLS4PH elements, and IPCABLE elements. Member end forces and finite element nodal forces, and finite element stresses and strains may be computed and stored for linear members and finite elements by using the COMPUTE TRANSIENT command described in Section 2.4.5.7, Volume 3, GTSTRUDL User Reference Manual.

All currently available dynamic analysis results processing functions support the results from nonlinear dynamic analysis.

Nonlinear dynamic analysis does not presently support the following capabilities:

1. Superelements

2. Joint constraints, i.e. joint ties and rigid bodies (Section 2.6.5, Volume 3, GTSTRUDL Reference Manual).

3. Dynamic degrees of freedom condensation using the DYNAMIC DEGREES OF FREEDOM command (Section 2.4.5.1, Volume 3, GTSTRUDL User Reference Manual.

If superelements are detected, the following error message is printed and the analysis is terminated:

```
**** ERROR_STDNL1 --     Superelement(s) detected.
                         Analysis terminated and SCAN mode
                         entered.
```

If joint constraints are detected, the following error message is printed and the analysis is terminated:

```
**** ERROR_STDNL1 --     Joint constraints specified but not
                         presently supported by nonlinear
                         dynamic analysis.
                         Analysis terminated and SCAN mode
                         entered.
```

If dynamic condensation of dynamic degrees of freedom is detected, the following error message is printed and the analysis is terminated.

**** ERROR_STDNL1 -- Dynamic DOF condensation requested but not presently supported by nonlinear dynamic analysis.
Analysis terminated and SCAN mode entered.

2.5.6.3 Nonlinear Dynamic Analysis Example

Figures 2.5.6.3-1 and 2.5.6.3-2 show sketches of a simple two-story space frame structure which is the subject structure of the nonlinear dynamic analysis example problem command listing given in Figure 2.5.6.3-3. Figure 2.5.6.3-1 shows the structure with joints labeled and Figure 2.5.6.3-2 shows the structure with beam and column members and floor finite elements labeled. Figure 2.5.6.3-3 contains the command input file for this example, including comments which describe the use of the nonlinear dynamic analysis commands described in the previous sections.

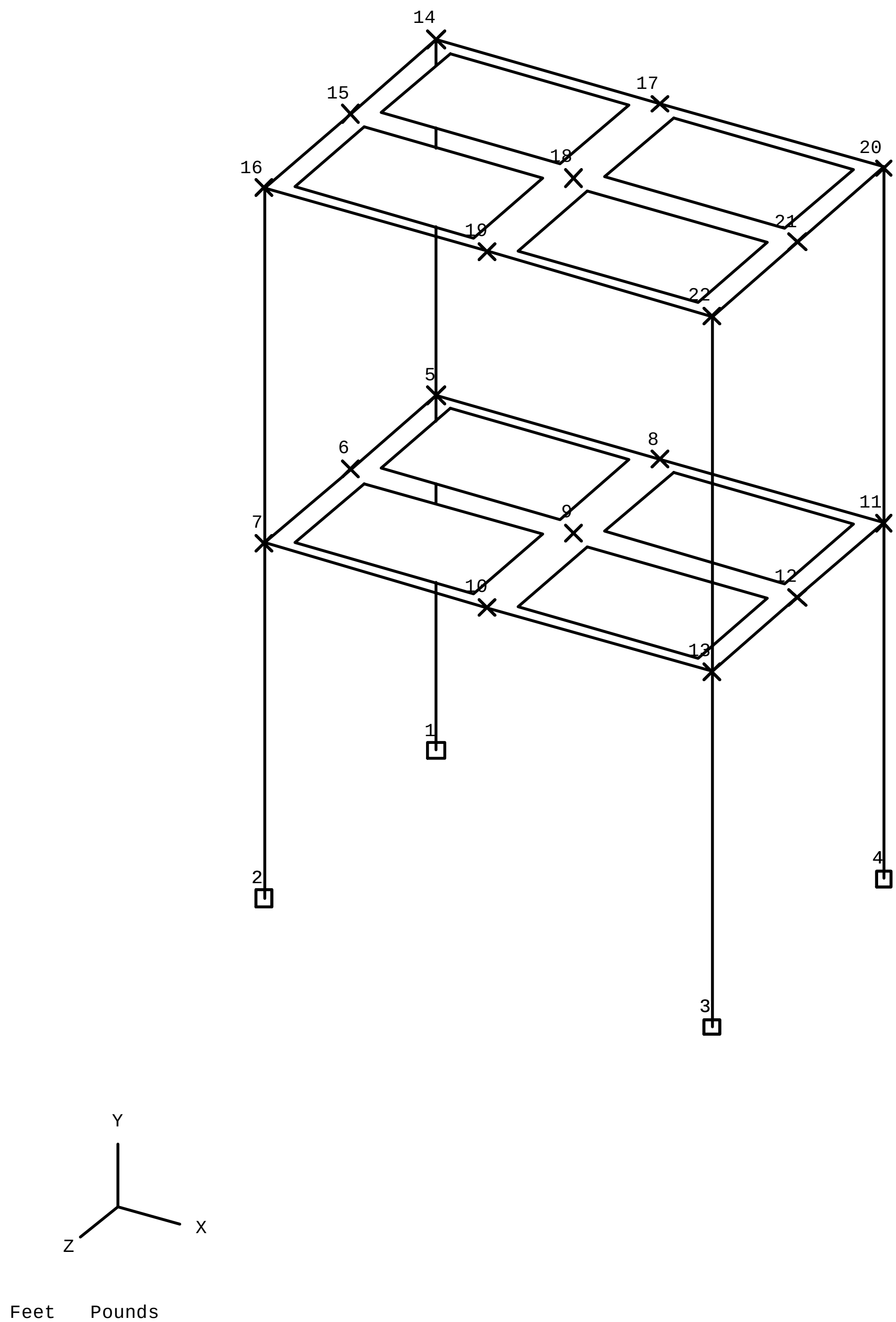


Figure 2.5.6.3-1 Nonlinear Dynamic Analysis Example Structure – Joints Labeled

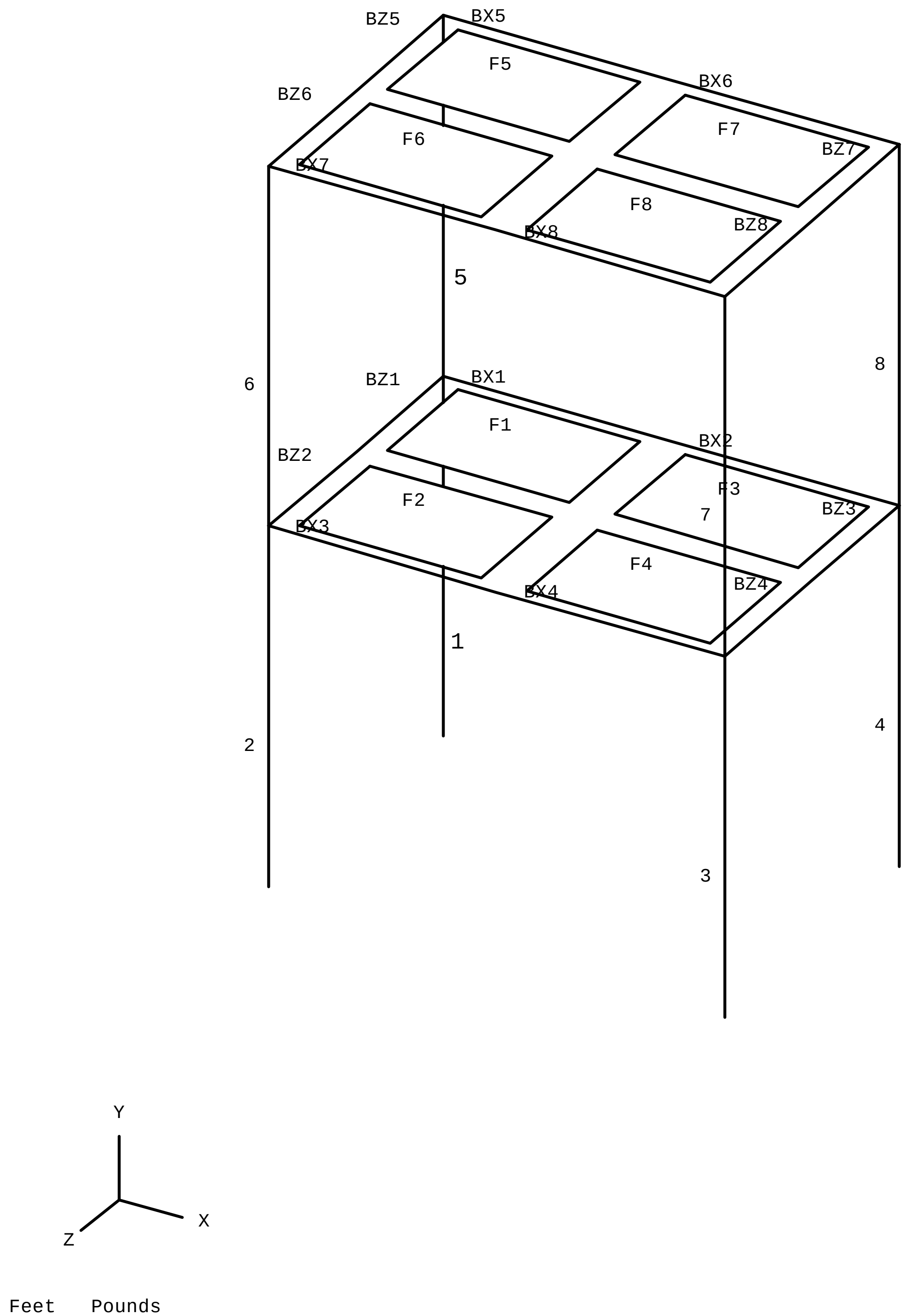


Figure 2.5.6.3-2 Nonlinear Dynamic Analysis Example Structure – Members and Finite Elements Labeled

```
STRUDL 'NlDyEx' 'Nonlinear Dynamic Analysis Example Problem'
$* **
$*****************************************************************
$*****************************************************************
$* **
$
$ Geometry
$
UNITS FEET
JOINT COORD
 1     0.0        0.0      0.0  S
 2     0.0        0.0     10.0  S
 3    15.0        0.0     10.0  S
 4    15.0        0.0      0.0  S

GENERATE 3 JOINTS ID 5 1 X 0.0 Y 12.0 Z 0.0 5.0
REPEAT 2 TIMES ID 3 X 7.5
REPEAT 1 TIME ID 9 Y 12.0

TYPE SPACE FRAME
MEMBER INCIDENCES
1 1 5
2 2 7
3 3 13
4 4 11
GEN 4 MEMBERS ID 5 1 FROM LIST 5 7 13 11 TO LIST 14 16 22 20
GEN 2 MEMBERS ID 'BX1' 1 FROM 5 3 TO 8 3
REPEAT 1 TIME ID 2 FROM 2 TO 2
REPEAT 1 TIME ID 4 FROM 9 TO 9
GEN 2 MEMBERS ID 'BZ1' 1 FROM 5 1 TO 6 1
REPEAT 1 TIME ID 2 FROM 6 TO 6
REPEAT 1 TIME ID 4 FROM 9 TO 9
TYPE PLATE
GEN 2 ELEMENTS ID 'F1' 1 FROM 5 1 TO 6 1 TO 9 1 TO 8 1
REPEAT 1 TIME ID 2 FROM 3 TO 3 TO 3 TO 3
REPEAT 1 TIME ID 4 FROM 9 TO 9 TO 9 TO 9

MATERIAL CONC
UNITS INCHES
MEMBER DIMENSIONS
  1 TO 8 RECT B 8.0 H 12.0
  'BX1' TO 'BX8' 'BZ1' TO 'BZ8' RECT B 8.0 H 18.0
ELEMENT PROPERTIES
  'F1' TO 'F8' TYPE 'SBHQ6' THICK 4.0

$
$ Define self weight and dead load 1.  This load is used as
$ the initial stress loading for the nonlinear dynamic analysis.
$
UNITS FEET LBS
DEAD LOADS 1 DIR -Y MEMBERS 1 TO 8 -
  'BX1' TO 'BX8' 'BZ1' TO 'BZ8'
MEMBER LOADS
```

```
  'BX1' 'BX2' FORCE Y GLOB UNI FR W -500.0 LA 0.0 LB 1.0
ELEMENT LOADS
  'F1' TO 'F8' BODY FORCES GLOBAL BY -150.0

$
$ Define nonlinear geometric behavior for column
$ members 1 to 8.  All other members and finite
$ elements remain linear.
$
NONLINEAR EFFECTS
  GEOMETRY MEMBERS 1 TO 8

$
$ Execute a nonlinear analysis for loading 1 to
$ establish the initial stress conditions.
$
MAXIMUM NUMBER OF CYCLES 10
CONVERGENCE TOLERANCE DISPL 0.001
LOAD LIST 1
NONLINEAR ANALYSIS

UNITS INCHES CYCLES SECS
LIST DISPLACEMENTS
LIST FORCES
LIST SUM REACTIONS

$
$ Add dynamic analysis data including inertia and
$ dynamic loading data.  This data is specified
$ in the same manner as for linear dynamic analysis.
$
UNITS LBS
INERTIA OF JOINTS LUMPED
INERTIA OF JOINTS WEIGHT
  3 4 5 6 TRANSLATION ALL 10000.0

TRANSIENT LOAD 'EQ-X'
SUPPORT ACCELERATION
  TRANSL X FILE 'ELCENTRO'
  INTEGRATE FROM 0.0 TO 10.0 AT 0.01
END TRANS LOAD

TRANSIENT LOAD 'EQ-Z'
SUPPORT ACCELERATION
  TRANSL Z FILE 'ELCENTRO' FACTOR 0.5
  INTEGRATE FROM 0.0 TO 10.0 AT 0.01
END TRANS LOAD

$
$ This block of DYNAMIC PARAMETERS commands contains the most
$ commonly used control data specifications for nonlinear
$ dynamic analysis including the MAXIMUM, CONVERGENCE TOLERANCE,
$ INITIAL STRESS, UPDATE, and RESULTS FILE commands.
$
```

```
DYNAMIC PARAMETERS
  MAXIMUM NUMBER OF EQUILIBRIUM CYCLES 30
  CONVERGENCE TOLERANCE 0.0001
  INITIAL STRESS LOAD 1
  UPDATE STIFFNESS EVERY 3 TIME STEPS
  PRINT MAX
  RESULTS FILE NAME 'N1DyEX'
END DYNAMIC PARAMS

$
$ Execute the nonlinear dynamic analysis
$
DYNAMIC ANALYSIS NONLINEAR

$
$ Perform normal backsubstitution and results processing
$ operations
$
COMPUTE TRANSIENT FORCES
COMPUTE TRANSIENT STRESSES
COMPUTE TRANSIENT REACTIONS LOADS

LIST TRANSIENT FORCES TIMES FROM 1 TO 10 MEMBER 1 2 'BX1' 'BZ1'
LIST TRANSIENT MAX FORCES MEMBERS 1 2 'BX1' 'BZ1'
LIST TRANSIENT MAX DISPL JOINT 5

CREATE PSEUDO STATIC LOAD 3 FROM TIME 2 OF LOAD 'EQ-X'
CREATE PSEUDO STATIC LOAD 4 FROM MAX OF LOAD 'EQ-X'
CREATE PSEUDO STATIC LOAD 5 FROM TIME 2 OF LOAD 'EQ-Z'
CREATE PSEUDO STATIC LOAD 6 FROM MAX OF LOAD 'EQ-Z'
CREATE PSEUDO STATIC LOAD 7 FROM TIME 2 OF LOADS 'EQ-X' 'EQ-Z'
CREATE PSEUDO STATIC LOAD 8 FROM MAX OF LOADS 'EQ-X' 'EQ-Z'

LOAD LIST 1 3 TO 8
UNITS INCHES KIPS
OUTPUT BY MEMBER
LIST DISPL JOINT 5
LIST FORCES MEMBERS 'BX1' 'BZ1'

SECTION FR NS 3 0.0 0.5 1.0
LIST SECTION FORCES MEMBERS 'BX1' 'BZ1'

FIN
```

Figure 2.5.6.3 Command Input File for Nonlinear Dynamic Analysis Example

This page intentionally left blank.

5.3.8 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:

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

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

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

•

•

•

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

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

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

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 ($\leq 10^{-5}$ inches), then GLOBAL is assumed. In addition, GLOBAL is automatically assumed for any damper element for which only one incident joint is specified.

[<u>CTX</u>] v_{CTX} [<u>CTY</u>] v_{CTY} [<u>CTZ</u>] v_{CTZ} [<u>CRX</u>] v_{CRX} [<u>CRY</u>] v_{CRY} [<u>CRZ</u>] 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:

$$\underline{\text{DEL}}\text{ETE }\underline{\text{DAM}}\text{PER }(\underline{\text{ELE}}\text{MENT }\underline{\text{DAT}}\text{A}) \begin{Bmatrix} i_D \\ 'a_D' \end{Bmatrix} \ldots \begin{Bmatrix} i_D \\ 'a_D' \end{Bmatrix}$$

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.

5.4 General Prerelease Features

5.4.1 Align Command

The ALIGN command will align joints in a line of members in a column to ensure that the column is parallel to the global Y axis. The section below is numbered as it will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.12.16 The ALIGN Command

This command will move joints at the "top" (top if the Y axis is vertical) of members parallel to the global Y axis, so that the "top" joint has the same X and Z coordinates as the "bottom" joint. This is to ensure that all these members follow the 'Special Case' for local reference frame orientation. See Section 1.10.4 in Volume 1 of the GTSTRUDL User Reference Manual or Section 8.5, "The BETA Angle", in the GTSTRUDL Analysis Users Guide. This command is useful for situations where coordinates are calculated to a precision that does not ensure alignment with the Y axis, such as coordinates created by an outside program, or use of the GENERATE or OBJECT COPY commands with many copies or large coordinates. Since the check for being parallel is very exacting in GTSTRUDL (within 0.01%), it is possible for a column line to have some columns that are 'Special Case' members and others that are not, resulting in inconsistent default orientations.

Notes:

This "out of alignment" problem can also occur for beams parallel to the Y global axis in “Z up” structures.

MEMBER ECCENTRICITIES are not considered during the alignment check because they are not used to determine the default member orientation.

Syntax:

ALIGN (MEMBERS list) (REPORT (ONLY)) (ALONG Y AXIS) -
(TOLERANCE v_1 (RATIO)) (ITEMIZE (WITH DETAIL))

Where,

list is an optional list of members, as defined by Section 2.1.2.2 of this volume.

v_1 is the specified TOLERANCE used to define “parallel to global Y”

Explanation:

MEMBERS If no list is given, all members will be checked.

REPORT Don't change joint coordinates, only report detected possible alignment problems. Since the adjustment procedure changes joints from the "bottom" of the structure (starting at the most negative Y location) and works toward the "top", 'REPORT ONLY' alignment checks may be slightly different because "lower" joints will not have been changed.

TOLERANCE
Members which have an X or Z projection equal to or less than this value will be considered parallel to Y and will be checked for alignment. The default value is 0.1 inch (2.5 mm).

RATIO This option specifies that the tolerance specified be used as a ratio: (X or Z projection)/(member length). This option allows you to have more control in special situations. To avoid possible problems the TOLERANCE value is not permitted to exceed 0.05.

ITEMIZE This option will generate a message for each joint that is changed.

DETAIL This option will also print the old coordinates and the new coordinates of each changed joint.

The ALIGN MEMBERS command will change joint coordinates to keep members parallel to the global Y axis, unless the REPORT option is specified. A list of all members that meet the criteria for being considered "parallel to global Y" is built, respecting the default or user-specified values for TOLERANCE and RATIO. Members are then 'aligned' (joint coordinates changed) from the "bottom", or most negative Y coordinate, to the "top".

Examples:

```
ALIGN REPORT ONLY ITEMIZE

****INFO_STALGN - The following joints need to be adjusted to align members
                  in the global Y direction:
          Joint 5         needs to be aligned with joint 1
          Joint 9         needs to be aligned with joint 5
```

```
ALIGN ITEMIZE WITH DETAIL

 ****INFO_STALGN - The following joints were adjusted to align members
                  in the global Y direction:
         Joint 5        was aligned with joint 1
            Old (X,Z):  0.10000E+00  0.10000E+00  New (X,Z):  0.00000E+00  0.00000E+00
         Joint 9        was aligned with joint 5
            Old (X,Z):  0.20000E+00  0.20000E+00  New (X,Z):  0.00000E+00  0.00000E+00
         Joint 13       was aligned with joint 9
            Old (X,Z): -0.30000E+00 -0.30000E+00  New (X,Z):  0.00000E+00  0.00000E+00
         Joint 16       was aligned with joint 12
            Old (X,Z):  0.45100E+03  0.10000E+01  New (X,Z):  0.45000E+03  0.00000E+00

 ****INFO_STALGN - A total of     4 joints were adjusted.
```

```
ALIGN ITEMIZE

 ****INFO_STALGN - The following joints were adjusted to align members
                  in the global Y direction:
         Joint 5        was aligned with joint 1
         Joint 9        was aligned with joint 5
         Joint 13       was aligned with joint 9
         Joint 16       was aligned with joint 12

 ****INFO_STALGN - A total of     4 joints were adjusted.
```

```
ALIGN

 ****INFO_STALGN - A total of     4 joints were adjusted.
```

If no members were determined to be misaligned:

```
ALIGN

 ****INFO_STALGN - No joints required adjustment.
```

This page intentionally left blank.

5.4.2 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:

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

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 θ_1, θ_2, θ_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.
```

5.4.3 RUN Command

The RUN command allows you run external programs or DOS batch (cmd) files with a GTSTRUDL command. This is useful for automating procedures that rely on GTSTRUDL generated data, such as a user created design program that needs member ends forces from GTSTRUDL.

The RUN command has been improved to allow new options and longer commands. Implementation has changed from the "C" system library to a Microsoft API, which is more robust way to run external programs.

Syntax

<u>RUN</u> ((<u>BAT</u>CH) (<u>K</u>EEP)) (<u>WAIT</u>) 'program'

where program = a ".exe", ".bat" or ".cmd" file or DOS command, along with arguments. The total length of 'program' is limited to 255 characters. You cannot use the quote/apostrophe character (') in 'program', but double quotes (") are acceptable.

Explanation:

BATCH — Indicates that a ".bat" or ".cmd" file or DOS command will be executed. ".bat" or ".cmd" file are DOS script files, containing DOS commands or other programs. The starting directory for the DOS environment is the current GTSTRUDL working directory.
The Microsoft API now being used needs to know that a DOS environment is needed before it runs.

KEEP — If BATCH is used, the KEEP option leaves the created DOS window active after the requested action is completed. You can then type new commands in the DOS window. You can kill the DOS window with the "X" in the upper, right hand corner when you are finished.

WAIT — Indicates that GTSTRUDL command processing should stop until 'program' has completed. If this option is not used, GTSTRUDL will continue and process the next command, i.e. if you are running a macro or an input file. Use the WAIT option if the results of 'program' are required for subsequent GTSTRUDL commands.

Examples:

RUN BATCH KEEP 'DIR/W'

This command will open a DOS window, display the contents of the GTSTRUDL working directory in the "/W" format, and leave the DOS window open so you can type more commands. The DOS window is a separate program, so you can continue GTSTRUDL operations while the DOS window is open.

RUN 'NOTEPAD myfile.txt'

This command will open Notepad with file 'myfile.txt'. Again, Notepad is a separate process, so you can continue GTSTRUDL operations while Notepad is open.

RUN BATCH WAIT 'find "$" input.gti > comments.gti'
CINPUT 'comments.gti'

The first command will use the DOS command "find" to locate the comments in the input file "input.gti" and put them into a file named "comments.gti". The WAIT option tells the GTSTRUDL command processor to wait until the DOS operation is complete before trying to CINPUT the generated file.

5.4.4 COUTPUT Command

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

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

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.

5.4.5 Reference Coordinate System Command

General form:

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

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

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 (θ_1, θ_2, and θ_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.

5.4.5-1 Printing Reference Coordinate System Command

General form:

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

Explanation:

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

5.4.6 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.

5.4.7 Add/Delete Without Commands

2.1.3.7 Delete Joints Without Joint Coordinates

General form:

DELETE JOINTS WITHOUT (JOINT) COORDINATES

Explanation:

The DELETE JOINTS WITHOUT JOINT COORDINATES command can be used to delete the joints that do not have joint coordinates. There are times when a user may have accidently specified data for a joint that does not exist or accidently specified a non-existent joint name in the PRINT or LIST commands. Once a joint name is specified in a command, the joint name is stored in the database. If joint coordinates have not been specified for a joint and STIFFNESS ANALYSIS is requested, the following error message is output:

```
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 30
```

When the DELETE JOINTS WITHOUT JOINT COORDINATES command is specified, the program searches through all the joint names and deletes the joint names that do not have JOINT COORDINATES.

Example:

```
{  100} >
{  101} > STIFFNESS  ANALYSIS
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 30
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 31
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 32
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 33
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 34
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 35
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 40
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 45
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 50
**** WARN_STDSY1 -- SCAN mode is ON.  Analysis will be halted.

{  102} >
```

```
{  103} > SCAN OFF
{  104} >
{  105} > DELETE JOINTS WITHOUT JOINT COORDINATES


Following joints have been deleted: ------------------------------
  30         31         32         33         34         35
  40         45         50
```

In the above example, joints 30 to 35, 40, 45, and 50 accidently were specified in the PRINT JOINT COORDINATES command. The command DELETE JOINTS WITHOUT JOINT COORDINATES deletes the joints that do not have joint coordinates.

2.1.3.8 Delete Members or Finite Elements Without Member or Element Incidences

General form:

<u>DEL</u>ETE <u>MEM</u>BERS <u>WIT</u>HOUT (<u>MEM</u>BER) <u>INC</u>IDENCES

or

<u>DEL</u>ETE (<u>FIN</u>ITE) <u>ELE</u>MENTS <u>WIT</u>HOUT (<u>ELE</u>MENT) <u>INC</u>IDENCES

Explanation:

The above two commands can be used to delete the members or finite elements that do not have member or element incidences. There are times when a user may have accidently specified data for a member or a finite element that does not exist or accidently specified a member or element name in the PRINT or LIST commands. Once a member or a finite element name has been specified in a command, the member or finite element name is stored in the database. If member or element incidences have not been specified for a member or element and STIFFNESS ANALYSIS is requested, the following error message is output.

**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 20
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 45

When the DELETE MEMBERS WITHOUT MEMBER INCIDENCES command is specified, the program searches through all the member names and deletes the members that do not have MEMBER INCIDENCES. When the DELETE FINITE ELEMENTS WITHOUT ELEMENT INCIDENCES command is specified, the program searches through all the finite element names and deletes the elements that do not have ELEMENT INCIDENCES.

Example:

```
{   97} >
{   98} > STIFFNESS  ANALYSIS
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 20
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 21
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 22
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 23
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 24
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 30
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 33
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 51
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 52
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 53
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 54
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 55
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 27
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 32
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 38
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 45
**** WARN_STDSY1 -- SCAN mode is ON.  Analysis will be halted.

{   99} >
{  100} > SCAN OFF
{  101} >
{  102} > DELETE MEMBERS WITHOUT MEMBER INCIDENCES


Following members have been deleted: ------------------------------
   20         21         22         23         24         30
   33         51         52         53         54         55
   27         32         38         45
```

In the above example, some of the members that are listed were accidently specified in the MEMBER PROPERTIES command and some in the PRINT MEMBER INCIDENCES command. The command DELETE MEMBERS WITHOUT MEMBER INCIDENCES deletes the members that do not have member incidences.

This page intentionally left blank.

5.4.8 The List Sum Forces Command

The LIST SUM FORCES command is used to perform a general computation of resultant forces for a specified set of joints, members, and finite elements, and is described in the section below, which is numbered as it will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.3.7.4 The LIST SUM FORCES Command

General form:

$$\underline{\text{LIST}}\ \ \underline{\text{SUM}}\ \ \underline{\text{FOR}}\text{CES}\ \ (\underline{\text{TITL}}\text{E}\ 'a_T')\ (\underline{\text{ABO}}\text{UT}\ \left\{\begin{array}{l} \underline{\text{JOI}}\text{NT}\left\{\begin{array}{l} i_c \\ 'a_c' \end{array}\right\} \\ [\underline{X}]\ v_X\ [\underline{Y}]\ v_Y\ [\underline{Z}]\ v_Z \end{array}\right\})\ -$$

$$(\underline{\text{ORI}}\text{ENTATION}\ \left\{\begin{array}{l} [\underline{\text{TH1}}]\ v_1\ [\underline{\text{TH2}}]\ v_2\ [\underline{\text{TH3}}]\ v_3 \\ \left\{\begin{array}{l} \underline{\text{MEM}}\text{BER} \\ \underline{\text{ELE}}\text{MENT} \end{array}\right\}\ \left\{\begin{array}{l} i_O \\ 'a_O' \end{array}\right\} \end{array}\right\})\ \ \underline{\text{JOI}}\text{NTS list}_j\ \ \underline{\text{MEM}}\text{BERS list}_m$$

Elements:

$'a_T'$	=	optional 16-character title. The default title is "None Given."
i_c/$'a_c'$	=	integer or alphanumeric name of the optional joint about which moment summation is computed.
v_X, v_Y, v_Z	=	decimal values of the global X, Y, and Z coordinates of the optional point about which moment summation is computed. A value of 0.0 is assumed for coordinates not specified.
v_1, v_2, v_3	=	decimal values of the rotation angles that define the orientation of the optional local reference frame in which the force summation results are to be reported. The rotation angles are the same ones used to define the local released support reference frame described in Section 2.1.7.1, Volume 1 of the GTStrudl Reference Manual and illustrated by Figure 2.1.7.1 of that same Reference Manual section.

i_O/'a_O' = integer or alphanumeric name of the member or finite element that is to be used as the basis of the local reference frame in which the force summation results are to be reported.

$list_j$ = list of joints at which forces and moments from the members and finite elements of $list_m$ are accumulated for the force summation computation.

$list_m$ = list of members and finite elements that contribute forces at the joints of $list_j$ for the force summation computation.

Explanation:

The LIST SUM FORCES command is used to calculate and report the summation of forces and moments at a specified point from a specified set of member and finite element nodal forces for all active static loading conditions. This command is similar to, but more general than, the CALCULATE RESULTANT command described in Section 2.3.7.3, Volume 3 of the GTSTRUDL Reference Manual. The LIST SUM FORCES command options are described as follows:

$$\underline{\text{ABO}}\text{UT} \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_C \\ 'a_C' \end{Bmatrix} \\ [\underline{\text{X}}]\ v_X\ \ [\underline{\text{Y}}]\ v_Y\ \ [\underline{\text{Z}}]\ v_Z \end{Bmatrix}$$

By default, the moment summation is computed by summing the member and finite element forces and moments about the geometric center of the joints specified in $list_j$, assuming the joints are equally weighted. The ABOUT specs provide for the identification of an optional JOINT or point, v_X, v_Y, v_Z, about which the moment summation is performed. The specified joint must be defined, but may be inactive.

$$(\underline{\text{ORI}}\text{ENTATION} \begin{Bmatrix} [\underline{\text{TH1}}]\ v_1\ [\underline{\text{TH2}}]\ v_2\ [\underline{\text{TH3}}]\ v_3 \\ \begin{Bmatrix} \underline{\text{MEM}}\text{BER} \\ \underline{\text{ELE}}\text{MENT} \end{Bmatrix} \begin{Bmatrix} i_O \\ 'a_O' \end{Bmatrix} \end{Bmatrix})$$

By default, the result of the force and moment summation is a list of three global resultant force components and three global resultant moment components for each active loading condition. The ORIENTATION specs provide a way to specify an optional alternative local reference frame for the summation report.

In the first method, three rotation angles v_1, v_2, and v_3 define the orientation of the local reference frame with respect to the global reference frame. The rotation angles are the same ones used to define the local released support reference frame described in Section 2.1.7.1, Volume 1 of the GTSTRUDL Reference Manual and illustrated by Figure 2.1.7.1 of that same Reference Manual section. According to the MEMBER/ELEMENT option, the summation results are reported in the local or planar reference frame of the specified MEMBER or finite ELEMENT.

JOINTS $list_j$ MEMBERS $list_m$

The LIST SUM FORCES command computes the force and moment summation from the nodal forces and moments at the joints specified in $list_j$ from the members and finite elements specified in $list_m$. $list_m$ may contain any mix of frame members, truss members, 2-D, and 3-D finite elements and $list_j$ may contain any set of joints, with the only qualification that each element in $list_m$ must be incident on at least one joint in $list_j$. In addition, every joint and element in $list_j$ and $list_m$ must be defined and active, and $list_m$ may not contain rigid bodies. $list_j$ may contain master joints and slave joints.

The force and moment summation is computed for all active static loading conditions for which member end forces and finite element nodal forces have been computed by a prior STIFFNESS ANALYSIS or by the CREATE PSEUDO STATIC, COMBINE, and CREATE LOAD COMBINATION commands. In the case of a pseudo static loading condition created from a response spectrum modal combination such as RMS or CQC, a force and moment summation is computed first for each mode of the root response spectrum load. The final mode combination force and moment summation is then computed by applying the mode combination rule of the pseudo static loading to the individual modal force and moment summations. In the case of a pseudo static loading condition created from the maximum of a transient loading condition, a force and moment summation is computed for each individual time point, from which the maximum force and moment summation component is then selected.

Error and Warning conditions

The LIST SUM FORCES command issues error and warning messages when the following conditions are encountered. In each case, command processing is terminated, but scan mode is not entered.

```
**** WARNING_STLFSM --  Joint and/or member list not defined.  LIST SUM
                        FORCES command ignored.
```

This message is issued when either the JOINT and/or MEMBER list are missing from the LIST SUM FORCES command.

```
**** WARNING_STLFSM -- Specified joint 500       does not exist or is
                       inactive.  LIST SUM FORCES command ignored.
**** WARNING_STLFSM -- Specified element 500      does not exist or is
                       inactive.  LIST SUM FORCES command ignored.
```

These messages mean that the indicated joint or element in $list_j$ or $list_m$ respectively has not been defined or is inactive.

```
**** WARNING_STLFSM -- Joint 500      for center of force summation does
                       not exist.  LIST SUM FORCES command ignored.
```

This message indicates that the specified joint in the ABOUT JOINT option is not defined.

```
**** WARNING_STLFSM -- None  of  the  specified  members are incident on
                       joint 24 .  LIST SUM FORCES command ignored.

**** WARNING_STLFSM -- Member 40   not incident on any of the specified
                       joints.  LIST SUM FORCES command ignored.
```

These messages are issued when the indicated joint is disconnected from the elements named in $list_m$, or when the indicated member is disconnected from the joints named in $list_j$ respectively.

```
**** WARNING_STLFSM --  There are no active loading conditions.  LIST SUM
                        FORCES command ignored.
```

This message indicates that no active loading conditions where found.

```
 **** WARNING_STLFSM -- Member/element  force  results not available for
                        member  25  loading  2.   LIST  SUM  FORCES  command
                        ignored.
```

This message indicates that static analysis results do not exist for the specified member and loading condition. The output of this message obeys the OUTPUT WARNING MESSAGE LIMIT command.

```
 **** WARNING_STLFSM -- Specified element RB1 is an invalid rigid body.
                        LIST SUM FORCES command ignored.
```

This message is issued when a rigid body is detected in $list_m$.

Example:

Example LSF01 demonstrates the use of the LIST SUM FORCES command for the simple steel-concrete composite beam shown in Figure 2.3.7.4-1 below. The nodal forces from Joints 41 to 45 at X = 40 ft. from plate elements 29 to 32 and beam member 56 (the more darkly-shaded elements) are used to compute the force summation at the cross section X = 40 ft. The command listing is shown in Figure 2.3.7.4-2. The report from the LIST SUM FORCES command is shown in Figure 2.3.7.4-3.

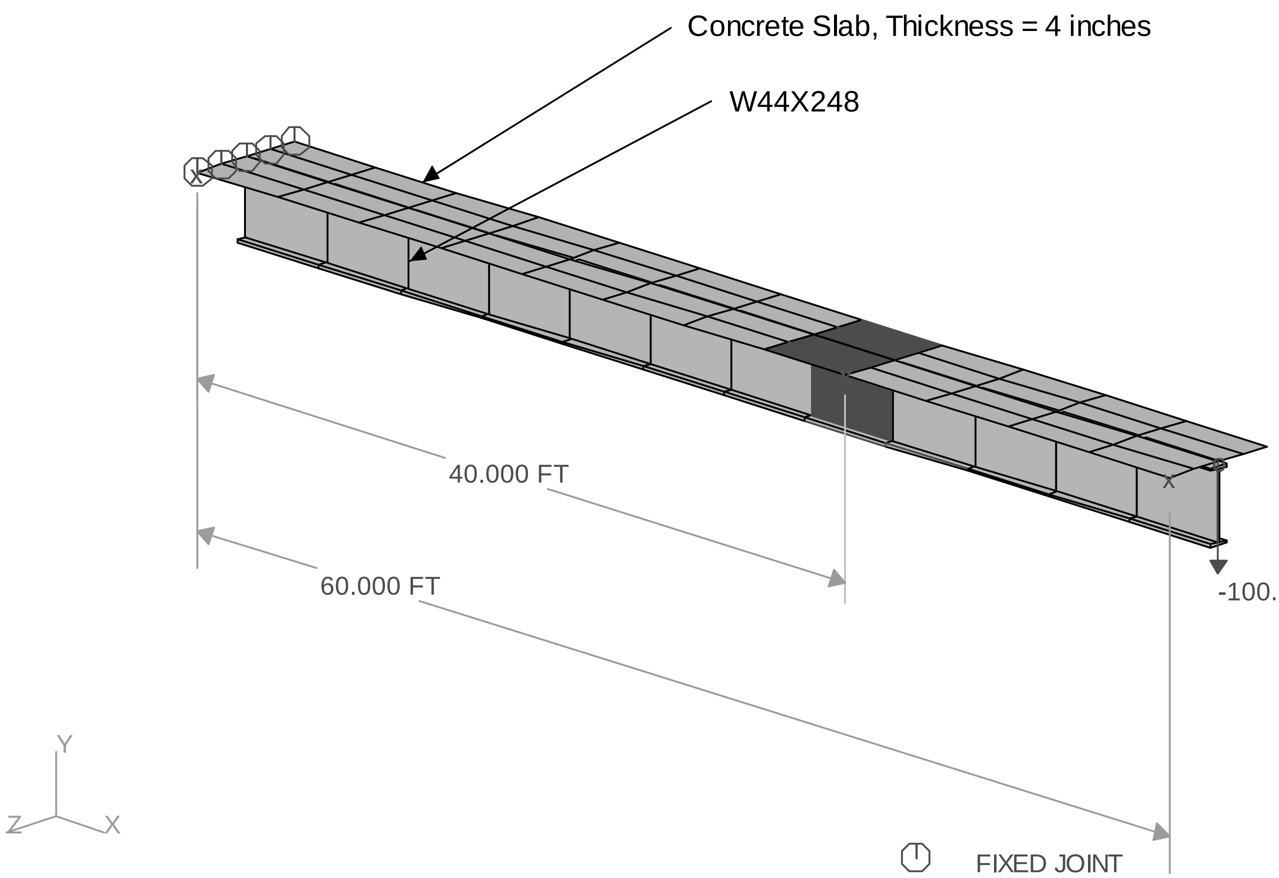


Figure 2.3.7.4-1 Example LFS01, Concrete-Steel Composite Cantilever Beam

```
$$* **
$$* ** This job demonstrates the LIST SUM FORCES command for a
$$* ** steel-concrete composite beam
$$* **
UNITS  FEET  KIPS  DEG   FAH
$$
$$
GTMenu POINT COORDINATES
   'P1'                  0.0000        0.0000       -3.0000
   'P2'                 60.0000        0.0000       -3.0000
   'P3'                  0.0000        0.0000        3.0000
$$
$$
GTMenu LINE INCIDENCES
```

```
   'C1'        'P2'        'P1'
   'C2'        'P1'        'P3'
$$
$$
UNITS  FEET  KIPS  DEG   FAH
$$
$$
JOINT COORDINATES GLOBAL
    1                    0.0000         0.0000        -3.0000
    2                    0.0000         0.0000        -1.5000
    3                    0.0000         0.0000         0.0000
    4                    0.0000         0.0000         1.5000
    5                    0.0000         0.0000         3.0000
    6                    5.0000         0.0000        -3.0000
    7                    5.0000         0.0000        -1.5000
    8                    5.0000         0.0000         0.0000
    9                    5.0000         0.0000         1.5000
    10                   5.0000         0.0000         3.0000
    11                  10.0000         0.0000        -3.0000
    12                  10.0000         0.0000        -1.5000
    13                  10.0000         0.0000         0.0000
    14                  10.0000         0.0000         1.5000
    15                  10.0000         0.0000         3.0000
    16                  15.0000         0.0000        -3.0000
    17                  15.0000         0.0000        -1.5000
    18                  15.0000         0.0000         0.0000
    19                  15.0000         0.0000         1.5000
    20                  15.0000         0.0000         3.0000
    21                  20.0000         0.0000        -3.0000
    22                  20.0000         0.0000        -1.5000
    23                  20.0000         0.0000         0.0000
    24                  20.0000         0.0000         1.5000
    25                  20.0000         0.0000         3.0000
    26                  25.0000         0.0000        -3.0000
    27                  25.0000         0.0000        -1.5000
    28                  25.0000         0.0000         0.0000
    29                  25.0000         0.0000         1.5000
    30                  25.0000         0.0000         3.0000
    31                  30.0000         0.0000        -3.0000
    32                  30.0000         0.0000        -1.5000
    33                  30.0000         0.0000         0.0000
    34                  30.0000         0.0000         1.5000
    35                  30.0000         0.0000         3.0000
    36                  35.0000         0.0000        -3.0000
    37                  35.0000         0.0000        -1.5000
    38                  35.0000         0.0000         0.0000
    39                  35.0000         0.0000         1.5000
    40                  35.0000         0.0000         3.0000
    41                  40.0000         0.0000        -3.0000
    42                  40.0000         0.0000        -1.5000
    43                  40.0000         0.0000         0.0000
    44                  40.0000         0.0000         1.5000
    45                  40.0000         0.0000         3.0000
    46                  45.0000         0.0000        -3.0000
    47                  45.0000         0.0000        -1.5000
    48                  45.0000         0.0000         0.0000
    49                  45.0000         0.0000         1.5000
    50                  45.0000         0.0000         3.0000
    51                  50.0000         0.0000        -3.0000
```

```
    52                   50.0000        0.0000        -1.5000
    53                   50.0000        0.0000         0.0000
    54                   50.0000        0.0000         1.5000
    55                   50.0000        0.0000         3.0000
    56                   55.0000        0.0000        -3.0000
    57                   55.0000        0.0000        -1.5000
    58                   55.0000        0.0000         0.0000
    59                   55.0000        0.0000         1.5000
    60                   55.0000        0.0000         3.0000
    61                   60.0000        0.0000        -3.0000
    62                   60.0000        0.0000        -1.5000
    63                   60.0000        0.0000         0.0000
    64                   60.0000        0.0000         1.5000
    65                   60.0000        0.0000         3.0000
$$
$$
UNITS  FEET  KIPS  DEG   FAH
$$
$$
$$
TYPE  SPACE FRAME
MEMBER INCIDENCES
   49           3            8
   50           8            13
   51           13           18
   52           18           23
   53           23           28
   54           28           33
   55           33           38
   56           38           43
   57           43           48
   58           48           53
   59           53           58
   60           58           63
$$
$$
UNITS  FEET  KIPS  DEG   FAH
$$
MEMBER ECCENTRICITIES GLOBAL
  49 TO 60 START Y -2.0000  END Y -2.0000
$$
$$
$$
UNITS  FEET  KIPS  DEG   FAH
$$
MEMBER PROPERTIES    TABLE 'WBEAM9  '  'W44x248                 '
  49 TO 60

$$
STATUS SUPPORT  -
    1            2            3            4            5

$$
$$
UNITS  FEET  KIPS  DEG   FAH
$$
CONSTANTS
     BETA      0.00000 ALL
$$
```

```
$$
TYPE  PLATE
GENERATE 4 ELEMENTS ID 1 1 FROM 1 1 TO 2 1 TO 7 1 TO 6 1
REPEAT 11 TIMES ID 4 FROM 5 TO 5 TO 5 TO 5
$$
$$
UNITS  FEET  KIPS  DEG   FAH
$$
CONSTANTS
     E    5.1839997E+05  ALL
     G    2.0735998E+05  ALL
     POI  1.7000000E-01  ALL
     DEN  1.4999039E-01  ALL
     CTE  5.5000000E-06  ALL
MATERIAL STEEL MEMBERS 49 TO 60
$$
UNITS  FEET  KIPS  DEG   FAH
$$
$$
ELEMENT PROPERTIES
  1 TO 48 TYPE 'SBHQ6' THICK 0.3333
$$
UNITS  FEET  KIPS  DEG   FAH
$$
LOADING  1
JOINT LOADS
  63  FORCE Y -100.0

STIFFNESS ANALYSIS

UNITS KIPS FEET
LIST FORCES MEMBER 56
LIST ELEMENT FORCES ELEMENTS 29 TO 32

$$* **
$$* ** Resultant sum of forces in the composite beam at
$$* ** X = 40 ft., using the member and finite element nodal
$$* ** forces at joints 41 to 45 of elements 29 to 32 and
$$* ** member 56
$$* **
LIST SUM FORCES TITLE 'Example LSF01' ELEMENTS 29 TO 32 56  -
  JOINTS 41 TO 45
FINISH
```

Figure 2.3.7.4-2   List of Commands for Example LSF01

```
 {  177} > $$* **
 {  178} > $$* ** Resultant sum of forces in the composite beam at
 {  179} > $$* ** X = 40 ft., using the member and finite element nodal
 {  180} > $$* ** forces at joints 41 to 45 of elements 29 to 32 and
 {  181} > $$* ** member 56
 {  182} > $$* **
 {  183} > LIST SUM FORCES TITLE ‘LSF01' ELEMENTS 29 TO 32 56 JOINTS 41 TO 45

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


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

Member and Finite Element Force Resultants
==========================================

Title: Example LSF01

Sum of Forces and Moments about      40.0000       0.0000       0.0000
Local Axis Orientation Angles:  Z       0.0000  Y       0.0000  X       0.0000

  Loading         FX              FY              FZ              MX              MY              MZ
  -------        ------          ------          ------          ------          ------          ------

  1                0.0000       -100.0000          0.0000          0.0000          0.0000      -2000.0001
```

Figure 2.3.7.4-3   Report From the LIST SUM FORCES Command, Example LSF01

5.4.9 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

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

.
.
.

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

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:

GTMenu LINE INCIDENCES

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

$$\vdots$$

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

Elements:

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

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

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.