GT STRUDL ®

GT STRUDL
Release Guide Version 27

Volume 1 of 2

June 2003
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 27, with a release date in the GTSTRUDL title block of June 2003.

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

This page intentionally left blank.

CHAPTER 1

INTRODUCTION

Version 27 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 26 release. Chapter 3 provides you with details regarding error corrections that have been made since the Version 26 release. Chapter 4 describes known problems with Version 27. 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 27 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

5.3 Analysis Prerelease Features
- 5.3.1 Calculate Error Estimate Command
- 5.3.2 Finite Element Dictionary Revision - Temperature Gradient Loading
- 5.3.3 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis
- 5.3.4 Dynamic Analysis External File Solver to Improve Efficiency of Dynamic Analysis Results Computation
- 5.3.5 Output of Response Spectrum Results
- 5.3.6 Form Static Load Command
- 5.3.7 Form UBC97 Load Command
- 5.3.8 Form IS1893 Load Command
- 5.3.9 Nonlinear Effects 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.

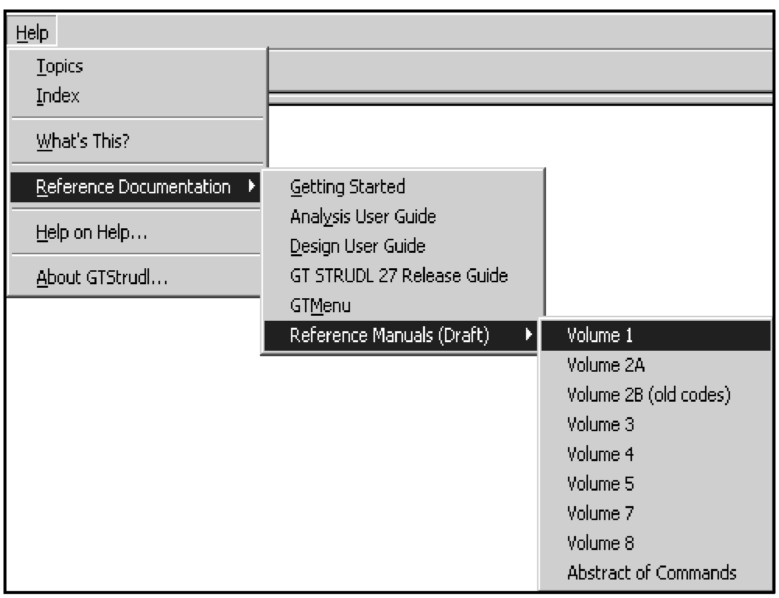
Note that GTMenu is described in Volume 2 of this Release Guide.

Chapter 2
New Features in Version 27

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 27. This release guide is also available online upon execution of GTSTRUDL under Help/Reference Documentation/GTSTRUDL 27 Release Guide. Other documentation has also changed in Version 27. You should also review the following online documentation which is also available under Help:

- GTMenu
- Getting Started
- Analysis Guide
- Design Guide

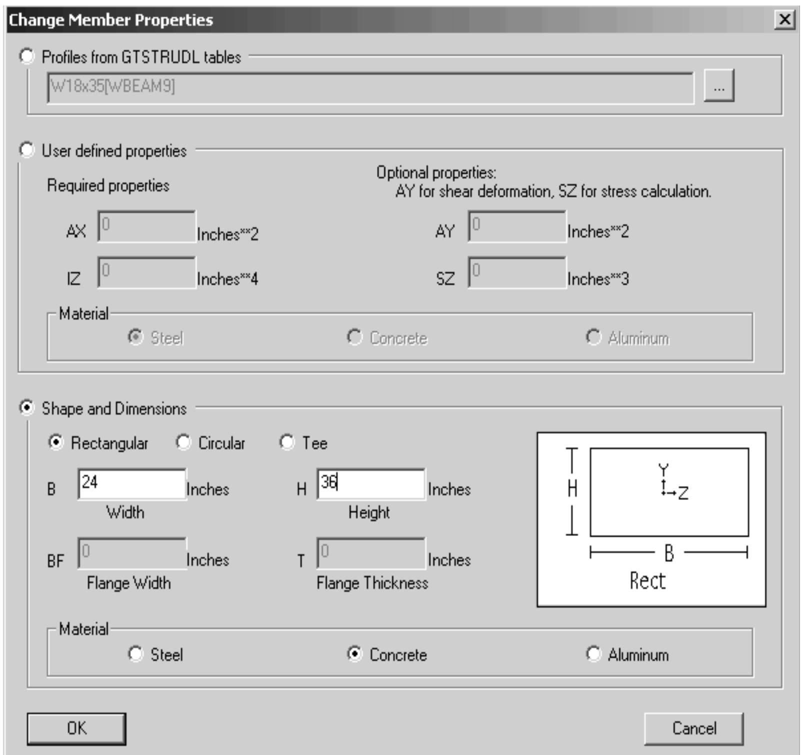
Furthermore, the GTSTRUDL Reference Manuals are now available in draft form under Help as shown below:



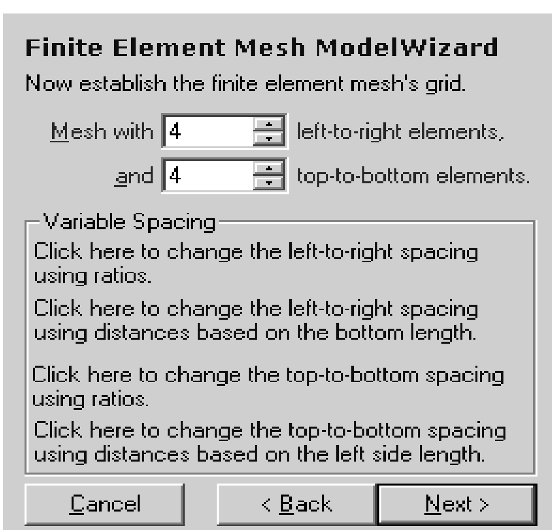
2.1 GTSTRUDL Startup

Model Wizard

The default material property for member properties specified by Shape and Dimensions in the Model Wizard is now Concrete instead of Steel as shown in the dialog below:



The Finite Element Mesh model Wizard now allows you to specify variable spacing when creating a finite element mesh as shown in the dialog below:



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.

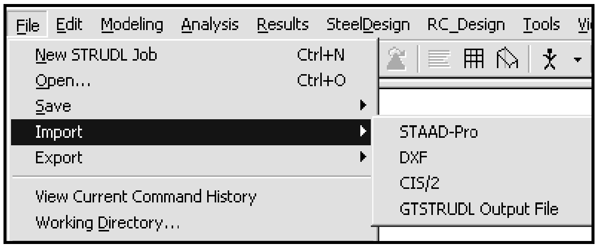
Menu Bar

The Menu Bar has changed. The Design menu has been changed to Steel Design and a new RC_Design pulldown has been added. The revised Menu Bar is shown below:

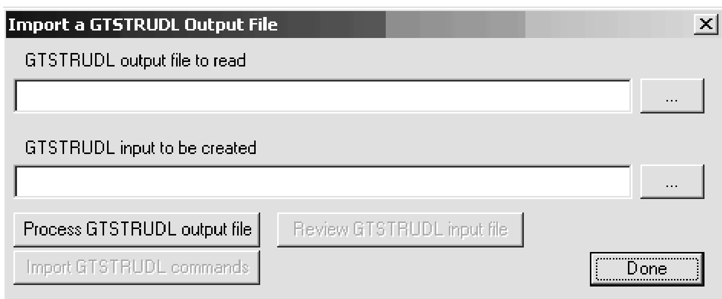
File Pulldown

The new File pulldown is shown below:

Also, under Import in the File pulldown, a new option has been added to import commands from an existing GTSTRUDL Output file as shown below:

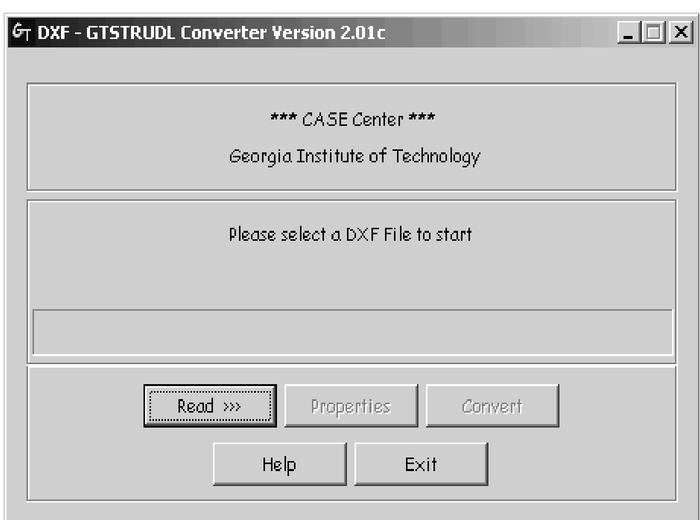


The following dialog appears when you select the GTSTRUDL Output File option:

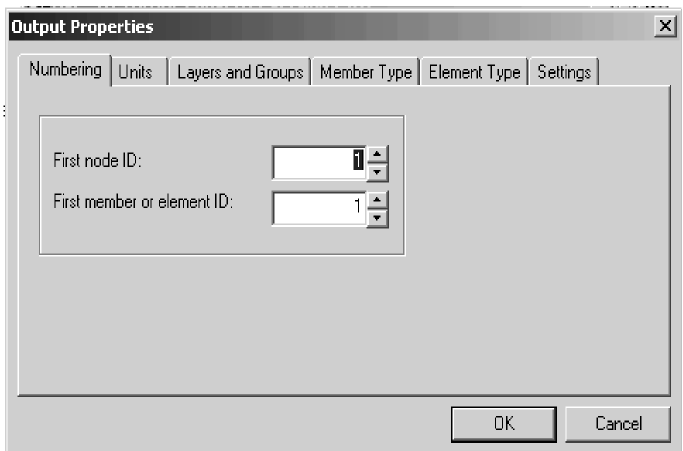


This feature will create a GTSTRUDL input file from an existing output file and then import the commands into the current GTSTRUDL session. You may also review the input file.

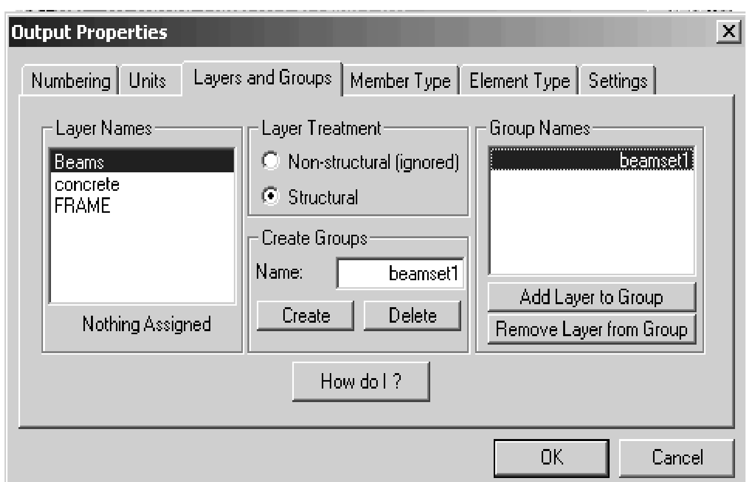
The dialogs which control the Import from a DXF file have changed and new features have been added. The new dialog which pops up after selecting File, Import, and then DXF is shown below:



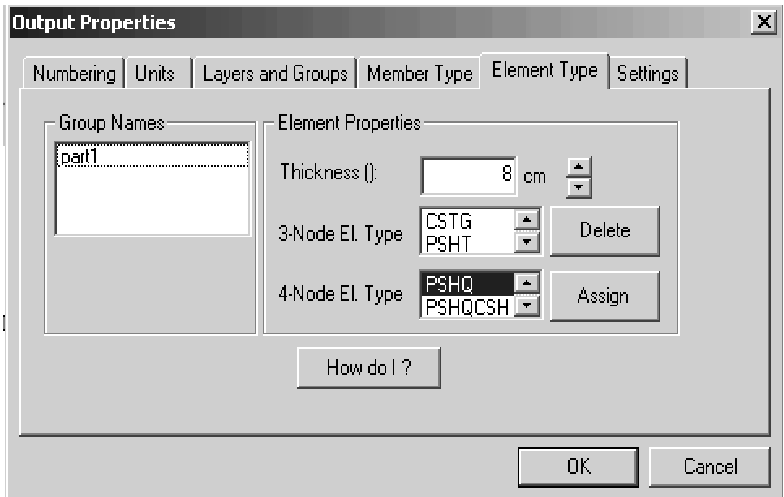
After selecting a DXF file using the Read button in the above dialog, you may assign Properties as shown in the following new Output Properties pop-up dialog:



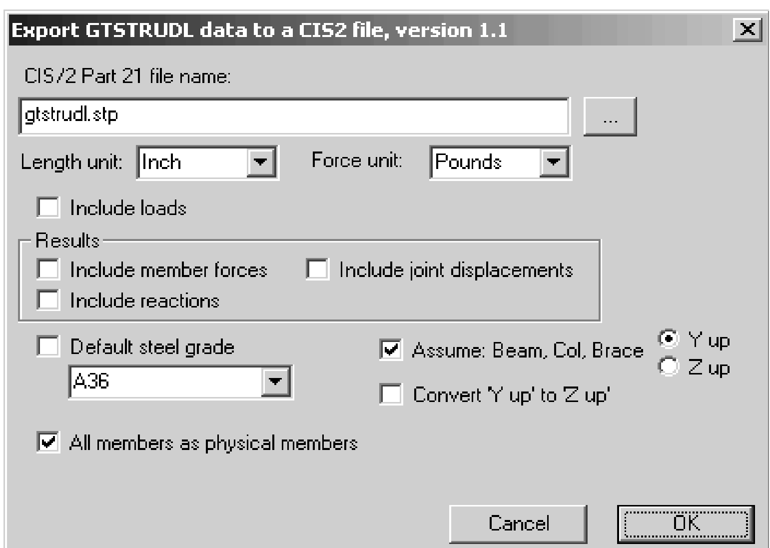
The Layers and Groups tab in the above dialog allows you to assign layers in the DXF file to GTSTRUDL Group names. The new Layers and Groups dialog is shown below:



Two dimensional finite elements can also now be created from a DXF file which contains closed polylines, 3Dface, and polygon mesh entities (Edge Surfaces in AutoCAD). The Element Type tab used to define the GTSTRUDL element type and the thickness is shown below:



New features have been added to the Export CIS/2 dialog as shown in the figure below:



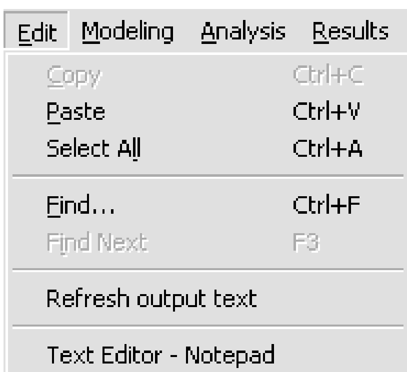
You may now specify the steel grade with the default being A36. You may also specify an assumed member type such as (beam, column, brace) and convert the vertical direction from Y to Z.

Another new feature has been added in the File pulldown which will allow you to view the current command history as shown below:

By selecting this option, a Notepad window will be opened showing all commands that have been given and that were automatically created from other dialog selections.

Edit Pulldown

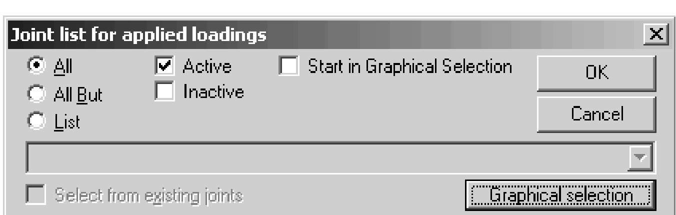
The Edit pulldown has changed as shown below:



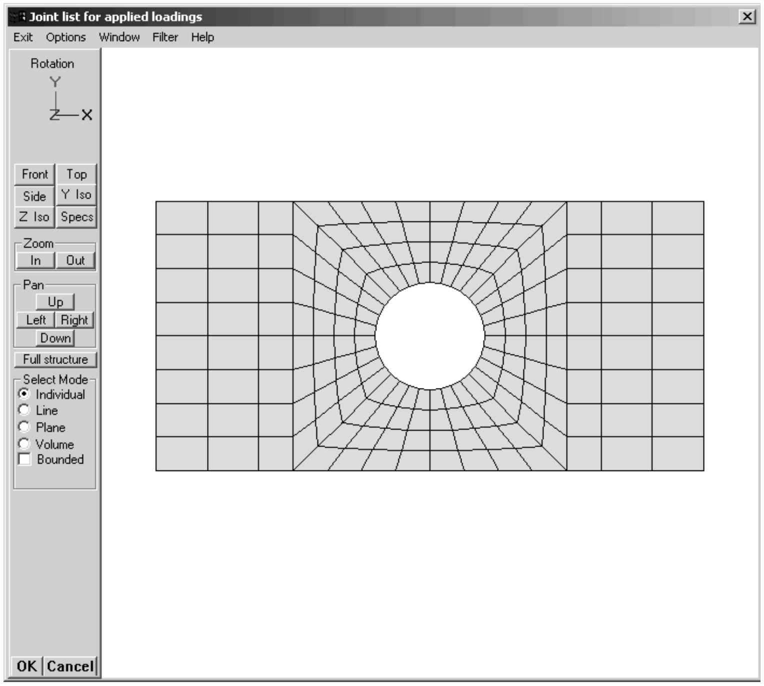
Selecting "Refresh output text" will refresh the output in the GTSTRUDL Output Window.   This is useful when lines of output in the Output Window are concatenated. Selecting "Text Editor - Notepad" will open a Notepad window.

Graphical Selection Feature

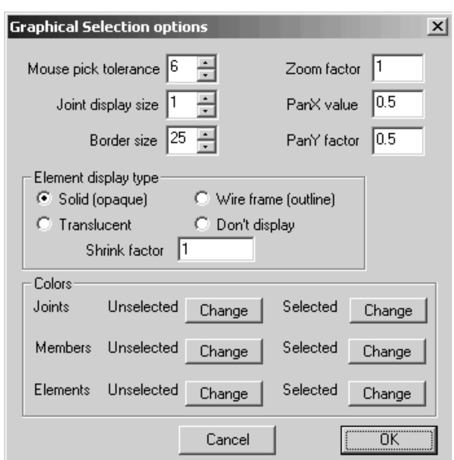
The Graphical Selection feature that is activated from list dialogs such as the one shown below now has the ability to also display and select finite elements.



An example of the display showing finite elements is shown below:

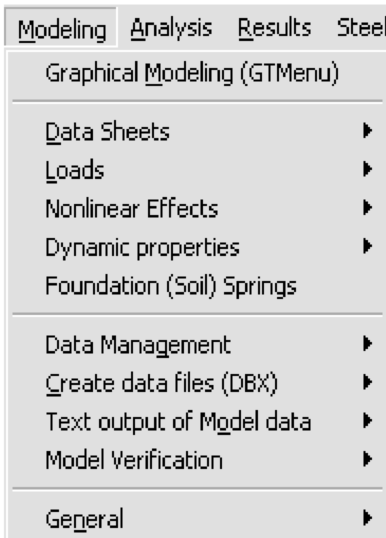


The Options dialog has been extensively modified so that you may now set the element display type as solid (as shown in the above figure), opaque, wire frame, or turn off the finite element display. You may also specify a shrink factor for the finite element display as well as set the colors for the selected and unselected elements. The new Options dialog is shown below:

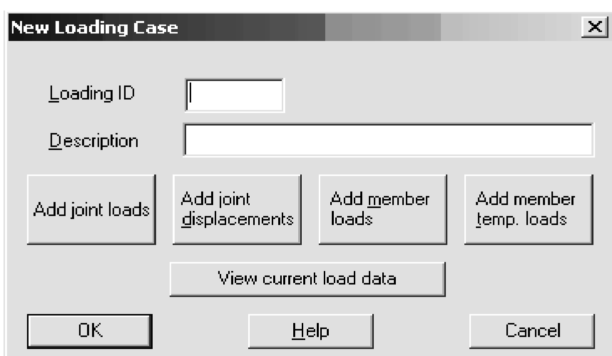


Modeling Pulldown

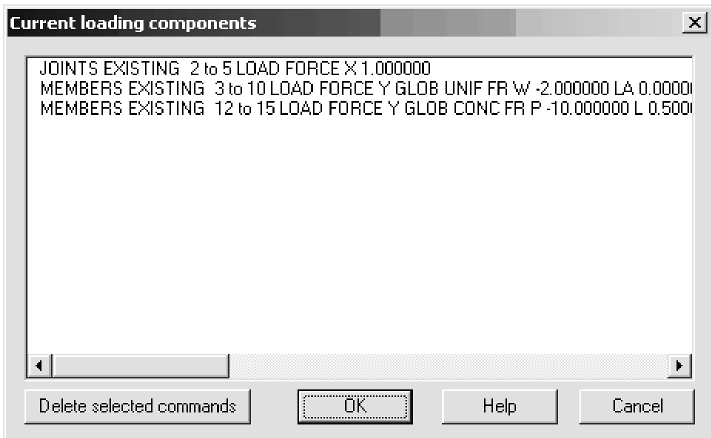
The Modeling pulldown has been restructured and new features have been added. The modified Modeling pulldown is shown below:



The "New Loading Case" option under Loads in the above pulldown has been modified to include the ability to view the data created in the dialog for joint loads, joint displacements, member loads, and member temperature loads as shown below:

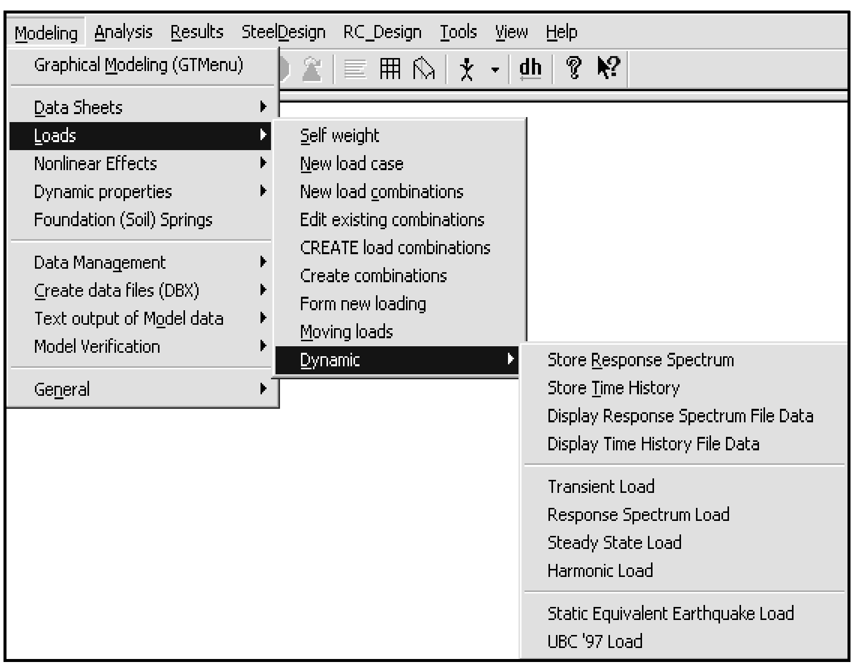


After creating data in the New Loading Case dialog shown above, clicking on the "View current load data" button will open the following pop-up dialog:



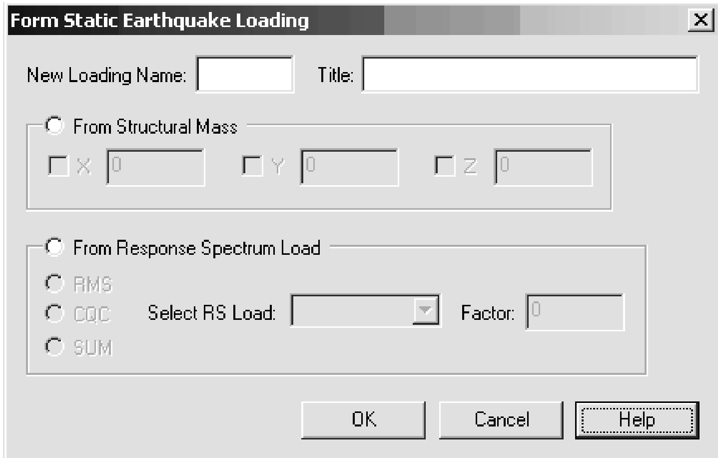
Using this pop-up, you may delete selected lines from the current load data.

New load options are available under Dynamic as shown below:



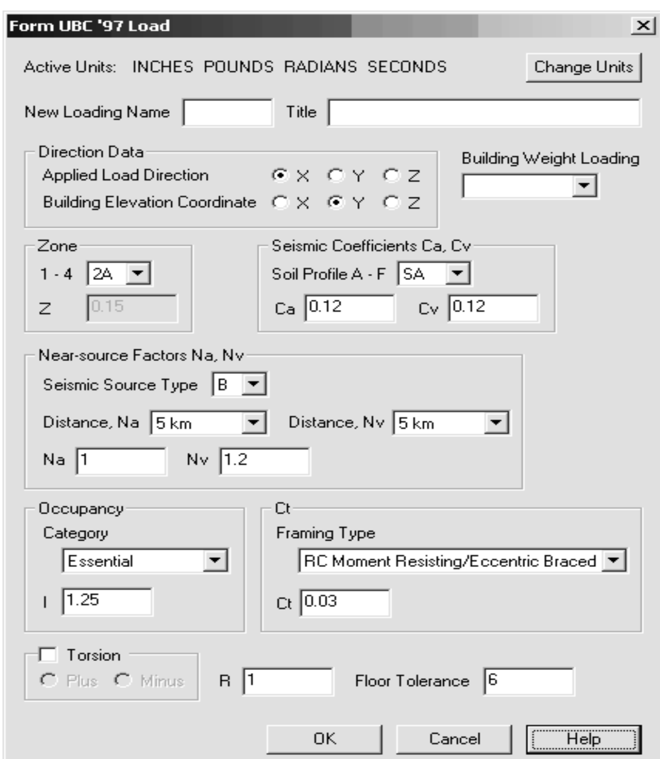
The Static Equivalent Earthquake Load option may be used to form a new independent static loading condition from either the structural mass or the results of a previous response spectrum analysis.

The dialog for this new feature is shown below:

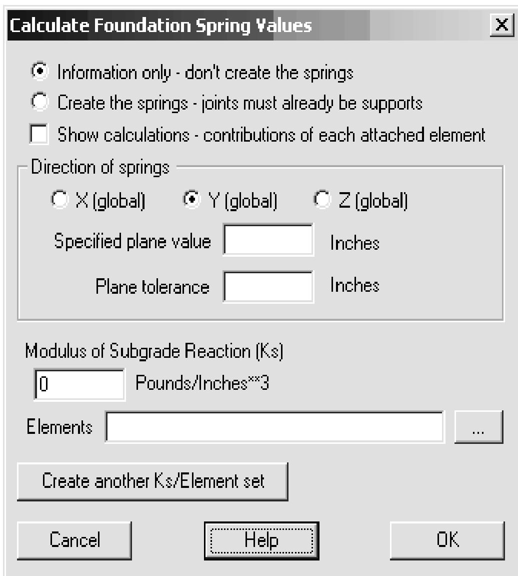


This dialog is used to form a new independent static loading condition from either the structural mass or the results of a previous response spectrum analysis. The new loading condition is intended to be a static representation of an earthquake load.

You may also form a new independent static loading condition based on the minimum design lateral seismic forces provisions of Section 1630 of the 1997 edition of UBC using the following new dialog:



A new dialog has been implemented to calculate elastic spring constants at a joint based on a modulus of subgrade reaction and the tributary area of finite elements attached to the joint. This new dialog is shown below:



Three options exist in this dialog:

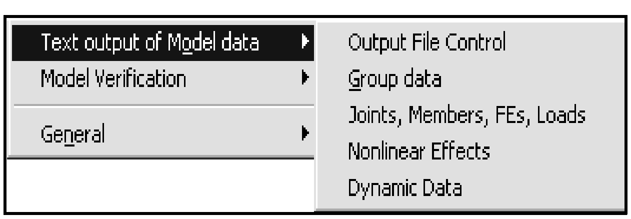
Information only - Spring calculation data (joint number and spring value) will be printed in a format that can be pasted into a GTSTRUDL input file, but the database will not be changed.

Create the springs - The same output as above will be printed, but the springs will be applied to the appropriate joints. Note that these joints must already be declared as supports.

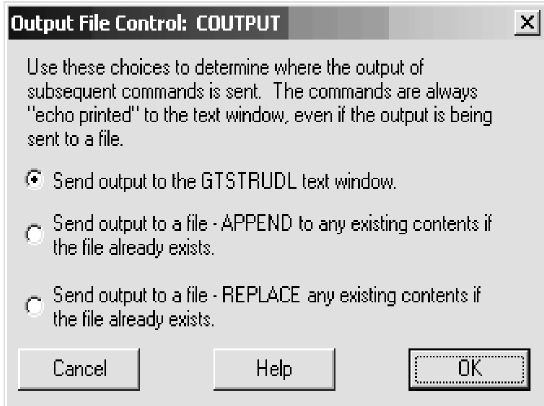
Show calculations - In addition to the output above, the contributions to the spring values from each finite element will be printed.

More information on this new feature may be found in Section 5.4.1 of this release guide.

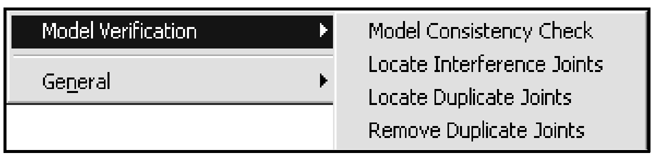
The Text Output option under Modeling has a new Output File Control option as shown below:



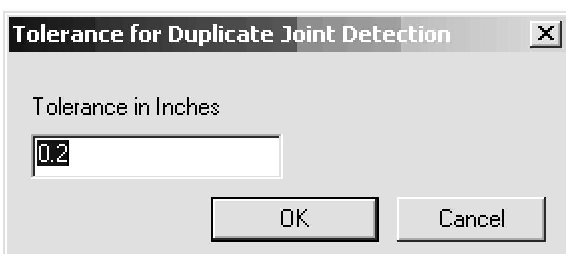
The new Output File Control dialog is shown below:



A new Model Verification option has been added as shown below:

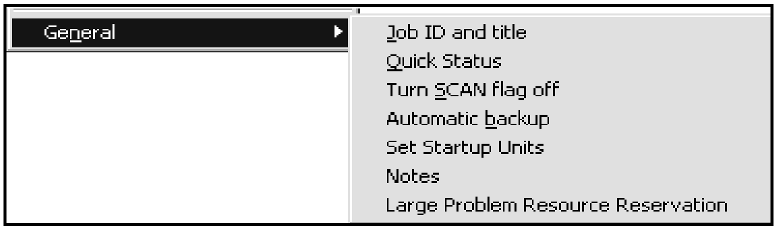


The Model Consistency Check option will perform a Consistency Check of the model for a static analysis. The Locate Interference option shown above has been available in previous releases of GTSTRUDL in the Analysis pulldown under "Analysis problems found". The Locate Duplicate Joints option and the Remove Duplicate Joints option shown above will pop-up the following dialog:



The Remove Duplicate Joints option will not only locate the duplicate joints within the specified tolerance but also remove them from the model.

The modified General pulldown is shown below:



The three new options are explained below:

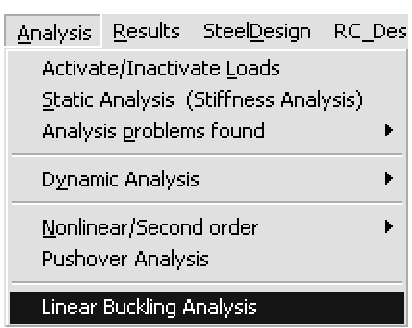
Set Startup Units
- will allow you to set the startup units that are used by GTSTRUDL for all users or just the current user.

Notes - allows you to specify text information that can be assigned a type or category, and additionally, be assigned to a set of components. The Notes option is described in more detail in Section 5.4.8 of this release guide.

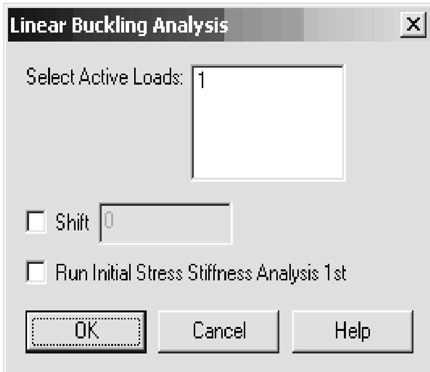
Large Problem Resource Reservation
- allows you to reserve Windows resources before GTSTRUDL actually uses the resources. This is done to help Windows work more efficiently and may significantly decrease processing time for large jobs. An example of the improvement from using this feature is presented in Section 2.5.

Analysis Pulldown

A new Perform Buckling Analysis option has been added to the Analysis pulldown as shown below and will allow you to perform a Linear Elastic Buckling Analysis.

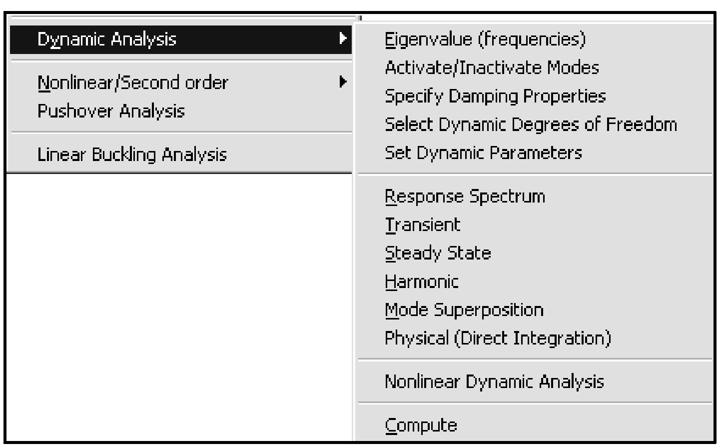


The Linear Buckling Analysis pop-up is shown below:

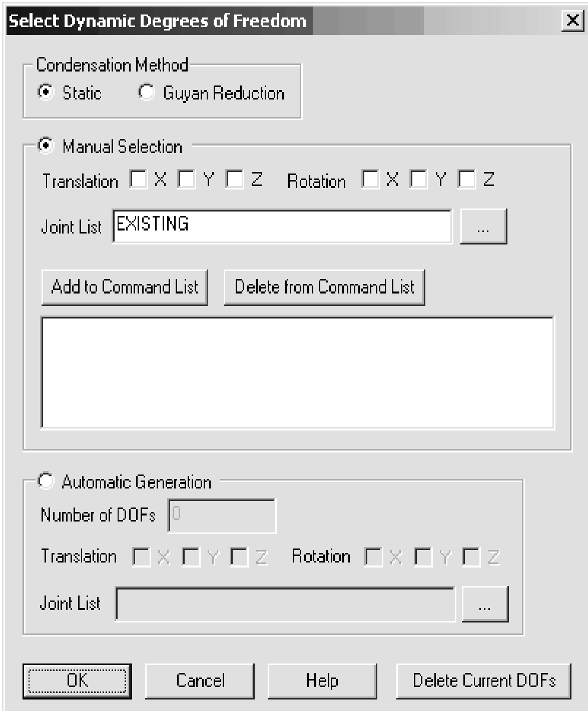


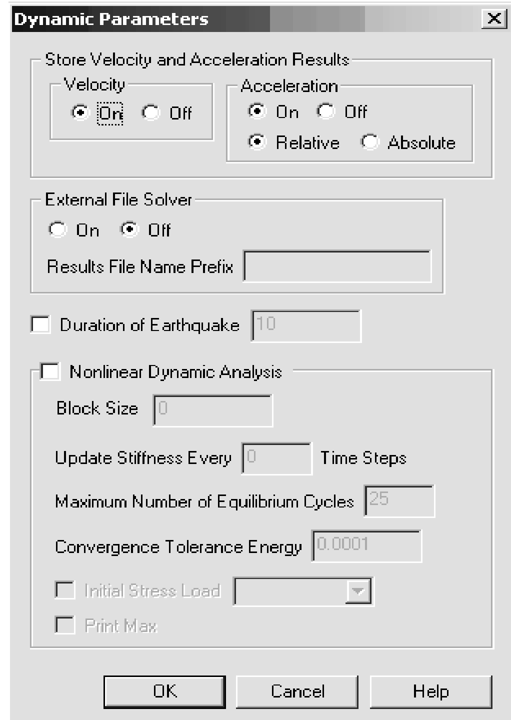
This dialog is used to execute a linear buckling analysis. A linear buckling analysis computes a buckling load multiplier and a corresponding buckling mode shape for each of a set of active static loading conditions, both independent and dependent. Results from a stiffness analysis for each of the selected active loadings must exist prior to execution of the buckling analysis. You may click the "Run Initial Stress Stiffness Analysis 1st" check box to execute a stiffness analysis prior to executing the linear buckling analysis.

The Dynamic Analysis option has also been modified as shown below:



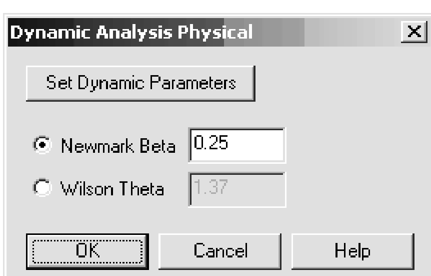
New dialogs have been added to Select Dynamic Degrees of Freedom and to Set Dynamic Parameters as illustrated below:



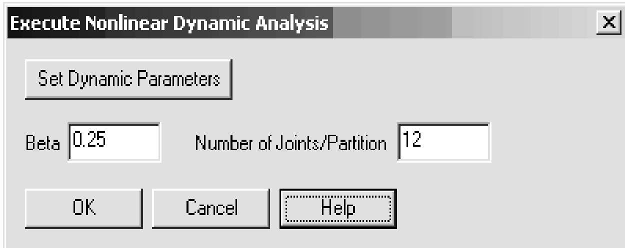


The Select Dynamic Degrees of Freedom dialog is used to specify those degrees of freedom to be retained for a dynamic analysis solution. The Dynamic Parameters dialog is used to specify miscellaneous control parameters for various types of dynamic analyses, including general linear and nonlinear transient analyses and response spectrum analysis.

A new Dynamic Analysis Physical dialog as shown below has also been added to initiate a direct integration transient dynamic analysis:



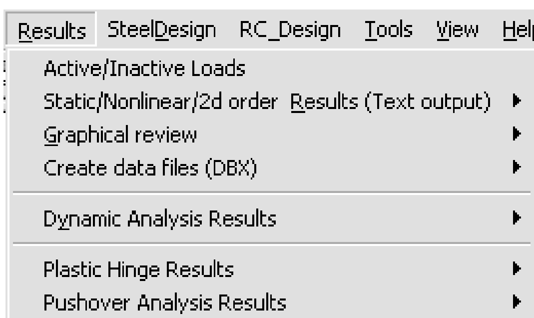
The following new Nonlinear Dynamic Analysis dialog has also been added:



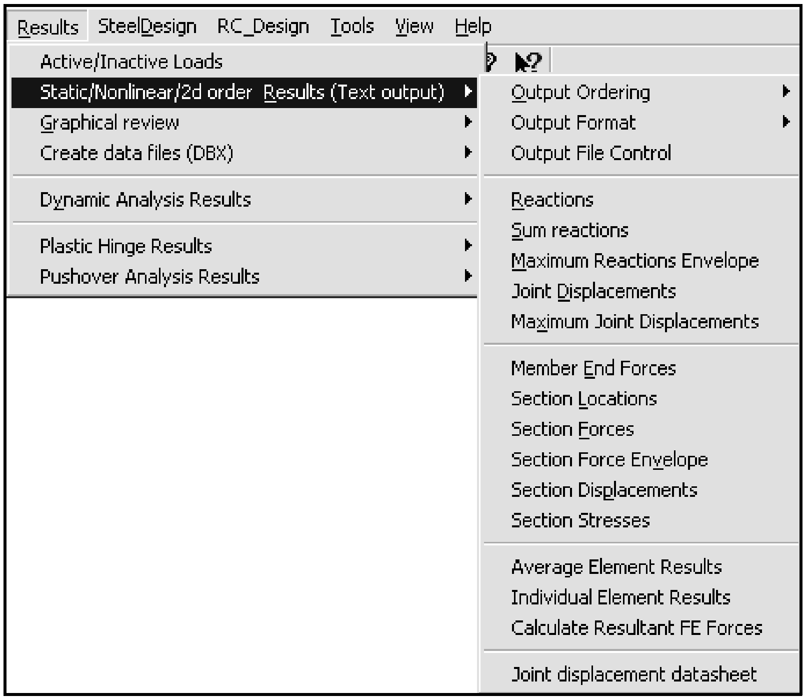
Nonlinear Dynamic Analysis is a prerelease feature and is described in Section 5.3.12 of this release guide.

Results Pulldown

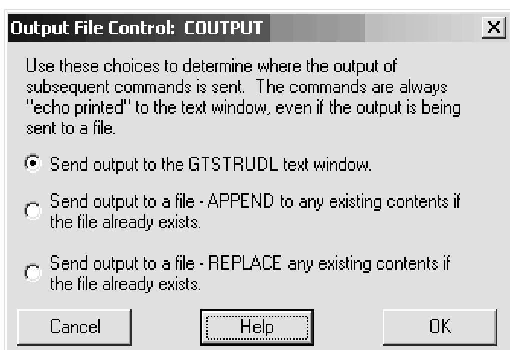
The Results Pulldown has been modified with new options for plastic hinge and pushover analysis results as shown below:



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

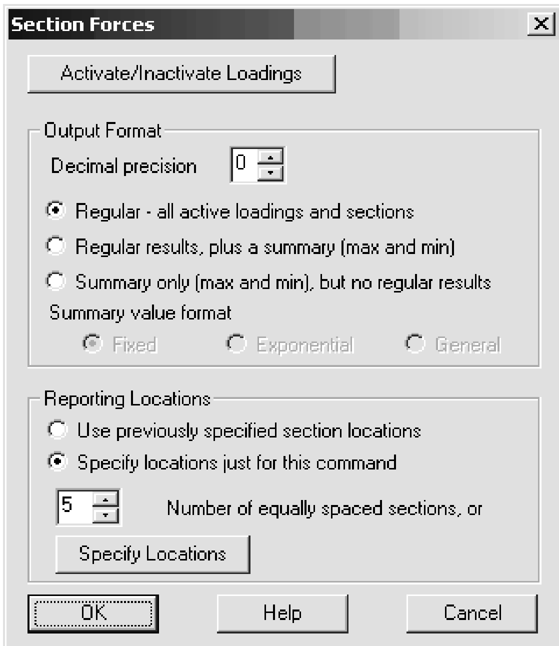


The new Output File Control dialog is shown below:



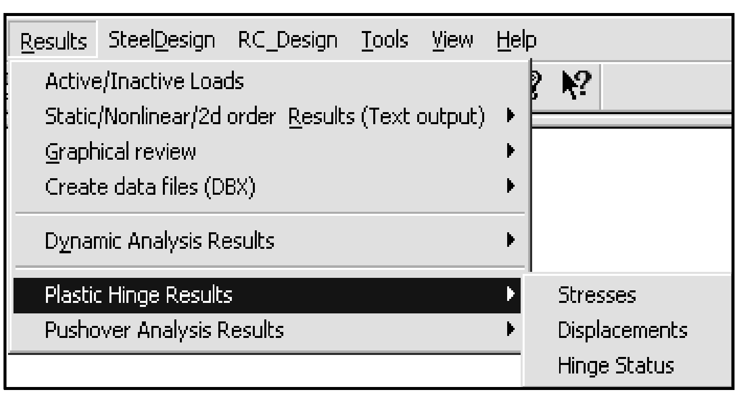
This dialog allows you to control the location of GTSTRUDL text output from PRINT or LIST commands with the default location of the output being the GTSTRUDL Output Window. This dialog includes the new REPLACE option discussed in Section 2.7.

The Section Forces dialog has a new option which will allow you to obtain a summary of the maximum and minimum values for each member. You may request the regular section force or stress output plus the summary output or just obtain the summary information. The modified Section Force dialog is shown below:



A new Section Stresses dialog has also been added. This dialog is similar to the Section Forces dialog shown above and also has the new summary options.

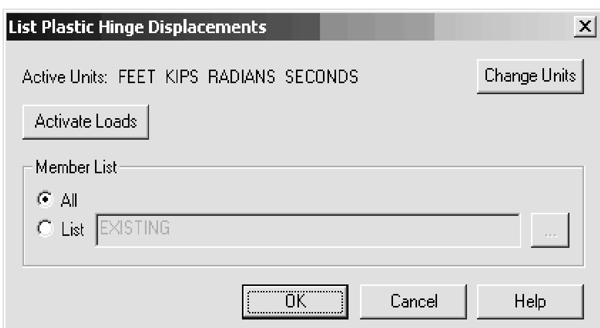
The new Plastic Hinge Results options are shown below:



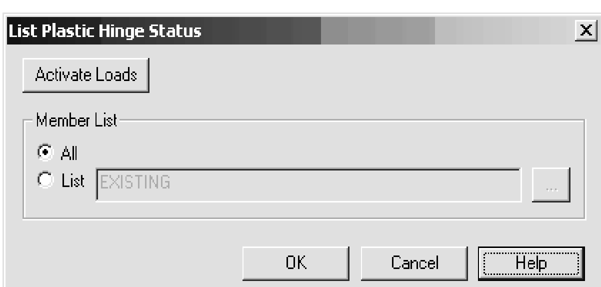
The following dialog allows you to output plastic hinge stresses:

This dialog is used to list the plastic hinge fiber stress results from independent loading conditions that were used for previous nonlinear and pushover analyses. You can specify output units, activate loads, and specify filters based on plastic hinge fiber characteristics and the member list.

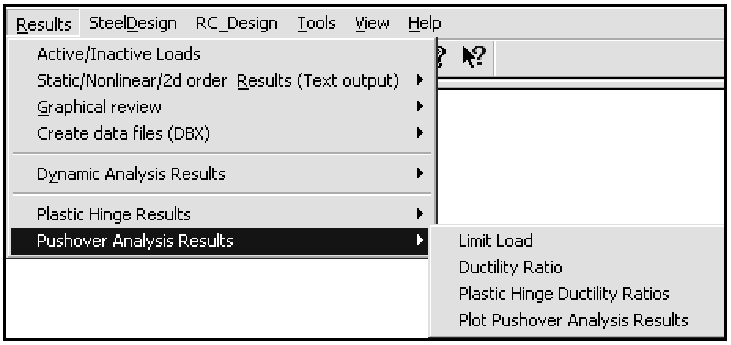
The plastic hinge relative displacements may be listed using the following new dialog:



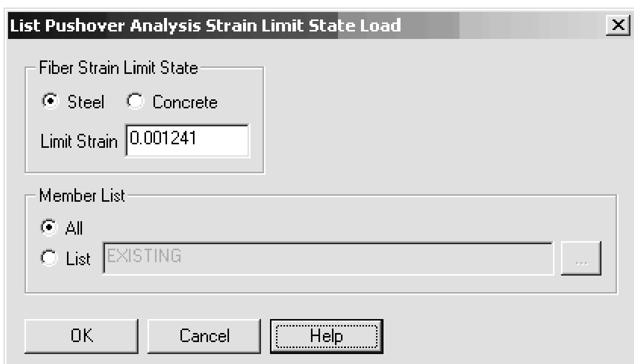
The plastic hinge status which is a cross section area percentage of the plastic hinge fibers that have reached or exceeded a predefined yield strain for the fiber material, may be listed using the following new dialog:



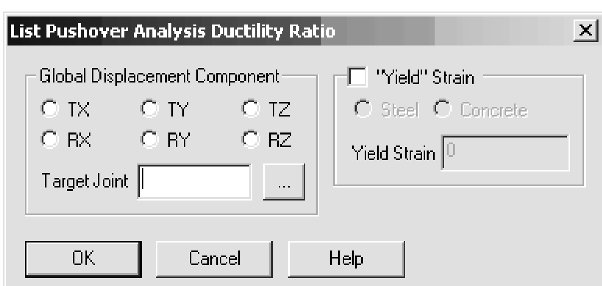
The new Pushover Results options are shown below:



You may output the limit load, ductility ratio, and plastic hinge ductility ratios using the following new dialogs:

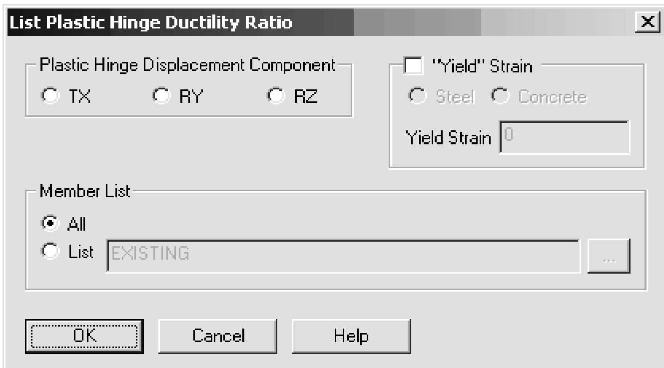


The above dialog is used to print a list of the names of the pushover analysis incremental loads at which the specified fiber limit state strain is first achieved by each plastic hinge in the specified members.

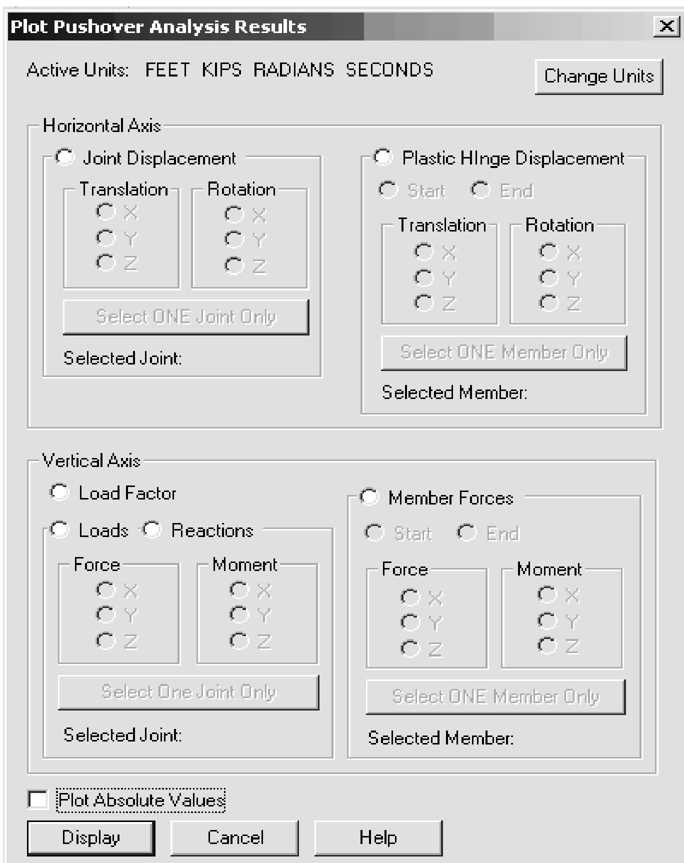


This dialog is used to calculate and print the global displacement ductility ratio for the selected degree of freedom at the specified target joint, based on the most recent pushover analysis.

This dialog is used to calculate and print the local ductility ratio for the selected relative displacement degree of freedom for the plastic hinges of the specified members, based on the most recent pushover analysis.

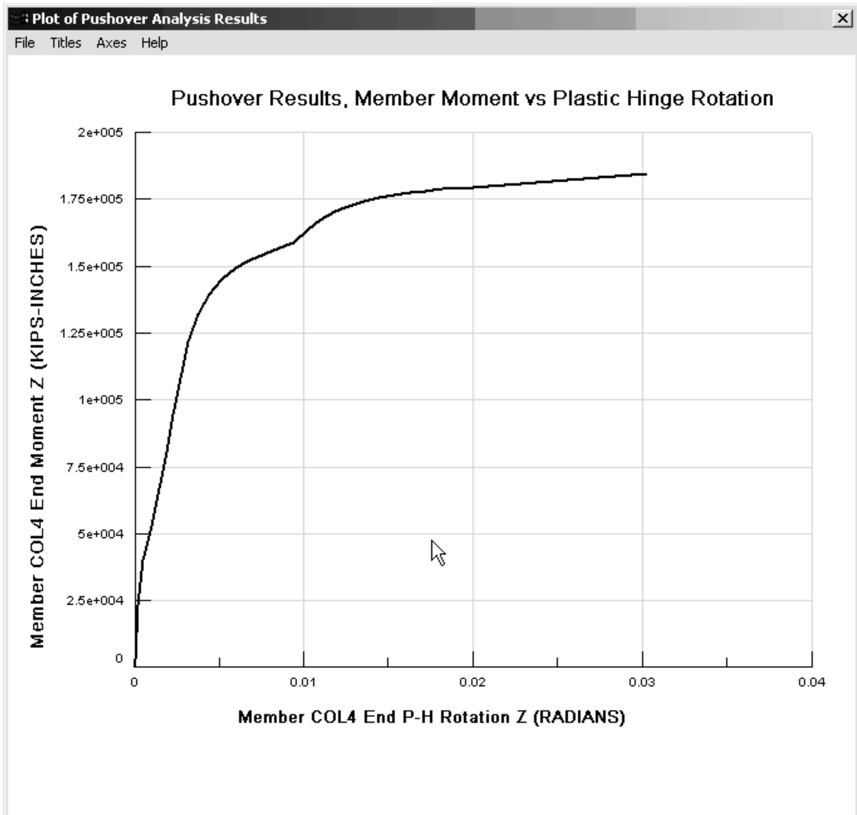


You may now graphically display pushover analysis results using the following new dialog:



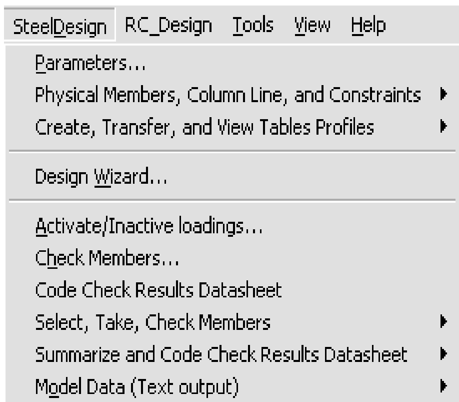
Using this dialog, you can plot the Load Factor, Loads, Reactions, or Member forces on the vertical axis. You can plot a Joint Displacement or a Plastic Hinge Displacement on the horizontal axis. Any combination of vertical axis vs horizontal axis results may be plotted. The active units can be changed for the results plot.

An example of a pushover plot is shown below:

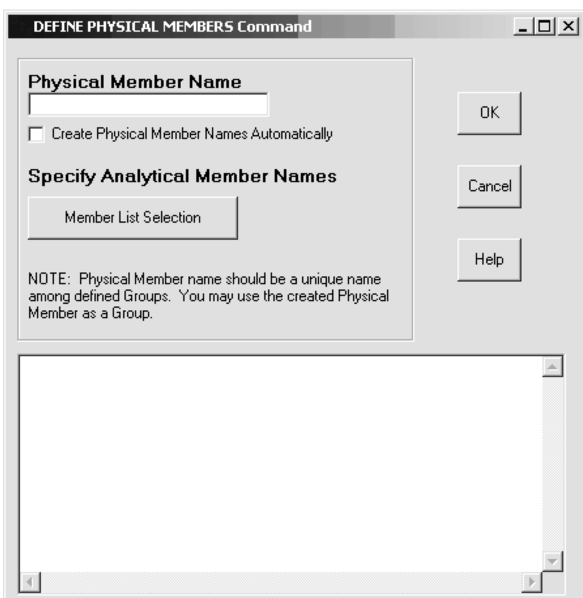


Steel Design Pulldown

The Steel Design Pulldown has been reorganized as shown in the figure below:

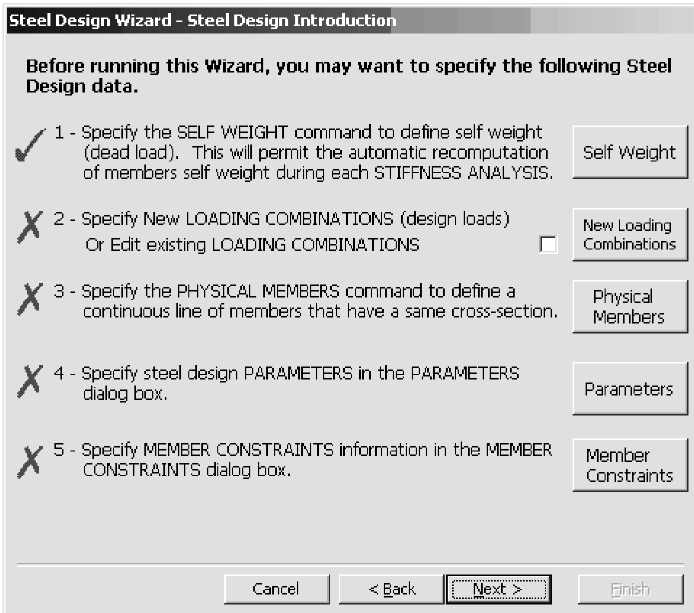


A new option has been added to the Physical Member dialog to automatically create physical member names as shown below:

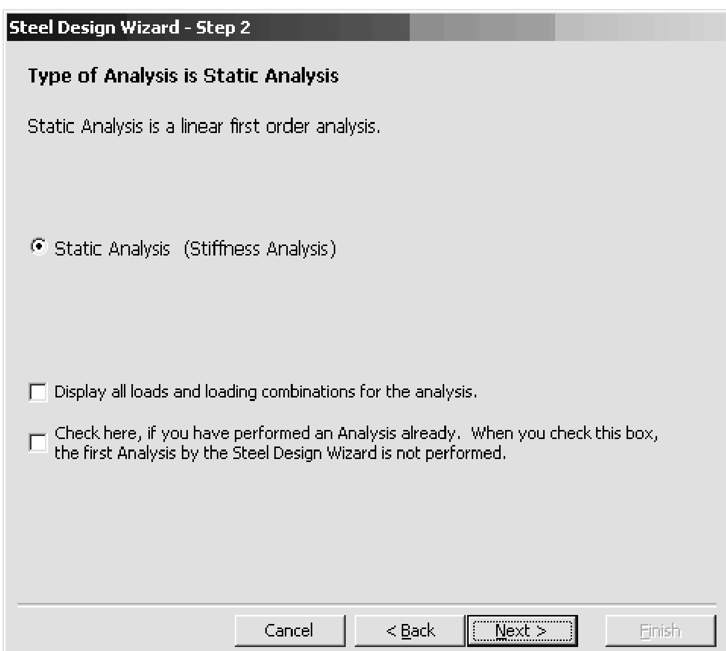


Column lines are no longer needed for the K factor computation. As a result, the Column Line option has been removed from the Steel Design Wizard.

New Loading Combination and Edit Load Combination dialogs have been added to the Steel Design Wizard as shown below:

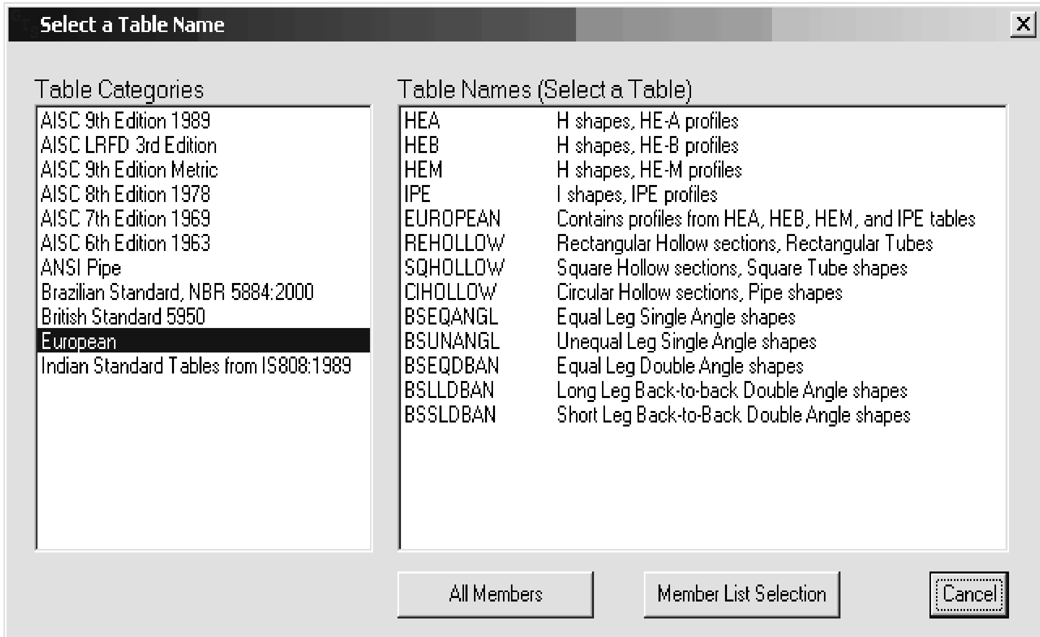


The load display page in the Steel Design Wizard for Stiffness Analysis is now optional as shown in the following dialog from the Steel Design Wizard:

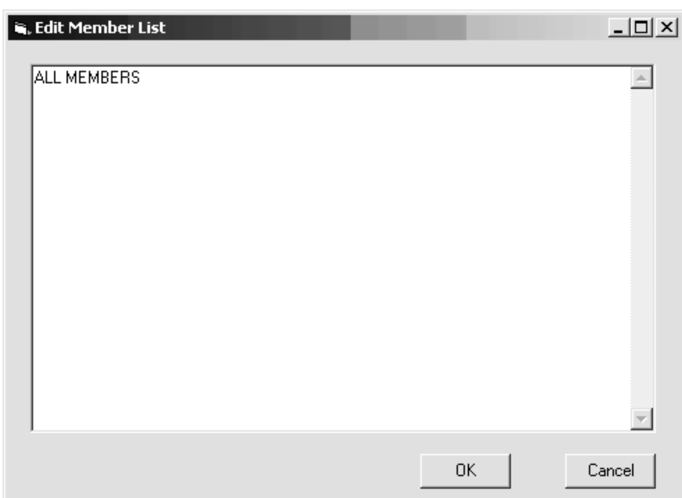


The user can check the check box in the above dialog to activate the load display page and select which loads are to be active and inactive for the next analysis.

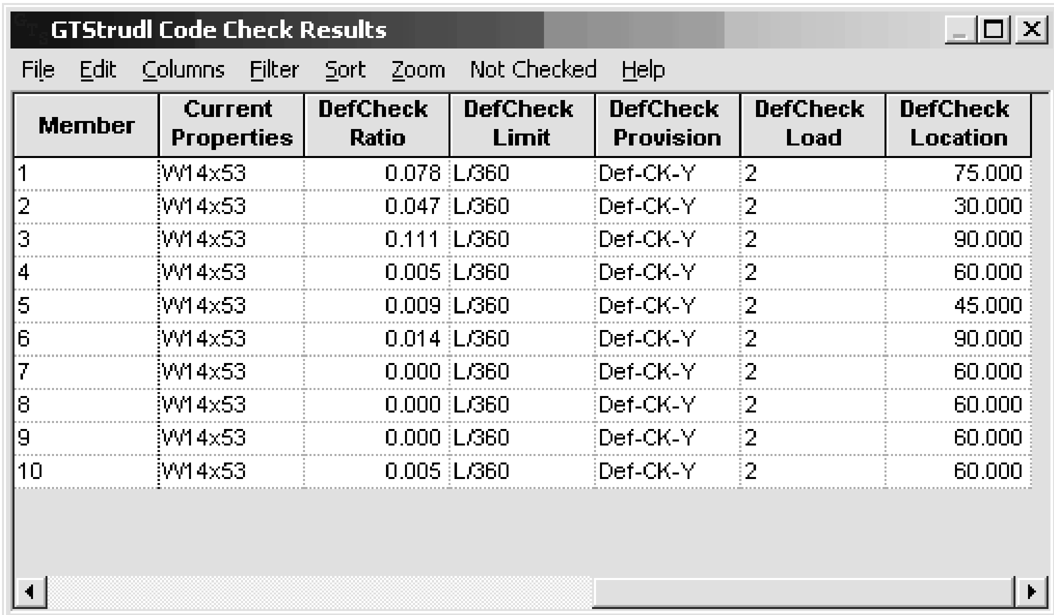
A new selection dialog for the parameter TBLNAM has been added into the Parameter dialog which displays GTSTRUDL Table Category that corresponds to the code selected by the user. Shown below is this new dialog with the available tables for the European EC3 code indicated:



When editing the member list in the PARAMETER dialog, a new text box shown below has been added to facilitate the editing of the member list:



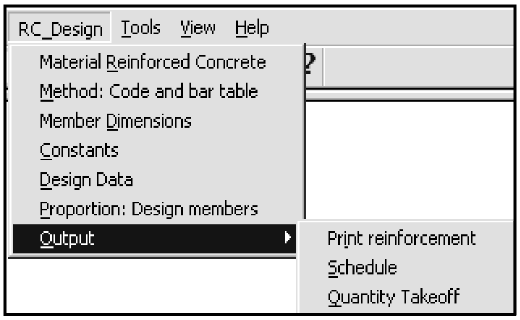
The Code Check Results datasheet now contains information from a Deflection Check as shown below:



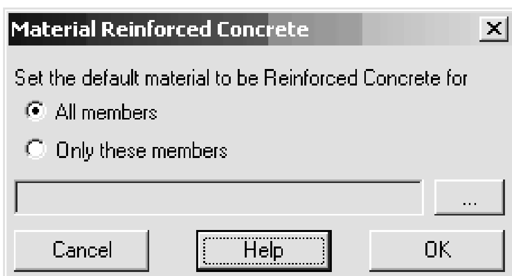
Member	Current Properties	DefCheck Ratio	DefCheck Limit	DefCheck Provision	DefCheck Load	DefCheck Location
1	W14x53	0.078	L/360	Def-CK-Y	2	75.000
2	W14x53	0.047	L/360	Def-CK-Y	2	30.000
3	W14x53	0.111	L/360	Def-CK-Y	2	90.000
4	W14x53	0.005	L/360	Def-CK-Y	2	60.000
5	W14x53	0.009	L/360	Def-CK-Y	2	45.000
6	W14x53	0.014	L/360	Def-CK-Y	2	90.000
7	W14x53	0.000	L/360	Def-CK-Y	2	60.000
8	W14x53	0.000	L/360	Def-CK-Y	2	60.000
9	W14x53	0.000	L/360	Def-CK-Y	2	60.000
10	W14x53	0.005	L/360	Def-CK-Y	2	60.000

RC_Design Pulldown

A new reinforced concrete design pulldown has been added to this release of GTSTRUDL. The options available under the RC_Design Pulldown are shown below:

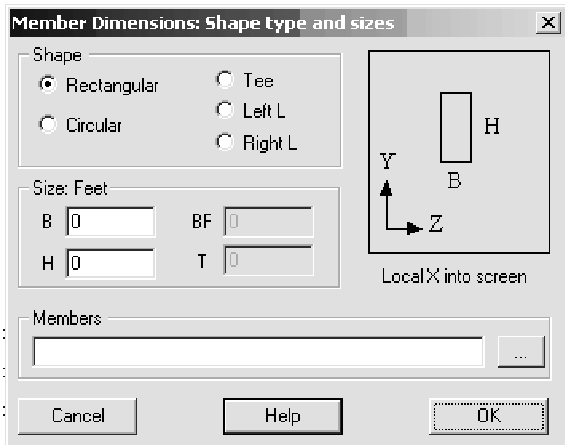


The following Material Reinforced Concrete dialog allows you to initialize standard concrete and reinforcing steel parameters and must be selected before designing or checking members.

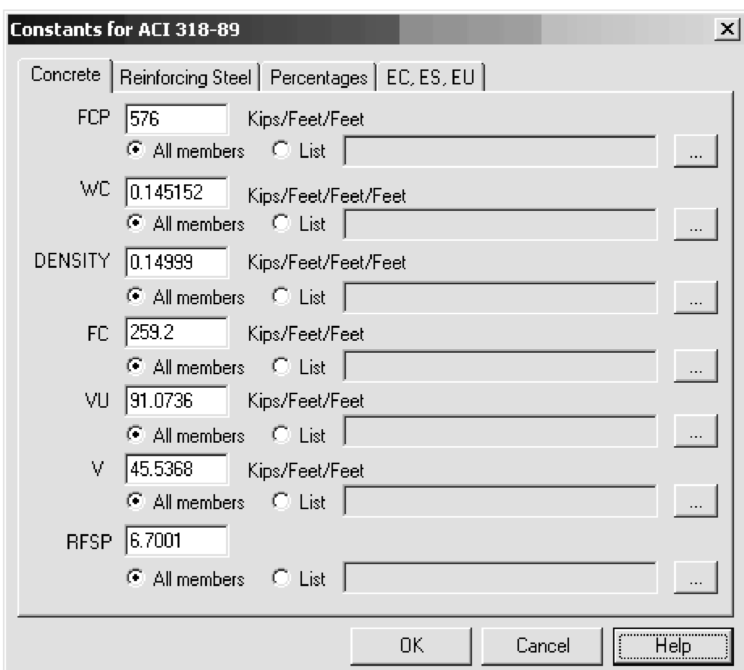


The Method dialog shown below is used to specify the concrete code and the reinforcing bar table to be used in a subsequent design:

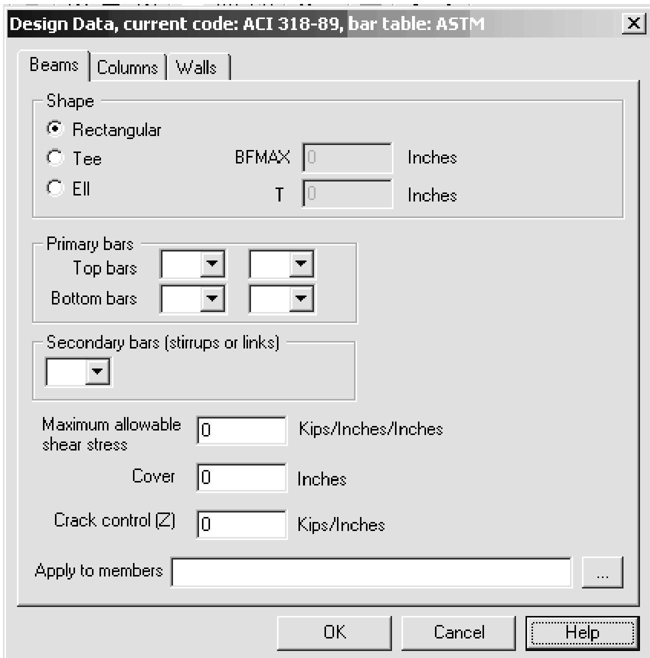
The following Member Dimension dialog is used to specify dimensions of concrete members for five different cross sections (rectangle, circle, tee, RL, and LL):



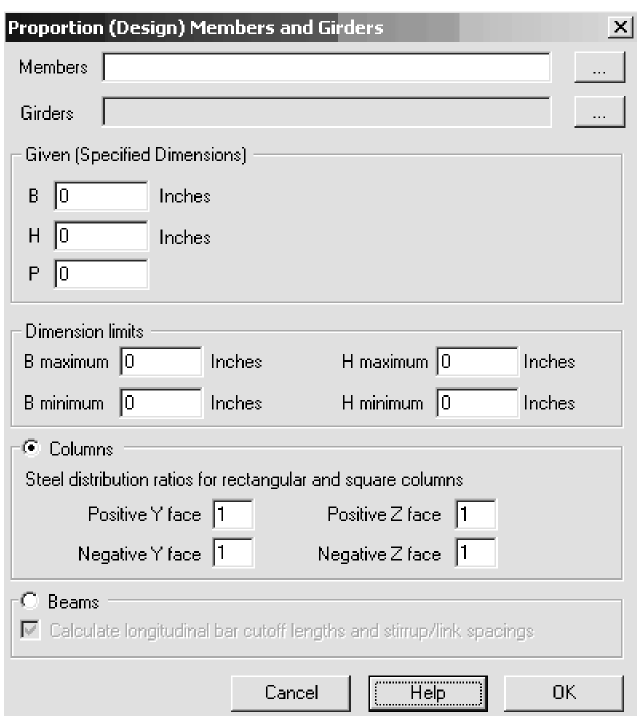
The Constants dialog allows you to specify additional constants which may be needed for the design depending on the selected design code. An example of the Constants dialog for the ACI-318-89 code is shown below:



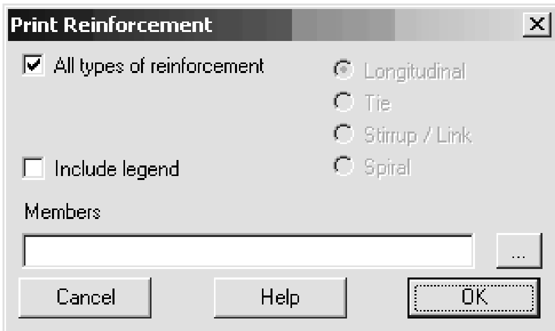
The Design Data dialog shown below allows you to specify the type of design member (beam, column, wall), shape, bar sizes to use, cover, and other considerations:

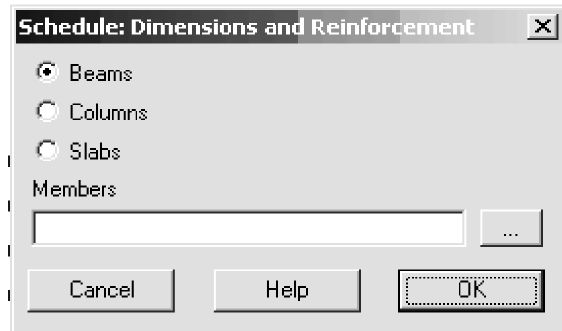


The following Proportion dialog designs the members based on the code, constants, and design data:



Results of the concrete design may be output using the following dialogs which output the reinforcement and schedules:





You may also output a quantity takeoff by clicking on the Quantity Takeoff option under Output.

Other Improvements

1. An “Activate/Inactivate loadings” button has been added to the Section Displacements dialog.
2. Datasheets now alert the user if the number of data entries exceeds the display limit.
3. The Active Modes dialog for Dynamics has been made larger and a new “Automatic Inactivation” feature added.
4. Component lists (joints, members, loads) are now written with the ‘n TO m” format for sequential series of names; for example, “1 2 3 4 5" now becomes “1 TO 5".
5. The maximum length of a component list has been increased to 1500 from the previous limit of 500.
6. A history of user library files (userdat.ds) is now kept.
7. Windows 2000 and XP now respect the Working Directory when opening a file selection dialog.
8. CABLE elements are displayed, but are not selectable in Graphical Selection.
9. The menu bar and some ‘File’ menu selections are ‘grayed out’ (not selectable) while GTMenu is running.
10. The version number of the GTSTRUDL Solver (e.g. “27.01") and completion date are now displayed in the “Help → About GTSTRUDL” box.
11. FINISH statistics are now more reliably printed. Previously, the last few lines of output would not be displayed on fast computers.

2.3 DBX

1. The WRITE MEMBER PROPERTIES LONG NAME function now uses a caching technique to speed up the file writing process. For large numbers of members, the speedup can exceed 80%, i.e. from more than 5 seconds to less than 1 second. The efficiency of the caching technique increases when there are many members with the same long names.

2.4 Dynamics

1. The LIST RESPONSE SPECTRUM command has been extended to include the BASE/STORY SHEAR option. 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 applied at 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.

2. The specification of spring stiffness damping factors in the JOINT RELEASES command has been expanded to allow for the specification of a separate damping factor for each individual spring stiffness value.

3. A new FORM UBC97 LOAD command has been implemented for the purpose of automatically generating a static lateral seismic load according to the minimum seismic load provisions of 1997 Uniform Building Code, Volume 2, Section 1630. See Section 5.3.6 of the Release Guide for more details.

4. Memory allocation has been significantly improved in modal time history analysis. Previously, memory allocation became particularly acute for the case when the DYNAMIC PARAMETERS STORE ABSOLUTE ACCELERATION and USE EXTERNAL FILE SOLVER commands had been given. The improved memory allocation has resulted in dramatic speed improvements of two to three orders of magnitude.

5. A new FORM IS1893 LOAD command has been implemented for the purpose of automatically generating a static load according to the Indian Standard IS1893 (Part I). See Section 5.3.7 of the Release Guide for more details.

6. A new element, the viscous damper or dash pot element, has been implemented for use in conjunction with the DYNAMIC ANALYSIS PHYSICAL and DYNAMIC ANALYSIS NONLINEAR commands. The viscous damper element provides for the modeling of viscous damping forces at any joint or between any two joints in the structure, with the

exception of slave joints. The damping properties of the viscous damper elements are assembled into the global system damping matrix of the equations of motion that are solved using the direction integration methods executed by the DYNAMIC ANALYSIS PHYSICAL and DYNAMIC ANALYSIS NONLINEAR commands.

7. The output from the LIST TRANSIENT/HARMONIC MAXIMUM command has been improved to include an additional line in which the integer time or frequency point number for each maximum response value is listed. Below is an example of the new format for LIST TRANSIENT MAXIMUM output:

```
{   117} > LIST TRANSIENT MAXIMUM VELOCITY JOINTS 2 4

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

    PROBLEM - DY401     TITLE - ST4611 -- Improve LIST TRANSIENT MAXIMUM time point output forma

    ACTIVE UNITS  INCH KIP  RAD  DEGF SEC

==============================================================================================================================
 LOADING -  2
==============================================================================================================================


     MAXIMUM JOINT VELOCITIES AT THE FREE JOINTS

JOINT                        /---------------------TRANS--------------------//---------------------ROTATION-----------------/
                                  X-TRANS          Y-TRANS          Z-TRANS        X-ROTATION       Y-ROTATION       Z-ROTATION

2         GLOBAL    MAXIMUM     2.059913       0.0000000E+00                                                      -0.5018346E-03
                     TIME       2.225000       0.0000000E+00                                                        2.205000
                    TIME PT       446                0                                                                 442

4         GLOBAL    MAXIMUM     2.336218       0.0000000E+00                                                      -0.5073535E-03
                     TIME       2.220000       0.0000000E+00                                                        2.205000
                    TIME PT       445                0                                                                 442
```

8. The CREATE RESPONSE SPECTRUM command has been expanded to include a PERIOD option that provides for the creation of acceleration, velocity, or displacement versus period response spectrum files in addition to the original frequency based files.

9. The following error message has been eliminated for dynamic analyses with consistent mass:

**** ERROR_THCE1 -- CONSISTENT MASS NOT AVAILABLE FOR BPHT, BPHQ, OR BENDING COMPONENT OF SBHT,SBHQ,SBHT6,SBHQ6 ELEMENTS LUMPED MASS WILL BE USED FOR CONSISTENT MASS

Many users had complained about this message and Table 2.4.3.3.1 of Volume 3 of the GTSTRUDL Reference Manuals indicates that the lumped mass will be used for the elements mentioned in the above message.

2.5 Efficiency Improvements

1. A new LARGE PROBLEM command has been implemented to reserve Windows resources before GTSTRUDL actually uses the resources. This is done to help Windows work more efficiently and may significantly decrease processing time for large jobs. When GTSTRUDL completes, all the Windows resources are released.

General Form:

LARGE (PROBLEM) SIZE i_1

Elements:

i_1 = an integer between 1 and 5 (inclusive). 5 is the largest problem size and may impact the operation of other programs running simultaneously with GTSTRUDL.

An example between Version 27 using Large Problem Size 5 and Version 26 is shown below:

Example:

27676 Joints
24540 Tridimensional elements
1 loading
Average + std. deviation of band =772

	Version 27	Version 26
TIME FOR CONSISTENCY CHECKS FOR 24540 MEMBERS	2.35	5.35
TIME FOR BANDWIDTH REDUCTION	8.78	63.59
TIME TO GENERATE 24540 ELEMENT STIF. MATRICES	28.28	344.04
TIME TO ASSEMBLE THE STIFFNESS MATRIX	5.76	35.30
TIME TO PROCESS 27676 JOINTS	0.61	17.78

TIME TO SOLVE WITH 2307 PARTITIONS	1484.51	1804.58
TIME TO PROCESS 27676 JOINT DISPLACEMENTS	1.97	22.60
TIME TO PROCESS 24540 ELEMENT STRESSES	16.23	58.76
TIME TO PROCESS 24540 ELEMENT REACTIONS	90.65	25.92
TIME FOR STATICS CHECK	3.45	14.59
Total CPU Time	1967	2745

This new command is described in more detail in Section 5.4.2.

2. The RESTORE command has been made more efficient especially for large models. Improvements of up to 95% have been experienced for large save files (.gts files). This increase in speed also occurs when Opening or Restoring .gts files using the dialogs in the GTSTRUDL Output Window and in GTMenu.

2.6 Finite Elements

1. A new command, CALCULATE SOIL SPRING VALUES, has been added as a prerelease feature in Version 27. This command allows you to calculate elastic spring constants for support joints based on a specified modulus of subgrade reaction and the tributary area of finite elements attached to the joint.

General Form:

$$\underline{\text{CALC}}\text{ULATE }(\underline{\text{SOI}}\text{L})\ \underline{\text{SPR}}\text{ING }(\underline{\text{VAL}}\text{UES})\ \underline{\text{DIR}}\text{ECTION} \left\{\begin{matrix}\underline{\text{X}}\\ \underline{\text{Y}}\\ \underline{\text{Z}}\end{matrix}\right\} v_1 \ -$$

$$((\underline{\text{PLA}}\text{NE})\ \underline{\text{TOL}}\text{ERANCE } v_2)\ ((\underline{\text{PLA}}\text{NE})\ \underline{\text{ANG}}\text{LE }(\underline{\text{TOL}}\text{ERANCE})\ v_3)\ -$$

$$\text{element value\_sets }(\underline{\text{APP}}\text{LY})\ (\underline{\text{SHO}}\text{W }(\underline{\text{CAL}}\text{CS}))$$

where element value_sets contain sets of subgrade reactions, values and corresponding lists of finite elements.

The following finite elements are supported by this command:

Plate bending

CPT, BPHT, BPR, BPP, BPHQ, IPBQQ

Plate

SBCT, SBCR, SBHQ, SBQCSH, SBHT, SBHT6, SBHQ6

Tridimensional (Solids)
TRIP, IPLS, IPQS, IPSL, IPSQ

This command is described in more detail in Section 5.4.1 of this Release Guide.

2. The warning message concerning the integration order not being specified for the IPQL, IPQL2, IPQL2B, IPQLQ3, IPQLQ4, IPSL, IPSQ, and TRANS3D elements will no longer be output. Many users had complained about these warning messages. The default integration order used for these elements is described in Section 2.3.5 of Volume 3 of the GTSTRUDL Reference Manual.

3. Temperature gradient loading capabilities have been added as a prerelease feature for the following elements:

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

The revised Finite Element Dictionary Table for these elements may be found in Section 5.3.2.

2.7 General

1. A new command, ROTATE LOADING, has been implemented as a new option that can be given under the independent loading commands LOADING, DEAD LOAD, and FORM LOAD. This command specifies another independent loading condition from which the global applied force and moment components are rotated and copied into the destination loading specified by the preceding LOADING, DEAD LOAD, or FORM LOAD command.

2. A new NOTE feature in GTSTRUDL is text information that can be assigned a type or category, and additionally, be assigned to a set of components. For example, a general note may contain the information "Model created from CAD file C:\Project\Plans.cad". Member 1 could have a note of category 'PieceMrk', with contents of "A217-3456-BFG".

Notes are created with the new NOTE command, and reviewed with new options for the PRINT command.

```
                 ⎧ GENERAL or MODEL ⎫
                 ⎪ JOINTS list      ⎪       ⎧ CATEGORY      ⎫
NOTES (i) (FOR   ⎨ MEMBERS list     ⎬ ) (   ⎨               ⎬ )  -
                 ⎪ LOADS list       ⎪       ⎩ TYPE category ⎭
                 ⎩ GROUPS list      ⎭
                 (Text) note text
```

A PRINT NOTE command has also been added to output the notes associated with joints, members, loads, and groups.

```
                    ⎧ GENERAL or MODEL ⎫
                    ⎪ JOINTS list      ⎪      ⎧ CATEGORY      ⎫
PRINT NOTES (FOR    ⎨ MEMBERS list     ⎬ ) (  ⎨               ⎬ )
                    ⎪ LOADS list       ⎪      ⎩ TYPE category ⎭
                    ⎩ GROUPS list      ⎭
```

More information regarding the NOTES and PRINT NOTES commands may be found in Section 5.4.8.

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

```
            ⎧ → APPEND   ⎫
COUTPUT  (  ⎨   REPLACE  ⎬  ) ('file_name')
            ⎩   STANDARD ⎭
```

where,

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

More details regarding the COUTPUT command may be found in Section 5.4.7.

4. 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

   RUN ((BATCH) (KEEP)) (WAIT) '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.

   More details regarding the RUN command may be found in Section 5.4.6.

5. The PRINT MEMBER DIMENSIONS command no longer requires the MATERIAL REINFORCED CONCRETE command to be given in order to output Member Dimension data.

6. Users can now change the starting units for GTSTRUDL, instead of having to start with active units of INCH, POUNDS, RADIANS, DEGF and SECONDS. Users can set any unit by creating a string Registry value in “Software\CASE\Settings”, etc. A new GTSTRUDL Output Window dialog entry has been added, ‘Modeling -> General -> Set Startup Units’, to make this easier. You can set new starting units for all users on the computer or just the current user.

7. A new SUMMARY option has been added to the LIST SECTION STRESS command. This option will output the algebraic maximum and minimum values of each stress type for each member in the list of members and an overall summary for all members in the list. An example is shown below:

   LIST SECTION STRESS WITH SUMMARY MEMBERS 1 TO 10

   A SUMMARY ONLY option also has been implemented which will output only the summary information as shown below:

LIST SECTION STRESS SUMMARY ONLY MEMBERS 1 TO 10

The OUTPUT FIELD and OUTPUT DECIMAL settings are used with this new option.

8. A new SUMMARY option has also been added to the LIST SECTION FORCE command. This option prints the algebraic maximum and minimum values for each section force component for each member in the list of members and an overall summary for all members in the list. An example is shown below:

   LIST SECTION FORCE WITH SUMMARY MEMBERS 1 TO 10

   A SUMMARY ONLY option also has been implemented which will output only the summary information as shown below:

   LIST SECTION FORCE SUMMARY ONLY MEMBERS 1 TO 10

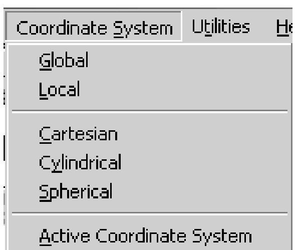
   The OUTPUT FIELD and OUTPUT DECIMAL settings are used with this new option.

2.8 GTMenu

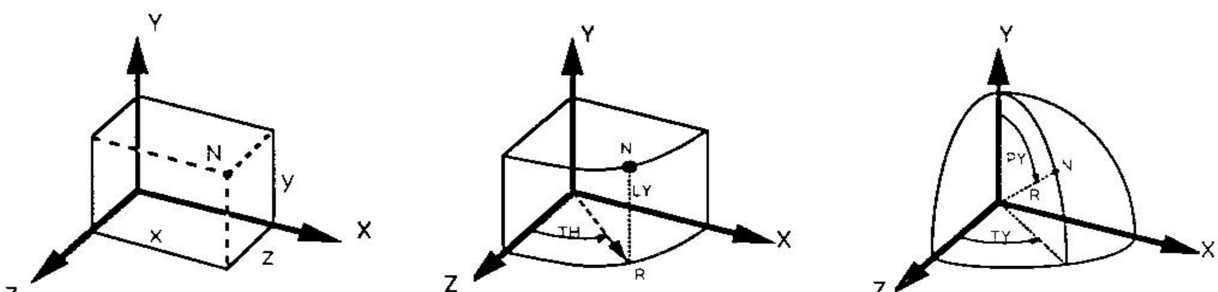
1. The Menu Bar in GTMenu has been rearranged as shown below:

The Coordinate System feature has been removed from the Button Bar and placed on the Menu Bar.

2. The Coordinate System pulldown on the Menu Bar is shown below:



The default coordinate system for all models is the Global Cartesian system. However, you may change to a Cylindrical or Spherical system shown in the figure below:

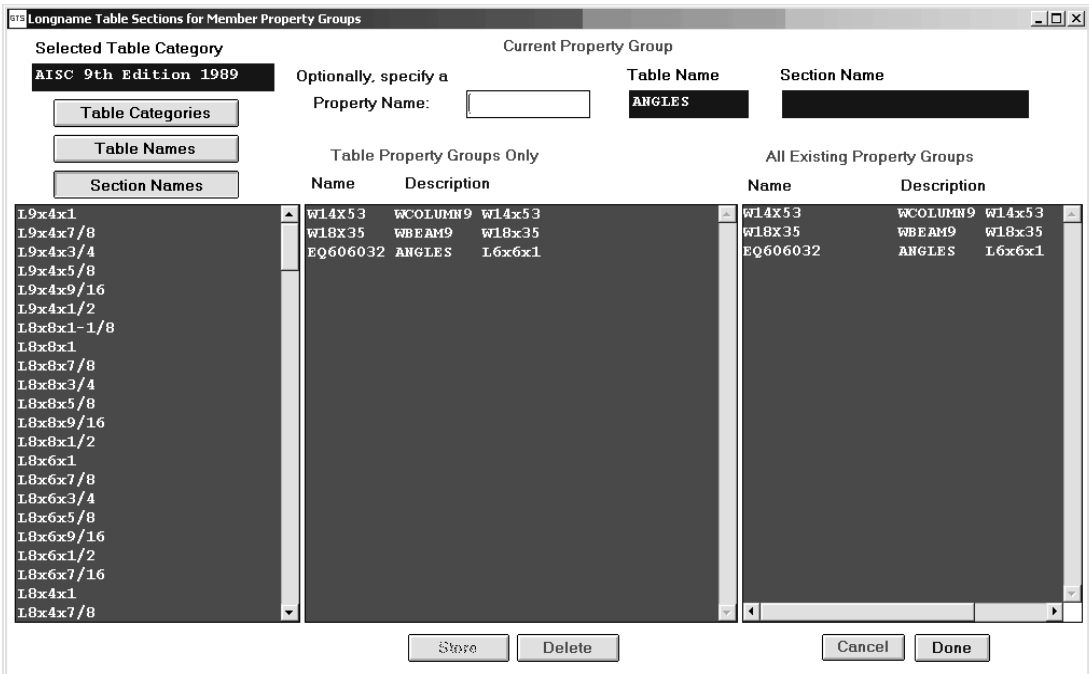


Cartesian Cylindrical Spherical

You may also change to a Local system which may be defined under the Create pulldown as Local Coordinate System. The Active Coordinate System item in the above pulldown will output the currently active coordinate system in the Message Area.

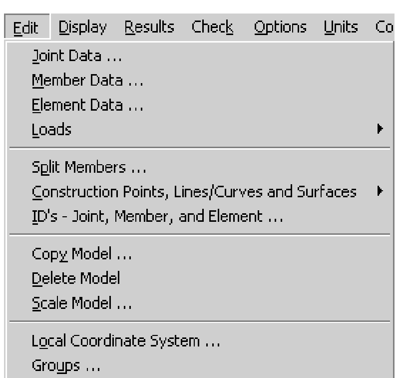
3. The default naming convention for Member Property Table Profiles has been changed to the Long Name naming convention (OUTPUT LONG NAME) which was

established as the default naming convention in Version 26. The Long Name naming convention allows the names for tables profiles to be up to 24 characters. The Short Name naming convention allowed the names to be only 8 characters which severely restricted the use of descriptive names for profiles such as angles. An example of the new format for the AISC 9th Edition Angles table is shown in the figure below:

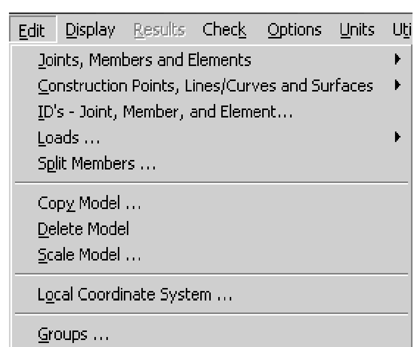


Due to the extended length of the profile name, a horizontal scroll bar has been added under the All Existing Property Groups panel as can be seen in the figure above. The Poperty Group Name shown at the top of the dialog is limited to 8 characters.

4. The Edit pulldown has been reorganized to separate Joints, Members, and Elements. A comparison between the Version 27 and Version 26 Edit Pulldowns is shown below:



Version 27



Version 26

5. Display Loads will now display the following loading types:

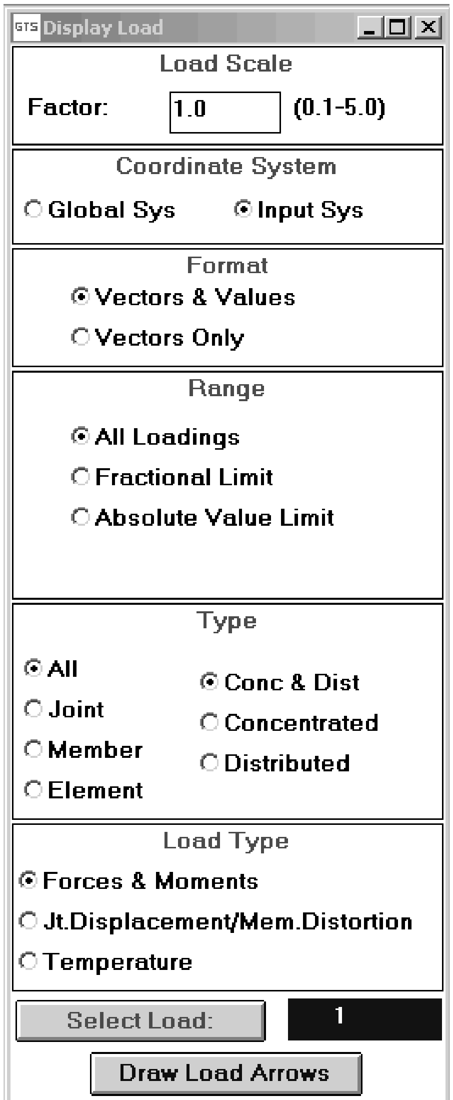
Joint Displacements - joint displacements are displayed in the same manner as joint loads.

Member Distortions - member distortion loading as displayed in the same manner as concentrated and distributed member loads.

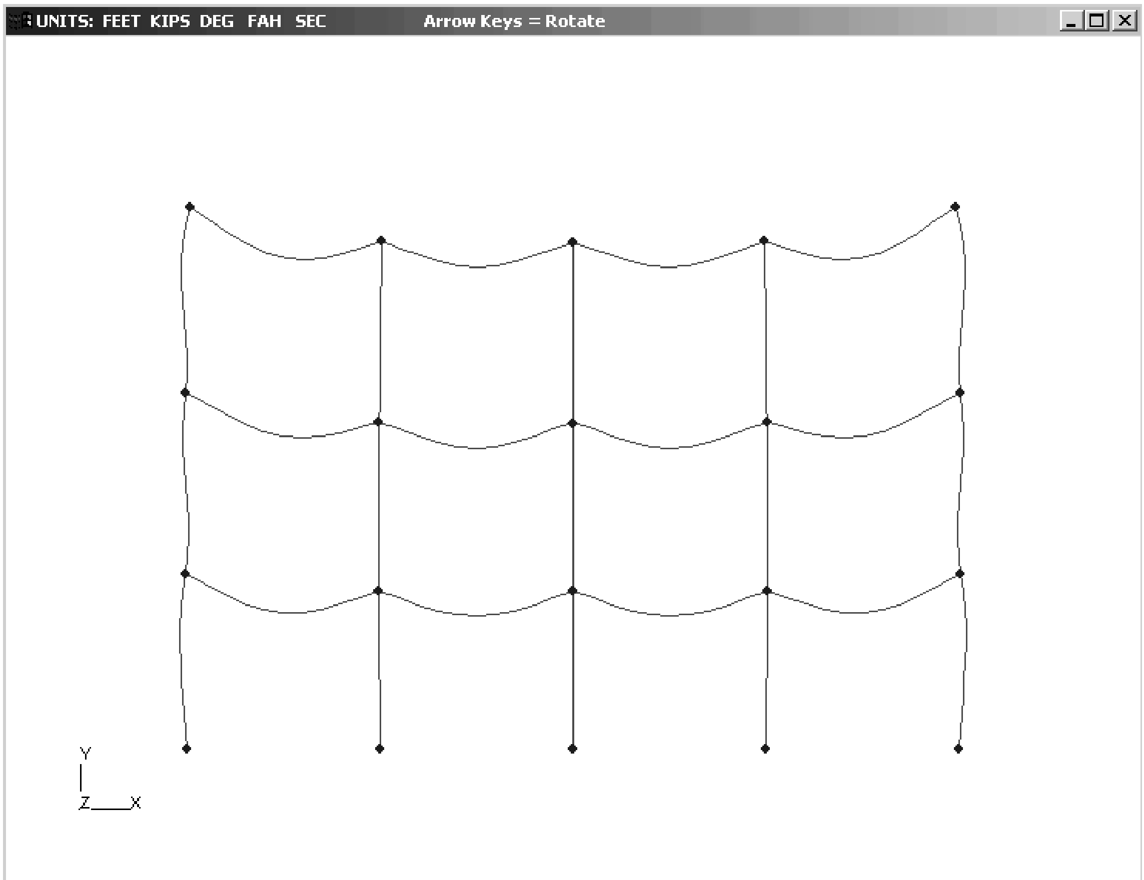
Member Temperatures - member temperature loadings are displayed in the same manner as member loads. Axial member temperature loads are displayed as axial member loads. Bending member temperature loads are drawn as distributed loads perpendicular to the member. A member temperature load which is defined as Bending Z is drawn as a distributed member load in the local z direction of the member.

Joint Temperatures - axial and gradient joint temperature loads for finite elements are drawn at the joints with a temperature change loading drawn as a box around the joint and a temperature gradient drawn as a moment vector at the joint.

The modified Display Load dialog is shown below:

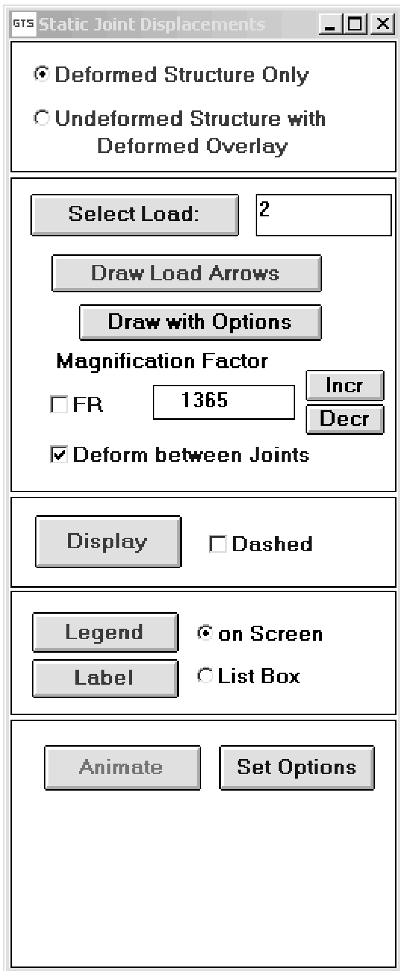


6. The ability to display the deformed shape of members between the joints has been added under the Results pulldown for Deformed Shape. An example of the display using this new option is shown below:



Previously, the deformed shape for members was only shown by drawing straight lines between the deformed positions of the joints. Now, section displacements at 11 locations along each member are computed and used to draw the deformed shape. The user may also point to a position on the member and have the value of the displacement annotated on the screen.

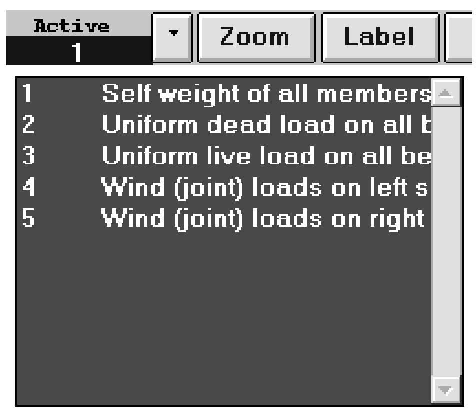
The modified Static Joint Displacement dialog is shown below:



Also added to this dialog is the ability to draw the loads on the structure. The ability to draw the loads on the structure has also been added to all other appropriate Result displays such as Joint Reactions, Force and Moment Diagrams, and Finite Element Contours.

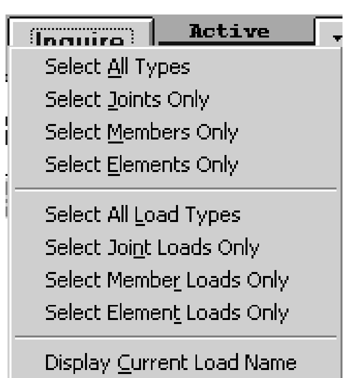
7. The Button Bar has been rearranged as shown below:

The main change on the Button Bar has been the addition of the Active Load item. The first active independent static load name is now displayed under the word Active on the Button Bar. Please note that only independent static loads are shown in this dialog. If you click on the down arrow to the right of the word Active, a list of the currently active independent static loadings will appear as shown below:

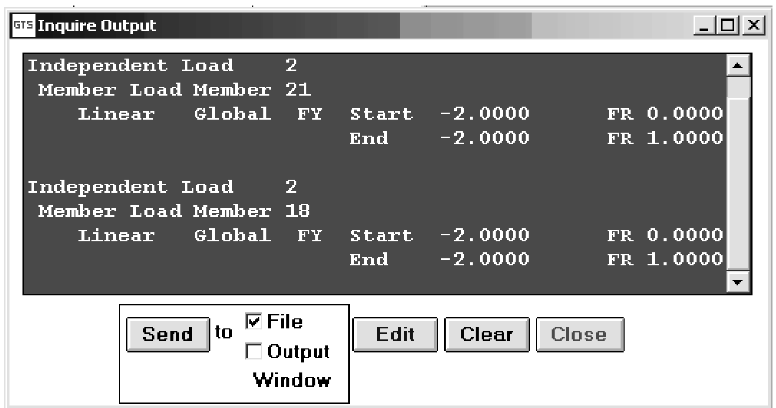


You may set another load as being the active load by clicking on the load.

8. The Inquire button has new features which will allow you to inquire on loadings applied to joints, members and elements. The modified Inquire pulldown is shown below:



By clicking one of the options related to loads in the pulldown on the previous page and then selecting a joint, member, or element in the graphical display area, the loads for the current active loading will be output in the Inquire Output dialog as shown below:

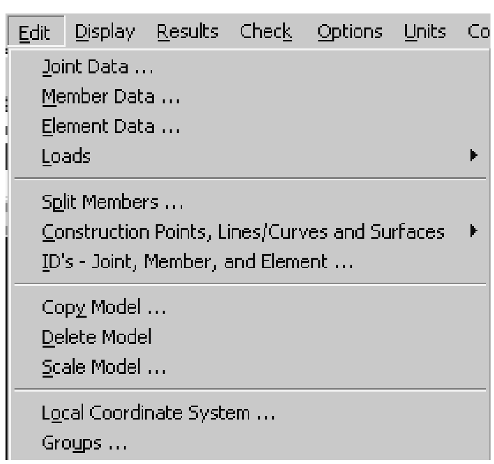


The loading information in the Inquire Output dialog may also be edited as described later in this section.

Selecting the Display Current Load Name option in the Inquire pulldown will pop-up a dialog that will allow you to change the currently active load.

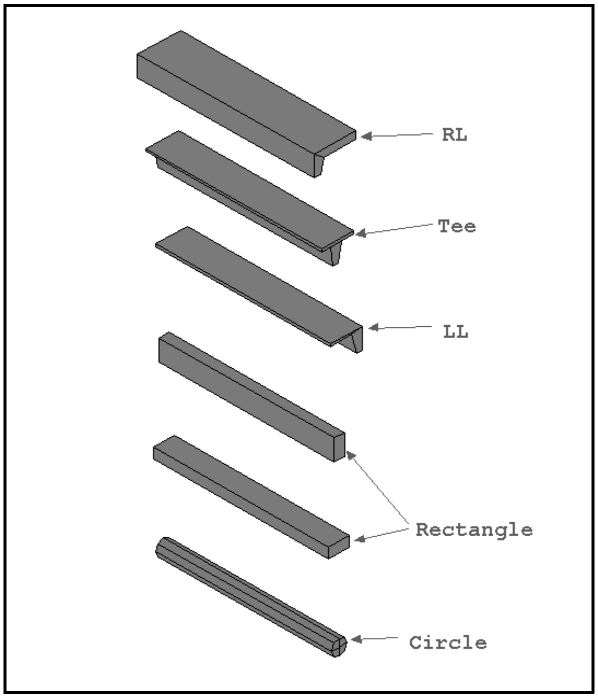
9. GTMenu will include any existing NOTES in the generated GTSTRUDL Input File. The new command which allows you to specify NOTES is described in Section 5.4.8 of this Release Guide.

10. The Edit pulldown has been rearranged to allow direct selections of joints, members, and elements as shown below:



11. Member Properties specified using the Member Dimension command can now be displayed as solid shapes (Redraw Solid) in GTMenu. Rectangle, Circle, Tee, LL, and RL cross sections may now be displayed. In addition, structural walls including barbell walls may now be displayed as solid shapes.

    An example of Redraw Solid for several of these cross sections is shown below:

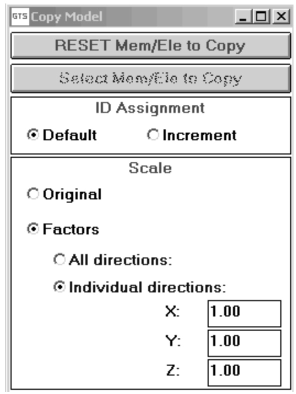


12. A previously selected font size is retained throughout a GTSTRUDL session. Previously, the font size would need to be respecified each time GTMenu was entered in a given session.

13. The default font size has been made smaller for higher display resolutions.

14. The cursor is displayed as an hourglass during operations which can take considerable time such as the Copy Model, Contour, and Redraw Solid features.

15. The Copy Model Scaling options have been redesigned as shown in the partial Copy Model dialog below:

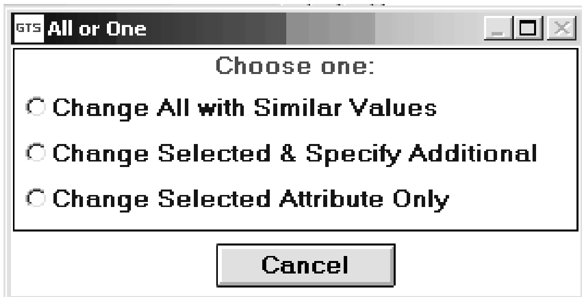


16. Right-click editing has been expanded with the following features:

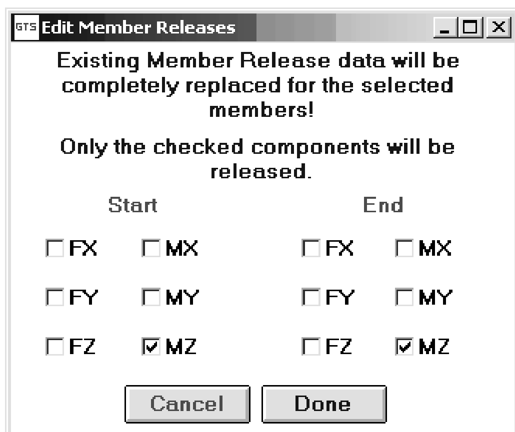
a. Dialogs are initialized with current values where appropriate
b. Specify additional entities to be changed
c. Select all entities having the same values to be changed
d. Update the listing in the Inquire Output dialog after making the changes for a given joint, member or loading

An example of using some of these new features is shown below based on a right mouse click of one of the members in a frame structure:

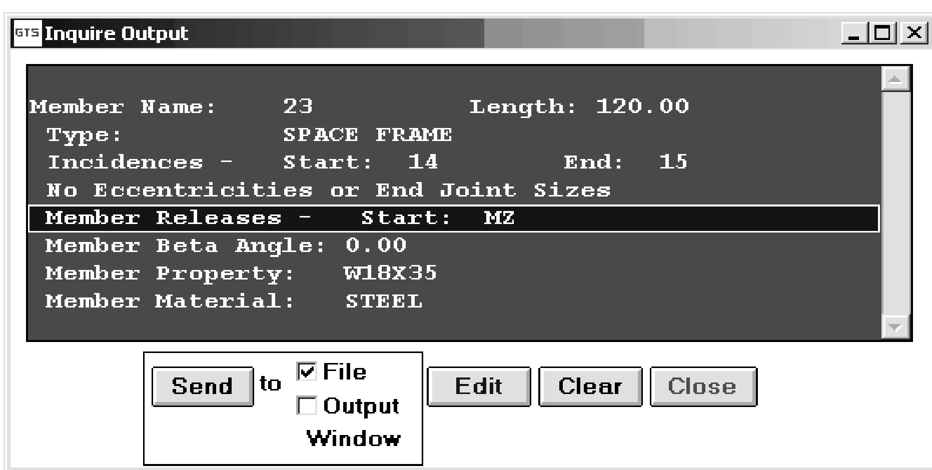
Inquire Output for Member 23 - double click Member Releases above to Edit the Releases. The following dialog appears which allows you to change all members with similar values, change selected members and also identify additional members, or change only the selected attribute which in this case is the member release on member 23:



After selecting the option to Change Selected Attribute Only, the Edit Member Release Dialog Appears with the current releases indicated in the dialog as shown below:

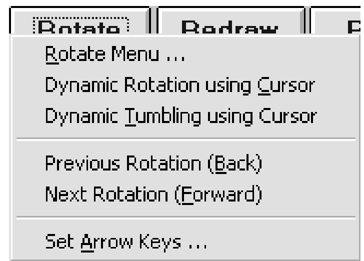


After removing the MZ release at the End of the member, Inquire Output is updated as shown below indicating that Member 23 has a MZ release only at the start:



17. Zoom, Rotate, and View changes are now saved in whatever order they are specified. The displays are accessed using menu picks Zoom Previous, Zoom Forward, Previous Rotation, and Next Rotation under the Zoom and Rotate buttons as shown below:

Zoom Button



Rotate Button

18. The text input file now contains Long Names for the names of steel profiles.

19. The GTSTRUDL Version number (Version 27) is now placed on the GTMenu Title Bar at the top of the GTMenu window.

20. A Reset button has been added to the Default Colors dialog which will reset the colors to the colors at the start of work on the current model. Thus, if you have changed colors while in GTMenu, you may now reset them back to the colors at the start of work on the current model.

21. New HotKeys have been added to GTMenu. The complete list of HotKeys is shown below:

There are 7 groups of Graphics HotKeys:

A. The arrow keys: "up", "down", "left" and "right".

B. The arrow mode keys:
"z" - Zoom: zoom in ("up") or zoom out ("down");
"p" - Pan: move the screen window;
"r" - Rotate: increment the screen rotation.

C. Other Graphics Display function keys:
"<"/"," and ">"/"."- rotate about screen Z axis;
"u" - go to a fUll structure (or View) display;
"b" - Back to previous Graphics Display;
"f" - Forward to next Graphics Display.

D. View selection keys and key sequences:
press a single digit View number;

press "0" followed by a 2 digit View number;
press "00" followed by a 3 digit View number.

E. Inquire function keys:
"n" - iNquire about clicked-on entity: +<Ctrl> for load;
"j" - select Joints only: +<Ctrl> for load;
"m" - select Members only: +<Ctrl> for load;
"e" - select Elements only: +<Ctrl> for load.

F. Orientation key and key sequences:
"i" - Isometric projection;
"xy" projection;
"xz" projection;
"yz" projection.

G. The HELP key ("h") - The above information.

22. GTMenu is no longer limited to the display of 32,768 joints, members, or elements. The limit has been increased to 1,000,000. However, other computer resources may limit the size of the model that can be created and displayed in GTMenu.

23. Information in the Inquire Output box may now be sent to the GTSTRUDL Output Window as well as a file. An example of the output from the Inquire sent to the GTSTRUDL Output Window is shown below:

```
-------------------------------------------------------------------
  Output from GTMenu follows:

Member Name:    26          Length: 120.00
 Type:          PLANE FRAME XY
 Incidences -   Start:  18       End:  19
 No Eccentricities or End Joint Sizes
 No Member Releases
 Member Beta Angle: 0.00
 Member Property:   W18X35
 Member Material:   STEEL

Member Name:    22          Length: 120.00
 Type:          PLANE FRAME XY
 Incidences -   Start:  13       End:  14
 No Eccentricities or End Joint Sizes
 No Member Releases
 Member Beta Angle: 0.00
```

Member Property: W18X35
Member Material: STEEL
End of Output from GTMenu
----

24. When generating an input file in GTMenu, the number of items listed before a new header has been increased from 20 to 100. This will result in fewer header items, longer lists, and somewhat reduce the number of lines in the created input file.

25. When entering GTMenu with a model which contains superelements, the pop-up indicating that Default Element Types will be assumed will no longer occur. Superelement definition information will now be retained upon exiting GTMenu with changes to the model.

26. Superelement definition lines are written by the input file generator as appropriate, including

    (a) boundary node specs
    (b) element specs
    (c) and similarity specs.

27. After editing the Beta angle for members, the new profile with axes is now drawn in the graphical display area.

28. Tool Tips now automatically appear when the cursor is placed over items in the Button Bar.

29. Curve definitions now use 360 straight line segments to approximate the curve.

30. Prompts for each of the 3 points used to define a circular arc curve have been added to the Message Area. A right click is no longer required after specifying the third point in order to generate the circular arc.

2.9 Nonlinear Analysis

1. The DEFINE CABLE NETWORK command has been expanded to include absolute tolerance specifications for the SAG COORDINATE and INITIAL TENSION options. This permits you to specify an absolute prestress analysis convergence tolerance tailored specifically for an individual cable network. You may continue to

specify the original CONVERGENCE TOLERANCE GEOMETRY value using the CABLE PRESTRESS ANALYSIS DATA command. This value applies to the prestress analysis convergence checking if the new SAG COORDINATE or INITIAL TENSION tolerance value is not specified.

The new PRINT CABLE PRESTRESS ANALYSIS DATA command has been added. This command permits you to print out the data that was previously specified by the DEFINE CABLE NETWORK and CABLE ANALYSIS DATA commands.

2. Plastic hinge nonlinearity has been extended to provide for the description and specification of custom piece-wise linear stress-strain curves for the steel and concrete material of plastic hinges. The new STORE STRESS STRAIN CURVE, DELETE STRESS STRAIN CURVE, and PRINT STRESS STRAIN CURVE commands provide for the description and management of the piece-wise linear stress-strain curve data. The new CURVE option in the NONLINEAR EFFECTS, PLASTIC HINGE command provides for the assignment of custom stress-strain curves to define the steel and concrete stress-strain behavior of plastic hinges.

3. The new pushover analysis output command LIST PUSHOVER LIMIT LOAD has been added. This command is used to list the intermediate pushover analysis incremental storage load at which a specified plastic hinge limit strain is first equaled or exceeded for the plastic hinges in each of a specified set of members.

4. The CREATE RESPONSE SPECTRUM command has been expanded to include a PERIOD option that provides for the creation of ACCELERATION/VELOCITY/ DISPLACEMENT vs PERIOD response spectrum files in addition to the original FREQUENCY based files.

5. The following enhancements have been made in the areas of general nonlinear analysis output commands and pushover analysis output commands:

    1. The LIST PLASTIC HINGE STRESSES command has been added. This command lists the plastic hinge fiber normal stresses for a set of specified members and active loading conditions. Special control options are used to select maximum or minimum stress output and output for a single fiber and output for steel or concrete fibers only.

    2. The YIELD STRAIN parameter has been added to the LIST PLASTIC HINGE DUCTILITY FACTOR and LIST PUSHOVER DUCTILITY

FACTOR commands. This new option provides for the specification of a specific yield strain value for the computation of these ductility factor results. See Section 5.3.11 for more information on these features.

6. A new nonlinear spring element, the NLS4PH element, has been implemented. The NLS4PH element is a special version of the nonlinear spring element (Section 2.5.3, Volume 3 of the GTSTRUDL Reference Manual), where the stiffness properties are described by bilinear, symmetric, hysteretic, force-displacement and moment-rotation curves. The NLS4PH element may be used in both nonlinear static and dynamic analysis. The ELEMENT PROPERTIES command for the NLS4PH element may be found in Section 5.3.9.

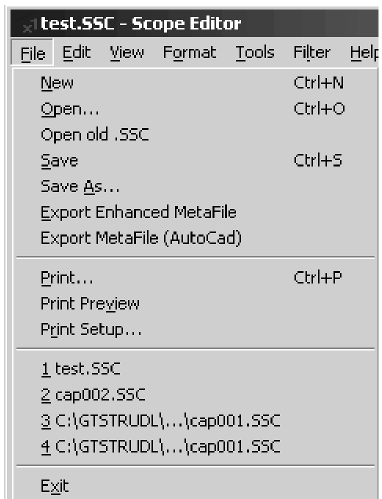
7. The plastic segment, a new version of the plastic hinge nonlinear effect, has been implemented. The NONLINEAR EFFECTS command (see Section 5.3.8) has been extended to include the PLASTIC HINGE/SEGMENT option to select the desired model for member plastic hinge nonlinear material behavior. The PLASTIC HINGE selection means that the member plasticity behavior is modeled by a discrete, zero-length spring mechanism lumped at the start and/or end of the member. The PLASTIC SEGMENT selection means that the member plasticity behavior is distributed along the length of a segment of length LH (specified plastic hinge length) at the specified start and/or end of the member. The PLASTIC SEGMENT model predicts member stiffness more accurately than the PLASTIC HINGE model, but in this initial implementation, members specified as having the PLASTIC SEGMENT nonlinear effect do not support member loads of any type.

2.10 Offshore

1. The depth below the water surface, Z, is now automatically computed for the hydrostatic collapse check for the APIWSD20 and APILRFD1 codes. Currently, the user must specify the parameter Z for each member below the water surface.

2.11 Scope Editor

1. The Scope Editor has been enhanced to Export a Metafile as shown in the figure below:



This Metafile may be brought into AutoCAD. Also added under File is the ability to Open Scope Editor documents (.SSC files) from earlier releases. This was necessary because the display style for joints and supports in GTMenu and the format for saving ellipse and circle data in the Scope Editor changed in this release.

2.12 Steel Design

1. The Third Edition of the LRFD code has been implemented. This code is referred to as LRFD3 and is available for the following shapes:

   I shapes
   Single Angles (axial load only)
   Double Angles (axial load only)
   Round HSS (Pipes)
   Square and Rectangular HSS (Structural Tubes)

   Parameters for this code are described in Section 5.2.1.

2. The British steel code based on BS 5950-1:2000 has been implemented. This code is available for the following shapes:

I shapes
Single Angles (axial load only)
Circular Hollow Sections (Pipes)

Parameters for this code are described in Section 5.2.3.

3. Column lines are no longer needed for the K factor computation. If the K factor computation has been requested and the COLUMN LINE command has not been specified, an automatic search is performed to detect columns attached to the start and end of the member that is being code checked or designed.

4. When the K factor computation is requested, the computed K factors will now be automatically printed after code CHECK and SELECT results as shown in the output below.

```
MEMBER    TABLE       LOADING     SECTION   PROVISION ACTUAL/           SECTION FORCES           UNITS
   CODE      PROFILE     NAME        LOCATION  NAME      ALLOWABLE     FX/MT      FY/MY      FZ/MZ   TRIALS
  /---------/-----------/-----------/---------/---------/----------/----------/----------/----------/---------/

   1         WCOLUMN9    2              10.000 H1-1 COM     0.813      -29.841      0.896      0.000 FEET KIP
   ASD9      W6x15                             B7 COMP      0.411        0.000      0.000     -6.210     4
   KY =   1.00     KZ =   1.34     Column at End = 2
```

A new parameter, Print-K whose default value is YES, can be set to NO to turn off this automatic K factor output.

5. New LRFD3 steel grades have been added to the ASD9 code.

2.13 Steel Tables

1. New Brazilian steel tables have been added from ABNT, NBR 5884:
   CS - welded plate girder shapes for columns (I shapes)
   CVS - welded plate girder shapes for beams and columns (I shapes)
   VS - welded plate girder shapes for beams (I shapes)
   More information regarding the profiles in these tables may be found in Section 5.2.7.

2. New HSS tables have been added from the LRFD Third Edition code:
   RoundHS - round HSS (pipes)
   RectHSS - rectangular and square HSS (structural tubes)
   More information regarding the profiles in these tables may be found in Section 5.2.2.

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 Dynamic Analysis

1. An abort will no longer occur when a lumped mass/weight with a damping factor is added to a fixed degree of freedom. (GPRF 2002.02)

2. The composite modal damping matrix is now correctly re-assembled when the only source of damping is modal damping proportional to joint inertias. In previous versions the second and later dynamic assembly operations due to DYNAMIC ANALYSIS EIGENVALUE, DYNAMIC ANALYSIS MODE SUPERPOSITION, and ASSEMBLE FOR DYNAMICS commands did not properly re-assemble the composite modal damping matrix when the only source of damping is modal damping proportional to joint inertias. (GPRF 2002.04)

3. The CREATE RESPONSE SPECTRUM command no longer aborts if the number of time points for the integration computation exceeds 1 million. (GPRF 2002.05)

4. The LIST TRANSIENT MAXIMUM command for transient analysis joint results no longer produces incorrect results for planar master joints. (GPRF 2002.06)

5. The error in global mass matrix assembly of mass moment of inertia degrees of freedom from joint masses applied to slave nodes of rigid pin elements has been corrected. (GPRF 2002.08)

6. Accelerations, velocities, and displacements are now computed correctly for slave degrees of freedom when a stiffness analysis execution is inserted between an eigenvalue analysis and a transient and/or steady state analysis. As a result of this correction, member and element results also are computed correctly for members and elements that are connected to slave degrees of freedom. (GPRF 2002.10)

7. The FORM MISSING MASS LOAD command no longer destroys the response spectrum spectral acceleration results computed by previous response spectrum analyses. (GPRF 2003.1)

8. Dynamic analysis no longer aborts when member added inertia is applied to variable members and the INERTIA OF JOINTS LUMPED/CONSISTENT command is not given. (GPRF 2003.3)

3.2 Finite Elements

1. The DEVELOP AND SAVE command for the calculation of superelement stiffness and load matrices will no longer abort during the process of writing superelement data to the user data set if the number of boundary nodes exceeds 236. (GPRF 2003.2)

3.3 General

1. Truss members with temperature loads and member releases no longer abort. Consistency checking will note releases on the truss member and turn SCAN on. (GPRF 2001.3)

2. When the limit of 255 entries (tables or "files") in a user library (userdat.ds) is exceeded, GTSTRUDL will no longer abort, but will print an error message informing the user of the condition. (No GPRF issued)

3. STIFFNESS ANALYSIS will no longer abort if MEMBER RELEASES and MEMBER TEMPERATURE loads have been specified for a plane or space truss member. (GPRF 2002.03)

4. The LIST SECTION DISPLACEMENT command will no longer produce incorrect results in the Local Z direction for members with a start Moment Y release where the joint displacement for start and end joints are different. (GPRF 2002.12)

5. The output of the maximum bandwidth will no longer overflow the output field and be printed as **** if the maximum bandwidth is greater than 9999. (No GPRF issued)

3.4 GTMenu

1. Transient animation will no longer abort due to a memory allocation failure. This problem occurred on models which contained either a large structure or a large number of transient time points. (No GPRF issued)

2. An abort no longer occurs if a View file that is to be written to is a Read-Only file.(No GPRF issued)

3. An abort will no longer occur when the user contours a stress component which is not applicable to the elements in the model. For instance, if a user had plane stress elements in the model and selected SZZ to be contoured, an abort would occur. Now, an Alert Box is popped up indicating that the stress component is not applicable.(No GPRF issued)

4. An abort will no longer occur when an Edit operation is canceled from an Inquire Output box. (No GPRF issued)

5. Element force results are no longer destroyed after entering GTMenu and making changes to a model. (No GPRF issued)

6. Splitting a group of members having a large number of member loads no longer causes an abort to occur. (No GPRF issued)

7. The Global Plane mode now operates correctly when defining a Group. (No GPRF issued)

8. Contour dialog options (planar vs solid elements) are now reset when changing views to reflect the elements in the view. (No GPRF issued)

9. Variable finite element load magnitudes no longer run together without a space between them in the Input File generated by GTMenu. (No GPRF issued)

10. An abort will no longer occur when using a HotKey to activate a deleted view. For example in previous versions if you typed in 9 in order to activate view number 9, an abort would occur if View number 9 had been deleted. (No GPRF issued)

11. Changing the member type from space frame to space truss in GTMenu will no longer result in erroneous statics check failures. Previously, if an analysis had already been performed and the user changed a member from a space frame member to a space truss member in GTMenu and then performed another analysis, static check failures would result and the results would be incorrect. This problem occurred only if an analysis had already been performed and the user changed the member type from a frame member to a truss member and then performed another analysis. (No GPRF issued)

12. The Global Plane mode now works correctly in the Results Code Check dialog. (No GPRF issued)

13. When generating an input file, the Job ID is no longer erased when a carriage return is entered.
(No GPRF issued)

14. When generating an input file, a continuation mark is no longer inserted for the job title or load title. (No GPRF issued)

15. The Check Model Load Summation will produce correct results for triangular plate elements with projected element surface forces. (No GPRF issued)

16. The Display of Supports will now display elastic spring symbols for all joints which have elastic supports. Previously, only the first joint with a particular set of elastic springs would have the spring symbol drawn on the joint. (No GPRF issued)

17. A view is no longer truncated when the view is defined as a global plane and the coordinates of the joints in the view are negative. (No GPRF issued)

18. Reading a view database file that has a pathname longer than 80 characters will no longer cause an abort. The pathname has been lengthened to 256 characters. (No GPRF issued)

19. Models which contain superelements will now analyze correctly after exiting GTMenu with changes made to the model. All superelement data specified before entering GTMenu will be retained after changes are made to the model. (No GPRF issued)

20. A warning message will now appear if the user attempts to create more than 1,000 views. Previously, an abort would occur if the view limit was exceeded. (no GPRF issued)

3.5 GTSTRUDL Output Window

1. The syntax for the command created from the Specified Damping dialog when Rayleigh damping (DAMPING PROPORTIONAL TO) has been specified and has been corrected.

2. Nonlinear Analysis has been added to the Dialog History list. Previously this dialog was not recorded by Dialog History.

3. Load group data now appear when the “Display Components” button is selected.

4. The DAMPING PERCENTS command is now correct when an integer value is input.

5. Block editing (“Change selected data”) for Member properties in the Members datasheet has been corrected.

3.6 Nonlinear Analysis

1. Nonlinear analysis no longer aborts when cable elements and members having variable properties are used in the same model. (GPRF 2002.07)

2. The NONLINEAR ANALYSIS CONTINUE command will no longer abort if new joints, members, and finite elements are given prior to the command. (GPRF 2002.09)

3. The NONLINEAR ANALYSIS command will no longer abort for structures that have IPCABLE elements and when the active load list contains at least two loading conditions when the NONLINEAR ANALYSIS command is given. (GPRF 2002.11)

3.7 Nonlinear Dynamic Analysis

1. Nonlinear dynamic analysis no longer produces erroneous member force results for the first of the group of TENSION/COMPRESSION ONLY members if these members are not the last in the entire sequence of members and finite elements that were defined and generated. (No GPRF issued)

2. Nonlinear dynamic analysis no longer aborts when the STORE ABSOLUTE ACCELERATION command was given. (No GPRF issued)

3.8 Reinforced Concrete

1. Attempting to delete non-existent MEMBER DIMENSIONS no longer causes an abort. (No GPRF issued)

2. Proportioning of flat plate and flat slabs is not allowed for code BS8110. Attempting to do this will now result in a warning message, instead of an abort. (No GPRF issued)

3.9 Utilities

1. The DXF Converter utility (File → Import → DXF) will no longer abort when there is more than one layer in the DXF file. (No GPRF issued)

2. The calculation of line intersections when a line segment cuts more than one other line segment has also been corrected in the DXF Converter. (No GPRF issued)

This page intentionally left blank.

CHAPTER 4

KNOWN DEFICIENCIES

This chapter describes known problems or deficiencies in Version 27. The following sections describe the known problems or deficiencies by functional area.

4.1 Dynamics

1. Global joint masses are not computed correctly for masses specified from the MEMBER ADDED INERTIA command for plane frame members with variable member properties. This error occurs under the following conditions:
   a. Member properties are defined as variable with 3 or more segments.
   b. The segment cross-section properties are defined by AX, IY, or IZ only.
   (GPRF 99.17)

4.2 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. The results from the LIST PRINCIPAL STRESSES command are incorrect for loading combinations for the IPQL series (IPQL, IPQL2, etc.) elements. (GPRF 91.44 AND 91.45)

3. 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)

4. 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)

5. 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)

6. 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)

4.3 General Input/Output

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

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

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

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

3. The path plus file name on a SAVE or RESTORE is limited to 256 characters. If the limitation is exceeded, the path plus file name will be truncated to 256 characters. (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.

6. If a loading contains JOINT TEMPERATUREs and that loading is later changed with the JOINT TEMPERATURES deleted, the joints which contained the deleted JOINT TEMPERATUREs will be incorrectly changed to planar element joints. Subsequent analyses may produce structural instabilities. This can be checked by the PRINT JOINT COORDINATE command and checking to see if the joints are listed as PLAN-EL (planar element). The PRINT JOINT COORDINATE command will show the joints have been changed to planar element (PLAN-EL) (GPRF 2001.07)

4.4 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.
(No GPRF issued)

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.

(No GPRF issued)

3. The input file generated when using the TRANS3D element is incorrect.

Workaround: Edit the input file by moving the TYPE and WITH NODES part of the ELEMENT PROPERTIES command to the line with the list of element ID's as shown below:

Original Input File:

```
ELEMENT PROPERTIES TYPE 'TRANS3D' WITH NODES 13 15
   '1     '  '2      '
```

Modified Input File:

```
   ELEMENT PROPERTIES
   '1     '  '2      '   TYPE 'TRANS3D'   WITH NODES 13 15
```

(No GPRF issued)

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)

6. A member's Profile Local axes, or Redraw Solid display are displayed incorrectly if an eccentricity is applied to the members which results in the actual member going from being parallel to global Y to not being parallel to global Y or vice versa.
   (No GPRF issued)

4.5 Rigid Bodies

1. Deletion of a rigid body erroneously inactivates any of the rigid body incident joints to which only one additional member or finite element is attached. A message similar to the one below will be output when this error occurs:

   **** INFO_STMJDL - - The following joints were made inactive when the above members were deleted ... .

   The user can workaround this problem by reactivating the joints using the ACTIVE command (GPRF 99.14)

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

4.6 Scope Environment

1. The LABEL BETA command rotates the cross-section shapes according to the position of the member after member eccentricities have been applied. The cross-section orientation should be determined in the joint-to-joint position of the member. (GPRF 92.11)

2. 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 27 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

5.3 Analysis Prerelease Features
- 5.3.1 Calculate Error Estimate Command
- 5.3.2 Finite Element Dictionary Revision - Temperature Gradient Loading
- 5.3.3 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis
- 5.3.4 Dynamic Analysis External File Solver to Improve Efficiency of Dynamic Analysis Results Computation
- 5.3.5 Output of Response Spectrum Results

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 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
Single Angles	Subjected to axial force only
Double Angles	Subjected to axial force only
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

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). 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.

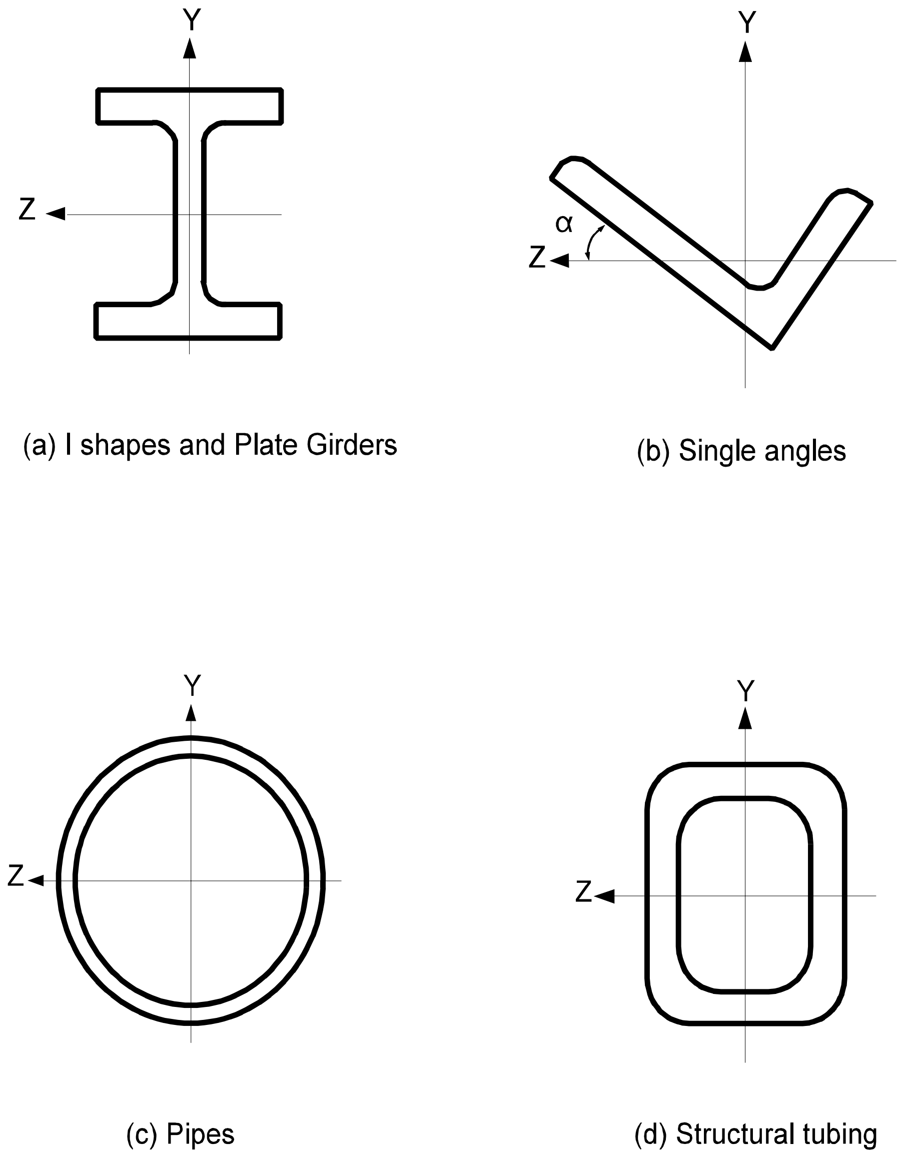
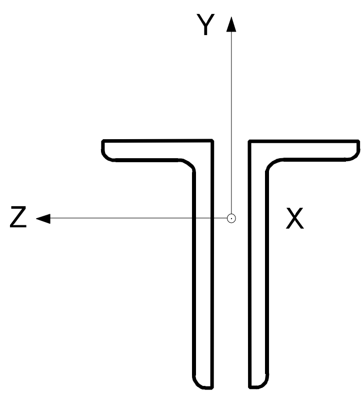
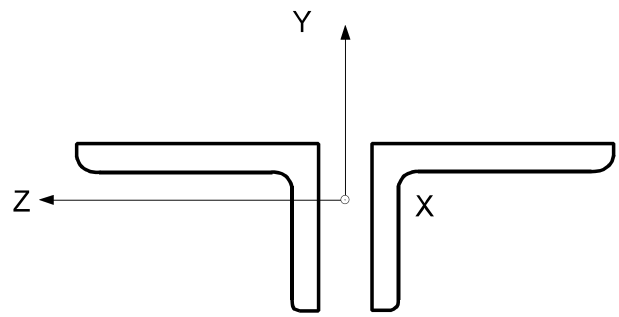


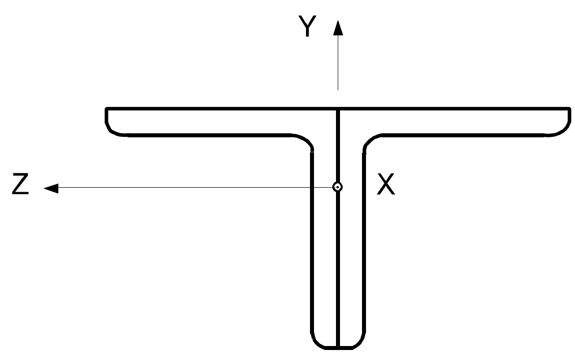
Figure LRFD3.1-1 Local Axes for Design with LRFD3



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



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



(g) 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, single and double angles) 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, single and double angles). The designer is reminded to check the torsional stresses for open cross-sections (I shape, single and double angles) 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 base on the user specified or designed number of the intermediate connectors.

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

Section	Title
Chapter D	**Tension members**
Section B7	Limiting slenderness ratios
Section D1	Design tensile strength
Chapter E	**Columns and other compression members**
Section B7	Limiting slenderness ratios

Section	Title
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 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
Chapter F	**Beam and other flexural members**
Section F1.1	Yielding
Section F1.2a	Bending coefficient dependent on moment gradient, Equation F1-3
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
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 Steel Hollow Structural Sections

Table 2.2-1	Limiting Wall Slenderness for Compression Elements

Section	Title
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 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 the 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 alphabetic order in this section.
4.	Sections LRFD3.4	Describes the subsections in 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 bending and axial 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.
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 bending, axial, 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 bending, axial, and torsional 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). LRFD3 code is applicable to the following cross-sections: 1. I shapes 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 and Double Angle profiles 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. See Table LRFD3.1-3 for choices.
CODETOL	0.0	Percent variance from 1.0 for compliance with the provisions of a code. The ratio of Actual/Allowable must be less than or equal to [1.0 + CODETOL/100].
PF	1.0	Area reduction factor for holesout in members subject to axial tension.
a	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 'STEEGRD' 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). 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 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 names of the columns attached to the start and end of the code checked member are 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 columns attached 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).
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).
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).
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).
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
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 needed. If the designed required number of connectors is larger than user specified value, the computed number of connectors is 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 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.

Table LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	Default Value	Meaning
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). 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). The default is computed as the length of the member.
Round HSS (Pipes) Torsion Check Parameters		
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). The default is computed as the length of the member.
Force Limitation		
FXMIN	0.5 (lb)	Minimum axial force to be considered by the code; anything less in magnitude is taken as zero.
FYMIN	0.5 (lb)	Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero.
FZMIN	0.5 (lb)	Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero.

Table LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Force Limitation (continued)		
MXMIN	20.0 (in-lb)	Minimum torsional moment to be considered by the code; anything less in magnitude is taken as zero.
MYMIN	20.0 (in-lb)	Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero.
MZMIN	20.0 (in-lb)	Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero.
Output Processing and System Parameters		
SUMMARY	NO	Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; see Sections 2.9 and 7.2 of Volume 2A 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 LRFD3 (continued)

LRFD3 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Output Processing and System Parameters (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 LRFD3.1-2

GTSTRUDL AISC Codes*

Code Name	Parameter Table	Application
LRFD3	LRFD3 Volume 2C	Checks compliance of I shapes (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, Single angle, Channel, Tee, Double angle, Solid round bar, Pipe, Solid Square and Rectangular bar, 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, Single angle, Channel, Tee, Solid round bar, Pipe, Solid Square and Rectangular bar, 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, Single angle, Channel, Tee, Solid round bar, Pipe, Solid Square and Rectangular bar, 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 Codes

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.

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, and 2L

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	3658	36 58	36 58	36 58	3658
A529-G50	5065	5065	NA	NA	NA
A529-G55	5570	5570	NA	NA	NA
A572-G42	4260	42 60	42 60	42 60	42 60
A572-G50	5065	50 65	50 65	50 65	50 65
A572-G55	5570	5570	5570	5570	5570
A572-G60	6075	6075	6075	NA	NA
A572-G65	6580	6580	6580	NA	NA
A913-G50	5060	5060	5060	5060	5060
A913-G60	6075	6075	6075	6075	6075
A913-G65	6580	6580	6580	6580	6580
A913-G70	7090	7090	7090	7090	7090

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, and 2L

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]	5065	5065	5065	5065	5065
A242	5070	50 70	46[b] 67[b]	42[a] 63[a]	42[a] 63[a]
A588	5070	50 70	50 70	5070	5070

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	3560	NA
A500-GB	4258	NA	4658
A500-GC	4662	NA	5062
A501	3658	NA	3658
A618-GI A618-GII Thickness ≤ 3/4	5070	NA	5070
A618-GI A618-GII Thickness > 3/4	4667	NA	4667
A618GIII	5065	NA	5065
A242-G46	NA	NA	4667
A242-G50	NA	NA	5070
A588	NA	NA	5070
A847	5070	NA	5070

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.

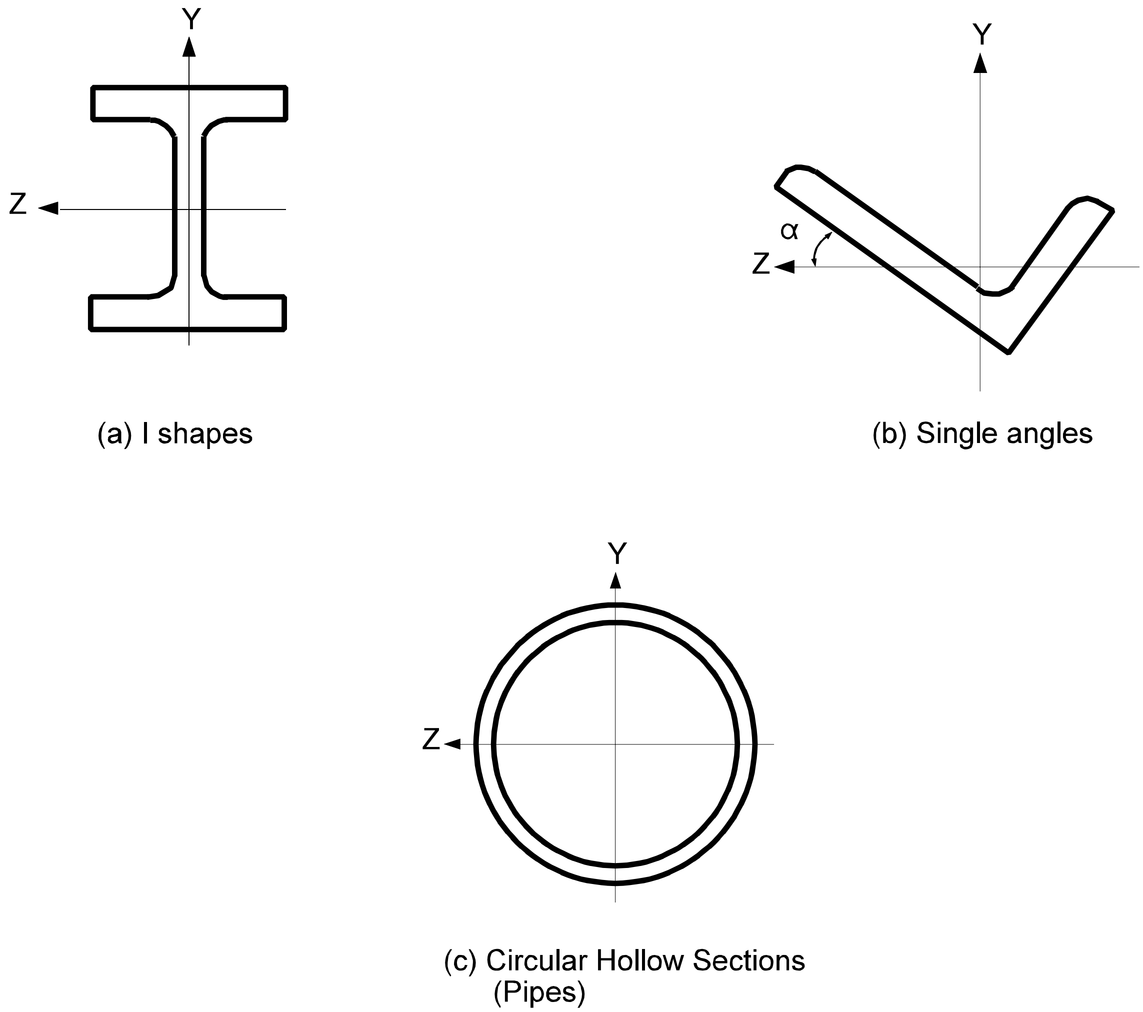


Figure 00BS5950.1-1 Local Axes for Design with 00BS5950

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

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

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

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

Parameter Name	**Default Value**	**Meaning**
Force Limitation (continued)		
FZMIN	2.224 (N)	Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero.
MYMIN	2260.0 (mm-N)	Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero.
MZMIN	2260.0 (mm-N)	Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero.
Output Processing		
MXTRIALS	500.0	Maximum number of profiles to be tried when designing a member. Default is larger than the number of profiles in most tables.
SUMMARY	NO	Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; see Sections 2.9 and 7.2 of Volume 2A 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		
	$t \le 16$	$16 < t \le 40$	$40 < t \le 63$	$63 < t \le 80$	$80 < t \le 100$	$100 < t \le 150$	$150 < t \le 200$	$200 < t \le 250$	$t \le 100$	$100 < t \le 150$	$150 < t \le 250$
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		
	$t \le 16$	$16 < t \le 40$	$40 < t \le 63$	$63 < t \le 80$	$80 < t \le 100$	$100 < t \le 150$	$150 < t \le 200$	$200 < t \le 250$	$t \le 100$	$100 < t \le 150$	$150 < t \le 250$
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

<table>
<tr><td colspan="4">a) non-sway mode*</td></tr>
<tr><td colspan="3">Restraint (in the plane under consideration) by other parts of structure</td><td>EFLEY and EFLEZ</td></tr>
<tr><td rowspan="4">Effectively held in position at both ends</td><td colspan="2">Effectively restrained in direction at both ends</td><td>0.7</td></tr>
<tr><td colspan="2">Partially restrained in direction at both ends</td><td>0.85</td></tr>
<tr><td colspan="2">Restrained in direction at one end</td><td>0.85</td></tr>
<tr><td colspan="2">Not restrained in direction at either end</td><td>1.0</td></tr>
<tr><td colspan="4">b) sway mode*</td></tr>
<tr><td>One end</td><td colspan="2">Other end</td><td>EFLEY and EFLEZ</td></tr>
<tr><td rowspan="3">Effectively held in position and restrained in direction</td><td rowspan="3">Not held in position</td><td>Effectively restrained in direction</td><td>1.2</td></tr>
<tr><td>Partially restrained in direction</td><td>1.5</td></tr>
<tr><td>Not restrained in direction</td><td>2.0</td></tr>
<tr><td colspan="4">* Excluding angle, channel or T-section struts designed in accordance with Section 4.7.10 of the BS 5950-1:2000 (95)</td></tr>
</table>

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 is used for this parameter.
UNLCFBF	Computed	Unbraced length of the compression flange for the bottom flange. When no value is specified, UNLCF and FRUNLCF is used for this parameter.

Table IS800 (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.

```
                                                     { ACI318-99        }
                                                     { ACI318-89        }
                                                     { ACI318-83        }
METHOD ( { → ULTIMATE (STRENGTH) } ) { ACI318-77        } -
         {   WORKING (STRESS)    }                   { ACI318-63        }
                                                     { (BSI) CP110-72   }
                                                     { (BSI) BS8110     }

         { → ASTM                 }
         {   CANADIAN (STANDARD)  }       { → NONSEISMIC        }
( BARS   {   UNESCO               } ) (   {   SEISMIC           } )
         {   KOREAN (STANDARD)    }       {   MODERATE SEISMIC  }
```

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	$29x10^6$psi
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

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

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

$$* \left\{ \begin{array}{l} \underline{\text{ENERGY}} \ (\underline{\text{NORM}}) \\ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \\ \underline{\text{PERCENT}} \ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{PERCENT}} \ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \\ \underline{\text{NORM}}\text{ALIZED} \ \underline{\text{PERCENT}} \ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{NORM}}\text{ALIZED} \ \underline{\text{PERCENT}} \ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \end{array} \right\} \ -$$

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

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

The L_2 norm is given by:

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

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

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

Difference from Average Method

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

Percent Maximum Difference Method

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

Percent Difference from Average Method

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

Normalized Percent Maximum Difference

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

Normalized Percent Difference from Average Method

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

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

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

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

1. Enter GTMenu.

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

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

4. Select the Loading.

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

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

5.3.2 Finite Element Dictionary Revisions - Temperature Gradient Loading Additions for Plate Bending and Plate Elements

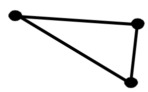
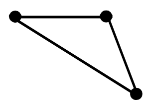
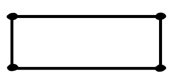
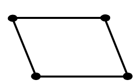
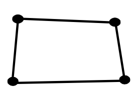
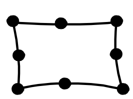
Temperature gradient loading capability has been added for the BPHT and BPHQ plate bending elements and the SBHT, SBHQ, SBHQCSH, SBHT6, and SBHQ6 plate elements. The revisions to the Finite element dictionary (Table 2.3.1, Volume 3) are shown on the following pages.

Table 2.3-1

GTSTRUDL Finite Element Dictionary

Element					Input				
Element Name	Shape	No. of Nodes	D.O.F.	Rigidity Matrix	Joint Loads		Element Loads		
					Concentrated	Temperature Gradient	Edge	Surface	Body

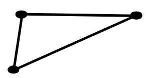
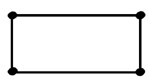
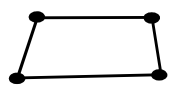
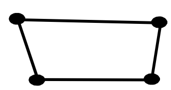
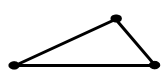
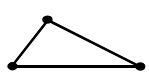
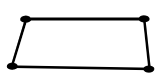
TYPE -- Plate Bending

Element Name	Shape	No. of Nodes	D.O.F.	Rigidity Matrix	Concentrated	Temperature Gradient	Edge	Surface	Body
CPT		3	u_3, u_4, u_5	x	x	x		x	x
BPHT		3	u_3, u_4, u_5	x	x	⊗		x	x
BPR		4	u_3, u_4, u_5	x	x	x		x	x
BPP		4	u_3, u_4, u_5	x	x			x	x
BPHQ		4	u_3, u_4, u_5	x	x	⊗		x	x
IPBQQ		8	u_3, u_4, u_5	x	x	x		x	x

⊗ New in Version 27

Table 2.3-1

GTSTRUDL Finite Element Dictionary (continued)

Element					Input					
					Joint Loads			Element Loads		
Element Name	Shape	No. of Nodes	D.O.F.	Rigidity Matrix	Concentrated	Temperature Change	Temperature Gradient	Edge	Surface	Body
TYPE -- Plate										
SBCT		3	u_1, u_2, u_3 u_4, u_5	x	x	x	x		x	x
SBCR		4	u_1, u_2, u_3 u_4, u_5	x	x	x	x		x	x
SBHQ		4	u_1, u_2, u_3 u_4, u_5	x	x	x	⊗		x	x
SBHQCSH		4	u_1, u_2, u_3 u_4, u_5	x	x	x	⊗		x	x
SBHT		3	u_1, u_2, u_3	x	x	x	⊗		x	x
SBHT6		3	u_1, u_2, u_3 u_4, u_5, u_6	x	x	x	⊗		x	x
SBHQ6		4	u_1, u_2, u_3	x	x	x	⊗		x	x

⊗ New in Version 27

This page intentionally left blank.

5.3.3 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

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

2.4.3.7 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

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

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

2.4.3.7.1 The DAMPER ELEMENT DATA Command

Tabular form:

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

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

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

•

•

•

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

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

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

Elements:

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

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

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

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

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

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

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

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

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

Explanation:

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

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

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

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

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

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

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

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

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

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

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

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

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

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:

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

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.

5.3.4 Dynamic Analysis External File Solver - Improve Efficiency of Dynamic Results Computation

A new database and file structure are now available for dynamic analysis. When the DYNAMIC PARAMETERS option USE EXTERNAL FILE SOLVER is used, the time to compute dynamic analysis results may be significantly reduced. The DYNAMIC PARAMETERS command with the new options, USE EXTERNAL FILE SOLVER and RESULTS FILE NAME is shown below. The section is numbered as it will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.4.5.3 Specification of Miscellaneous Dynamic Parameters

General form:

```
DYNAMIC  PARAMETERS
           ⎧ ⎧VELOCITY    ⎫ ⎧ON ⎫           ⎫
           ⎪ ⎩ACCELERATION⎭ ⎩OFF⎭           ⎪
   STORE   ⎨                                ⎬
           ⎪ ⎧ABSOLUTE⎫                     ⎪
           ⎩ ⎩RELATIVE⎭  ACCELERATION       ⎭
   DURATION  (OF  EARTHQUAKE)  v
         ⎧EXTERNAL⎫
   USE   ⎨        ⎬  (FILE SOLVER)
         ⎩INTERNAL⎭
   RESULTS  FILE  NAME  'fn'
END  (OF  DYNAMIC  PARAMETERS)
```

Elements:

v = duration of earthquake to be used in a NRC Double Sum Method response spectrum combination. The default is taken as 10 seconds.

'fn' = alphanumeric string to be used as the prefix in the construction of the dynamic analysis results file names when the USE EXTERNAL FILE SOLVER option is given. The length of the alphanumeric string is limited to 24 characters.

Explanation:

This tabular command is used to specify miscellaneous parameters related to dynamic analysis.

The STORE command determines whether nodal velocities and accelerations will be stored in a transient or steady state analysis. If either or both are not required, then the OFF option will bypass their computation and storage, which will result in decreased execution times and the use of less hard drive file space. Note that only displacements are used in the back substitution computation of forces, stresses, and reactions. If the STORE command is not given, then both velocities and accelerations are computed and stored.

Also under the STORE command, the ABSOLUTE and RELATIVE options provide for the choice between the computation and storage of either absolute or relative acceleration during a support acceleration transient analysis. This choice will be important if the CREATE TIME HISTORY command (Section 2.4.8.1) is to be used. If the command is not given, then RELATIVE is assumed.

The DURATION OF EARTHQUAKE command is used to specify the quantity needed in the NRC Double Sum Method of modal combination. The formula appears in Section 2.4.2.5.

The USE EXTERNAL FILE SOLVER command specifies that the results from a subsequent transient or response spectrum analysis are to be stored in dynamic analysis results files on the hard disk rather than virtual memory and the GTSTRUDL data base. For large transient analysis or response spectrum analysis jobs, this may reduce the execution time dramatically over the use of virtual memory to store the results. The execution time savings have been observed to be an order of magnitude or more. The USE INTERNAL FILE SOLVER command is used to revert back to the virtual memory storage of transient and response spectrum results if the USE EXTERNAL FILE SOLVER command had been specified previously.

The RESULTS FILE NAME command specifies an alphanumeric string to be used as a file name prefix in the creation of the transient and response spectrum results file names when the USE EXTERNAL FILE SOLVER command is given. 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'.

The convention for constructing the transient and response spectrum results file names is described as follows:

Dynamic results file name = fn + load id + .ext

where

fn	=	file name prefix as described above,
load id	=	the name of the dynamic loading for which the results are computed,
.ext	=	a three-character file name extension indicating the type of dynamic results stored in the file.

The values for .ext and the corresponding dynamic analysis results types are described as follows:

File Extension	Description
.dsp	joint displacements, computed automatically by a transient analysis or by the COMPUTE command for a response spectrum analysis (Section 2.4.5.9, Volume 3, GTSTRUDL User Reference Manual),
.vel	joint velocities, computed automatically by a transient analysis or by the COMPUTE command for a response spectrum analysis (Section 2.4.5.9, Volume 3, GTSTRUDL User Reference Manual),
.acc	joint accelerations, computed automatically by a transient analysis or by the COMPUTE command for a response spectrum analysis (Section 2.4.5.9, Volume 3, GTSTRUDL User Reference Manual),
.lmf	linear member and finite element forces, computed by the COMPUTE command for transient analysis (Section 2.4.5.7, Volume 3, GTSTRUDL User Reference Manual) and response spectrum analysis (Section 2.4.5.9, Volume 3, GTSTRUDL User Reference Manual),
.nmf	nonlinear member and element forces computed automatically by a nonlinear dynamic transient analysis,

.les linear finite element stresses and strains, computed by the COMPUTE command for transient analysis (Section 2.4.5.7, Volume 3, GTSTRUDL User Reference Manual) and response spectrum analysis (Section 2.4.5.9, Volume 3, GTSTRUDL User Reference Manual),

.rea support reactions, computed by the COMPUTE command for transient analysis (Section 2.4.5.7, Volume 3, GTSTRUDL User Reference Manual) and response spectrum analysis (Section 2.4.5.9, Volume 3, GTSTRUDL User Reference Manual),

.lds resultant joint loads, computed by the COMPUTE command for transient analysis (Section 2.4.5.7, Volume 3, GTSTRUDL User Reference Manual) and response spectrum analysis (Section 2.4.5.9, Volume 3, GTSTRUDL User Reference Manual).

It is recommended that the RESULTS FILE NAME command be given only once in a given job, regardless of the number of dynamic analyses performed for different transient or response spectrum loading conditions. For example, giving a new RESULTS FILE NAME command prior to each dynamic analysis for a new 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 or response spectrum 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.

The END command is used to terminate the tabular command input.

5.3.5 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 }                       }
                                     { {STORY}   (SHEAR)             }

                                   * { DISPLACEMENTS }
                                     { VELOCITIES    }
                                     { ACCELERATIONS }  -
                                     { FORCES        }
                                     { LOADS         }
                                     { REACTIONS     }

                                   * { { RMS        } }
                                     { { ABS        } }
                                     { { PRMS       } }
                                     { { CQC        } }
((MODAL)  COMBINATIONS               { { (NRC) TPM  } }  )   (list)
                                     { { (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.

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

General form:

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

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

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.
$'title_{sl}'$	=	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:

<u>MAS</u>S [<u>X</u>] v_x [<u>Y</u>] v_y [<u>Z</u>] 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:

$$\{F_{EM}\} = [\,M\,]\,\{v_{XYZ}\}\,g \qquad \text{Eq. 5.3.6-1}$$

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 5.3.6-1, 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 NEHRP Guidelines for the Seismic Rehabilitation of Buildings (FEMA Publication 273).

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

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:

$$\{ f_{RS} \}_i = - v_{RS} \Gamma_i S_{ai} [M] \{ \Phi_i \} \qquad \text{Eq. 5.3.6-2}$$

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.

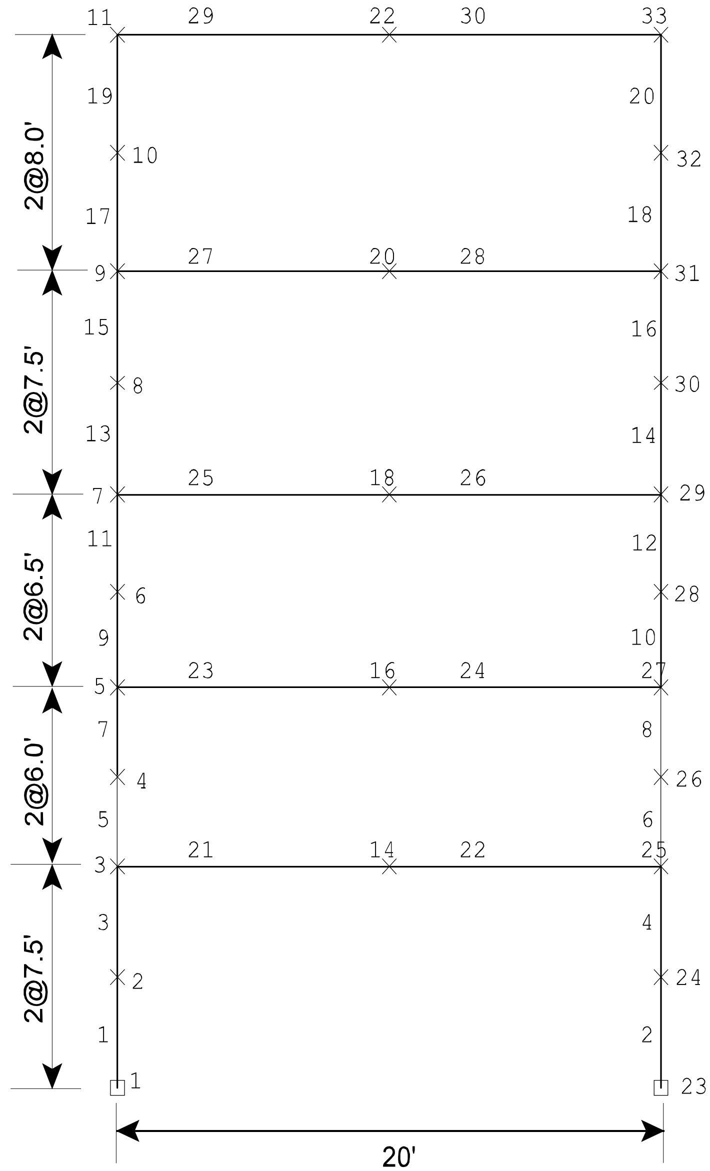
```
**** ERROR_STGELL -- Results do not exist for response
                     spectrum loading 1-G.2. Response
                     spectrum analysis has not yet been run or
                     the specified loading is not a response
                     spectrum load.  SCAN MODE entered.
```

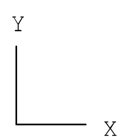
This error message indicates that while the specified response spectrum load is valid, the required response spectrum analysis for this load has not yet been executed. SCAN MODE is set and may be removed by giving the SCAN OFF command.

Example:

Figure 5.3.6-1 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 5.3.6-2.





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 5.3.6-1 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 5.3.6-2 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 5.3.6-2 Command Listing for Example SEL-1 (Continued)

Figure 5.3.6-3 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 5.3.6-3 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 5.3.6-3 PRINT APPLIED JOINT LOAD Results for Example SEL-1 (Continued)

This page intentionally left blank.

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

General form:

$$\underline{\text{FOR}}\text{M}\ \ \underline{\text{UBC97}}\ \ (\underline{\text{STAT}}\text{IC})\ \ (\underline{\text{SEI}}\text{SMIC})\ \ \underline{\text{LOAD}}\ \left\{\begin{matrix}'a_{sl}'\\ i_{sl}\end{matrix}\right\}\ ('\text{title}_{sl}')\ -$$

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

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

$$\underline{\text{CT}}\left\{\begin{matrix}\underline{\text{SMR}}\\ \underline{\text{RCMR}}\\ \underline{\text{OTHER}}\\ v_{CT}\end{matrix}\right\}\ \underline{\text{R}}\ \ v_R\ \ \left((\underline{\text{WITH}})\ \underline{\text{TOR}}\text{SION}\left\{\begin{matrix}\rightarrow \underline{\text{PLUS}}\\ \underline{\text{MIN}}\text{US}\end{matrix}\right\}\right)\ \ (\underline{\text{FLO}}\text{OR}\ \ (\underline{\text{TOL}}\text{ERANCE})\ \ v_{TOL})$$

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.

$'\text{title}_{sl}'$ = 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:

$$\underline{\text{DIR}}\text{ECTION} \begin{Bmatrix} \rightarrow \underline{X} \\ \underline{Y} \\ \underline{Z} \end{Bmatrix}$$

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.

$$\underline{\text{HEI}}\text{GHT} \;\; (\underline{\text{DIR}}\text{ECTION}) \begin{Bmatrix} \underline{X} \\ \rightarrow \underline{Y} \\ \underline{Z} \end{Bmatrix}$$

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.

$$\underline{\text{WEI}}\text{GHT} \;\; (\underline{\text{LOA}}\text{D}) \begin{Bmatrix} 'a_w' \\ i_w \end{Bmatrix}$$

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.

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

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.

$$* \left\{ \begin{matrix} \underline{\text{SOI}}\text{L} \left\{ \begin{matrix} \underline{SA} \\ \underline{SB} \\ \underline{SC} \\ \underline{SD} \\ \underline{SE} \\ \underline{SF} \end{matrix} \right\} \\ \underline{CA}\ \ v_{CA} \\ \underline{CV}\ \ v_{CV} \\ \underline{NA}\ \ v_{NA} \\ \underline{NV}\ \ v_{NV} \end{matrix} \right\}$$

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.

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

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.

$$\underline{\text{CT}}\ \begin{Bmatrix} \underline{\text{SMR}} \\ \underline{\text{RCMR}} \\ \underline{\text{OTH}}\text{ER} \\ v_{CT} \end{Bmatrix}$$

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.

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

General Form:

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

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

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

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

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:

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

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.

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

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.

$$\underline{\text{WEI}}\text{GHT}\ \ (\underline{\text{LOA}}\text{D})\ \begin{Bmatrix} 'a_W' \\ i_W \end{Bmatrix}$$

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.

$$\underline{\text{ZON}}\text{E}\ \begin{Bmatrix} \underline{\text{II}} \\ \underline{\text{III}} \\ \underline{\text{IV}} \\ \underline{\text{V}} \\ \underline{\text{Z}}\ v_Z \end{Bmatrix}$$

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.

$$\underline{\text{SOI}}\text{L}\ \ (\underline{\text{TYP}}\text{E})\ \begin{Bmatrix} \underline{\text{I}} \\ \underline{\text{II}} \\ \underline{\text{III}} \end{Bmatrix}$$

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.

PERIOD (TYPE) $\left\{ \begin{array}{l} \text{SMR} \\ \text{RCMR} \\ \text{OTHER} \\ v_T \end{array} \right\}$

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.

5.3.9 Nonlinear Effects Command

The Nonlinear Effect command has been modified to include nonlinear spring connections and plastic hinges. These modifications are shown in the section below which is numbered as it will appear when added to Volume 3 of the GTSTRUDL Reference Manuals.

2.5.2.1 Nonlinear Effects Command

The NONLINEAR EFFECTS command is used to specify those members for which geometric nonlinearity is to be considered and those members which are to act in tension or compression only.

General form:

NONLINEAR EFFECTS (list_{DEL})

(nonlinear member specs)

•

•

•

(nonlinear member specs)

where

$$\text{nonlinear member specs} = \left\{\begin{array}{l}\text{TENSION (ONLY)}\\ \text{COMPRESSION (ONLY)}\\ \text{GEOMETRY (AXIAL)}\\ \text{NLS Connection Specs}\\ \text{Plastic Hinge Specs}\end{array}\right\} \text{list}_1$$

and where

NLS Connection Specs =

$$\text{NLS (CONNECTION) MEMBERS list}_2 \; {}^{*}\left\{\begin{array}{l}\text{START Curve Specs}\\ \text{END} \quad \text{Curve Specs}\end{array}\right\}$$

$$\text{Curve Specs} = {}^{*}\left\{\begin{array}{l}\underline{\text{FOR}}\text{CE Force / Moment Specs}\\ \underline{\text{MOM}}\text{ENT Force / Moment Specs}\end{array}\right\}$$

$$\text{Force / Moment Specs} = {}^{*}\left\{\begin{array}{l}\underline{\text{X}}\left\{\begin{array}{l}i_X\\ 'a_X'\end{array}\right\}\\ \underline{\text{Y}}\left\{\begin{array}{l}i_Y\\ 'a_Y'\end{array}\right\}\\ \underline{\text{Z}}\left\{\begin{array}{l}i_Z\\ 'a_Z'\end{array}\right\}\end{array}\right\}$$

$$\text{Plastic Hinge Specs} = \underline{\text{PLA}}\text{STIC}\left\{\begin{array}{l}\underline{\text{HIN}}\text{GE}\\ \underline{\text{SEG}}\text{MENT}\end{array}\right\}{}^{*}\left\{\begin{array}{l}\underline{\text{STA}}\text{RT}\\ \underline{\text{END}}\end{array}\right\}\text{, Fiber Specs, -}$$

$${}^{*}\left\{\begin{array}{l}\text{Steel Specs}\\ \text{R - C Specs}\end{array}\right\}$$

$$\text{Fiber Specs} = \underline{\text{FIB}}\text{ER (}\underline{\text{GEOM}}\text{ETRY)}\ {}^{*}\left\{\begin{array}{l}[\underline{\text{NBF}}]\ i_{NBF}\\ [\underline{\text{NB}}]\ i_{NB}\\ [\underline{\text{NR}}]\ i_{NR}\\ [\underline{\text{NTF}}]\ i_{NTF}\\ [\underline{\text{NTW}}]\ i_{NTW}\\ [\underline{\text{ND}}]\ i_{ND}\\ [\underline{\text{NTWALL}}]\ i_{NTWALL}\\ [\underline{\text{NTH}}]\ i_{NTH}\\ [\underline{\text{LH}}]\ v_{LH}\end{array}\right\}$$

$$\text{Steel Specs} = \left\{\begin{array}{l}\underline{\text{CUR}}\text{VE}\left\{\begin{array}{l}i_{curve}\\ 'a_{curve}'\end{array}\right\}\\ [\underline{\text{FY}}]\ v_{FY}\ \ [\underline{\text{EH}}]\ v_{EH}\ \ [\underline{\text{ESH}}]\ v_{ESH}\ \ [\underline{\text{FSU}}]\ v_{FSU}\ \ \text{-}\\ \qquad\qquad [\underline{\text{ESU}}]\ v_{ESU}\ \ [\underline{\text{ALPHA}}]\ v_{\alpha}\end{array}\right\}$$

$$\text{R - C Specs} = \underline{\text{R - C}}\left\{\begin{array}{l}\underline{\text{RECT}}\text{ANGLE }[\underline{\text{B}}]\ v_B\ [\underline{\text{H}}]\ v_H\\ \underline{\text{CIRC}}\text{ULAR }[\underline{\text{B}}]\ v_{DIAM}\end{array}\right\}\left\{\begin{array}{l}\underline{\text{CUR}}\text{VE}\left\{\begin{array}{l}i_{curve}\\ 'a_{curve}'\end{array}\right\}\\ [\underline{\text{FCP}}]\ v_{FCP}\ \ [\underline{\text{EC0}}]\ v_{EC0}\end{array}\right\}\ \text{-}$$

$$[\underline{\text{FYS}}]\ v_{FYS}\quad (\underline{\text{BAR}}\text{S}\left\{\begin{array}{l}\underline{\text{ASTM}}\\ \underline{\text{CANA}}\text{DIAN}\\ \underline{\text{UNESCO}}\\ \underline{\text{KOR}}\text{EAN}\end{array}\right\})\ (\,{}^{*}\left\{\begin{array}{l}\underline{\text{STA}}\text{RT Bar Specs}\\ \underline{\text{END}}\text{ Bar Specs}\end{array}\right\})$$

Bar Specs = $\left(\begin{Bmatrix} \underline{\text{TOP}} \\ \underline{\text{CIRC}}\text{ULAR} \end{Bmatrix} i_{NT}\ i_{BT}\right)$, $(\underline{\text{BOT}}\text{TOM}\ \ i_{NBB}\ i_{BB})$, $(\underline{\text{SID}}\text{E}\ \ i_{NS}\ i_{BS})$, -

$\left(\begin{Bmatrix} \underline{\text{HOO}}\text{P}\ \ i_{BH}\ v_{HS} \\ \underline{\text{TIE}}\ \ i_{BTIE}\ \ i_{NLY}\ i_{NLZ}\ v_{TS} \end{Bmatrix}\right)$, $(\underline{\text{COV}}\text{ER}\ v_{COV})$

Friction Damper Specs = <u>FRI</u>CTION (<u>DAM</u>PER) [<u>FSL</u>] f_{slip}

Elements:

$$\text{list}_{DEL} = \text{list}_1 = \begin{Bmatrix} \underline{\text{ALL}}\ (\underline{\text{MEM}}\text{BERS}) \\ \text{list}_2 \end{Bmatrix}$$

list_2 = list of alphanumeric/integer member id's,

i_X/a_X = eight-character integer or alphanumeric identifier of the spring curve defining the NLS connection force-displacement behavior in the local member x direction or moment rotation behavior about the local member x axis,

i_Y/a_Y = eight-character integer or alphanumeric identifier of the spring curve defining the NLS connection force-displacement behavior in the local member y direction or moment rotation behavior about the local member y axis,

i_Z/a_Z = eight-character integer or alphanumeric identifier of the spring curve defining the NLS connection force-displacement behavior in the local member z direction or moment rotation behavior about the local member z axis,

i_{NBF} = number of plastic hinge fiber divisions along the flange width of steel wide flange, tee, and structural tube plastic hinge sections,

i_{NB} = number of fiber divisions along the width of rectangular reinforced concrete plastic hinge sections,

i_{NR} = number of fiber divisions along the radius of circular reinforced concrete plastic hinge sections,

i_{NTF} = number of fiber divisions through the flange thickness of steel wide flange, tee, and structural tube plastic hinge sections,

i_{NTW} = number of fiber divisions through the web thickness of steel wide flange, tee, and structural tube plastic hinge sections,

i_{ND} = number of fiber divisions through the web depth of steel wide flange, tee, and structural tube section or the number of fiber divisions through the depth of rectangular reinforced concrete plastic hinge sections,

i_{NTWALL}= number of fiber division through the wall thickness of steel structural tube and pipe plastic hinge sections,

i_{NTH} = number of fiber divisions around the circumference of steel pipe plastic hinge sections or circular reinforced concrete plastic hinge sections,

i_{curve} = integer identifier of the custom piece-wise linear stress strain curve for the concrete or steel stress-strain properties,

a_{curve} = alpha-numeric identifier of the custom piece-wise linear stress strain curve for the concrete or steel stress-strain properties,

v_{LH} = length of plastic hinge,

v_{FY} = yield stress of steel,

v_{EH} = strain hardening modulus of steel,

v_{ESH} = strain of steel at the onset of strain hardening,

v_{FSU} = peak steel stress,

v_{ESU} = steel strain corresponding to V_{FSU},

v_{α} = residual stress factor,

v_B = overall width of rectangular reinforced concrete plastic hinge sections,

v_H = overall depth of rectangular reinforced concrete plastic hinge sections,

v_{DIAM} = overall diameter of circular reinforced concrete plastic hinge sections,

v_{FCP} = compressive strength, f'_c, of unconfined concrete,

v_{EC0} = ultimate strain of unconfined concrete,

v_{FYS} = yield stress of hoops, spirals, or ties in reinforced concrete plastic hinges,

i_{NT} = number of top reinforcing bars for rectangular reinforced concrete plastic hinge sections or the number of circular reinforcing bars for circular reinforced concrete plastic hinge sections,

i_{BT} = size of top reinforcing bars for rectangular reinforced concrete plastic hinge sections, or circular reinforcing bars for circular reinforced concrete plastic hinge sections,

i_{NBB} = number of bottom reinforcing bars for rectangular reinforced concrete plastic hinge sections,

i_{BB} = size of bottom reinforcing bars for rectangular reinforced concrete plastic hinge sections,

i_{NS} = number of reinforcing bars on the sides of rectangular reinforced concrete plastic hinge sections,

i_{BS} = size of reinforcing bars on the sides of rectangular reinforced concrete plastic hinge sections,

i_{BH} = bar size of hoop or spiral reinforcing in circular reinforced concrete plastic hinge sections,

v_{HS} = on-center hoop spacing or spiral pitch,

i_{BTIE} = bar size of tie reinforcing in rectangular reinforced concrete plastic hinge sections,

i_{NLY} = number of tie legs in the local member y direction of rectangular reinforced concrete plastic hinge sections,

i_{NLZ} = number of tie legs in the local member z direction of rectangular reinforced concrete plastic hinge sections,

v_{TS} = on-center tie spacing,

v_{COV} = clear cover for rectangular and circular reinforced concrete plastic hinge sections,

f_{slip} = tension/compression force at which slip occurs for the friction damper effect.

Explanation:

The TENSION ONLY, COMPRESSION ONLY, GEOMETRY, NLS connection, and Plastic hinge options are described below:

TENSION ONLY

Indicates that the specified plane and space truss members may carry only tension loads. The sign of the axial force in these members is checked at the end of each equilibrium iteration or cycle (See Figure 2.5.1.2-1) and members in which compression is detected are eliminated from the solution in the next equilibrium iteration.

The TENSION ONLY spec applies to plane truss, space truss, plane frame, and space frame members only.

COMPRESSION ONLY

Indicates that the specified plane and space truss members may carry only compression loads. The sign of the axial force in these members is checked at the end of each equilibrium iteration or cycle (See Figure 2.5.1.2-1) and members in which tension is detected are eliminated from the solution in the next equilibrium iteration.

The COMPRESSION ONLY spec applies to plane truss, space truss, plane frame, and space frame members only

COMPRESSION AND TENSION ONLY may not be specified for the same member.

GEOMETRY

Indicates that the specified plane truss, plane frame, space truss and space frame members and rigid body elements can model nonlinear geometric effects resulting from axial force and member displacement. In frame members, these effects are often referred to as P-Δ and P-Y effects. An example of the formulation of the geometric stiffness used to account for this nonlinearity may be found in Reference 1. At the present time, geometric nonlinearity for frame members is limited to doubly-symmetric cross sections. Strains and rotations must be "small" for the plane frame and space frame members and rotations must be "small" for rigid body elements. Rotations may be of any finite size for the plane truss and space truss members; however, strains must remain "small." Nonlinear geometric behavior may be modeled in combination with tension or compression only behavior for plane and space truss members.

The AXIAL option causes the effects of elongation and shortening due to member chord rotation to be approximated for the nonlinear geometric behavior of plane and space frame members. Although this effect is of little significance for typical frame structures, it is required in order to predict the mode of nonlinear behavior typified by the tension force induced in a simple beam whose supports are restrained from moving together longitudinally. The AXIAL option is not operative by default.

Note:

1. The term "small" with regard to strains means that the strains are in the range where the assumption of linear elastic engineering stress-strain behavior is valid (1).

2. The term "small" with regard to rotations means that rotations are in the range where $\sin \theta \sim \theta$.

3. The GEOMETRY option may be specified for plane and space truss members and plane and space frame members only.

NLS Connection Specs

The NLS connection specs are used to define nonlinear spring element connections attached to the START and/or END of a frame member. The nonlinear spring connection curve specs and force/moment specs provide for the specification of any combination of three uncoupled displacement degrees-of-freedom and three uncoupled rotation degrees-of-freedom. The force-displacement and moment-rotation properties of the NLS connection degrees-of-freedom are defined with respect to the local member coordinate system and are derived from the nonlinear spring curves identified by the names i_X/'a_X', i_Y/'a_Y', and i_Z/'a_Z'. The nonlinear spring curves, names and properties, are defined by the NONLINEAR SPRING PROPERTIES command described in Section 2.5.3.3 below.

NLS connections may be used in combination with any other nonlinearity, including plastic hinges, with the exception that the specified NLS connection degrees-of-freedom are not permitted to overlap with the plastic hinge axial and two bending moment degrees-of-freedom.

Plastic Hinge Specs

The plastic hinge specs provide for the definition of plastic hinge formation at the start and/or end of a frame member. Plastic hinges are defined by the PLASTIC HINGE/SEGMENT START/END specification, Fiber Specs, Steel Specs, and R-C Specs for reinforced concrete plastic hinges. In the discussion that follows, the term "plastic hinge" refers to both the plastic hinge and plastic segment models.

The PLASTIC SEGMENT implementation does not permit member loads for the designated members.

The plastic hinge specification begins with the selection of the PLASTIC HINGE/SEGMENT START/END options, which indicate the type of member end plasticity model and the member ends at which the plastic hinges are permitted to form. The HINGE selection indicates that the plasticity is modeled as a discrete zero-length spring lumped at the START and/or END of the specified members. The SEGMENT selection indicates that the plasticity is distributed along the length of a segment of length LH (plastic hinge length) at the START and/or END of the specified members. The plastic segment model is an enhanced version of the plastic hinge model and in particular, provides a somewhat more accurate representation of member flexibility throughout the full range of member deformations. The specification of START and END is optional, with both START and END as the default if neither is specified.

The START and END specification is followed by the Fiber Specs, which define the geometric characteristics of the plastic hinge. A plastic hinge may be considered as a member segmental body attached to the START/END of the member, having a finite length and a specific cross section shape and size. The plastic hinge therefore has a finite volume. The plastic hinge may be reinforced concrete or totally steel. The steel plastic hinge may have a wide flange, tee, channel, pipe, or structural tube cross section shape based on table sections or pipe properties defined for the member by the MEMBER PROPERTIES command. The reinforced concrete plastic hinge may have a rectangular or circular cross section shape.

In order to track the nonlinear material behavior through the plastic hinge volume, the cross section is divided into a grid of prismatic "fibers" according to the specified fiber division data item values of the Fiber Specs. The fiber division data values give the number of fiber divisions with respect to the length, width, and thickness of various plastic hinge cross section components such as the flanges and web of a wide flange cross section. Table 2.5.2-1 below summarizes the available plastic hinge cross section shapes and the Fiber Specs data items required to specify the fiber grid divisions.

Cross Section Shape	Required Fiber Specs Data Values	
Wide Flange	NBF –	number of fiber divisions across the width of the flange.
	NTF –	number of fiber divisions through the thickness of the flange.
	ND –	number of fiber divisions through the clear depth of the web, not including the flange thicknesses.
	NTW –	number of fiber divisions through the thickness of the web.
	LH –	plastic hinge length.
Tees	Same as Wide Flange	
Channels	Same as Wide Flange	

Pipe	NTH –	number of fiber divisions around the circumference of the pipe.
	NTWALL –	number of fiber divisions through the thickness of the pipe wall.
	LH –	plastic hinge length.
Structural Tube	NBF –	number of fiber divisions across the straight portion of the width of the tube.
	ND –	number of fiber divisions through the straight portion of the depth of the tube.
	NTWALL –	number of fiber divisions through the thickness of the tube wall.
	LH –	plastic hinge length.
R-C Rectangle	NB –	number of fiber divisions across the width of the rectangular cross section.
	ND –	number of fiber divisions through the depth of the cross section.
	LH –	plastic hinge length.
R-C Circular	NR –	number of fiber divisions through the radius of the cross section.
	NTH –	number of fiber divisions around the circumference of the cross section.
	LH –	plastic hinge length.

Table 2.5.2-1 Required Fiber Specs Data Values

Note that the Fiber Specs also include the data item parameter LH, which is used to specify the length of the plastic hinge. The default value for LH is 1.0 inches; however, testing to date has shown that values between 2.0 and 4.0 inches produce reasonable results. Smaller values produce stiffer behavior, larger values produce more flexible behavior. However, very small values for LH, less than 0.50 inches, can cause strains in the plastic hinge fibers to become excessively large very quickly, increasing the likelihood that the steel peak strain parameter ESU will be exceeded. This will dramatically reduce the strength and stiffness of the plastic hinge cross-section because the fiber steel stress is assumed to be 0.0 for strains greater than ESU.

Note further that the fiber division data items for the structural tube shape apply only to the straight portions of the walls. One fiber division is always assumed for the curved corner portions.

Figure 2.5.2-1 shows a plastic hinge segment of length LH and having a tee cross section shape. The flange is divided into two fibers through the thickness and 18 fibers across the width (NTF 2, NBF 18). The web is divided into 16 fibers through the depth and one fiber through the thickness (ND 16, NTW 1).

The larger the number of fiber divisions for a plastic hinge cross section, the more accurate the plastic hinge behavior, but also the more costly the solution in terms of the time needed to compute plastic hinge stiffnesses and forces, and the time and space needed to store the plastic hinge stress and strain data. The total number of fiber divisions can get out of hand very quickly, particularly for reinforced concrete plastic hinges where the total plastic hinge cross section area may be quite substantial.

Additional important physical characteristics of plastic hinges include the following:

1. The cross section principal y and z axes and the centroidal x axis have the same orientation as the local member coordinate system. Plastic hinge displacements and forces are defined with respect to the member/plastic hinge principal axes.

2. The PLASTIC HINGE has two coincident nodes at which three displacement degrees-of-freedom are defined with respect to the local member coordinate system: axial displacement along the centroidal axis and two rotations about the principal axes. For a plastic hinge at the start of the member: the plastic hinge degrees-of-freedom at the plastic hinge end node are connected to the same degrees-of-freedom at the member start node. For a plastic hinge at the end of a member: the plastic hinge degrees-of-freedom at the plastic hinge start node are connected to the same degrees-of-freedom at the member end node.

3. With the PLASTIC SEGMENT model, the start/end of the elastic portion of the member is replaced with a plastic segment of finite length LH. The plastic segment has a start node and end node and is assembled with the adjusted elastic length [L - ns(LH), where ns is the number of specified plastic segments] of the member according to conventional finite element methods. The assembled plastic segment member therefore has 12 + 6*ns degrees of

freedom. For example, a member having plastic segments at both the start and end has 24 degrees of freedom.

4. For the purpose of computing stresses in plastic hinges with the rolled steel cross section shapes, the cross sections are assumed to be totally compact; that is, any effects of in-plane cross section distortion, such as local buckling, are not considered.

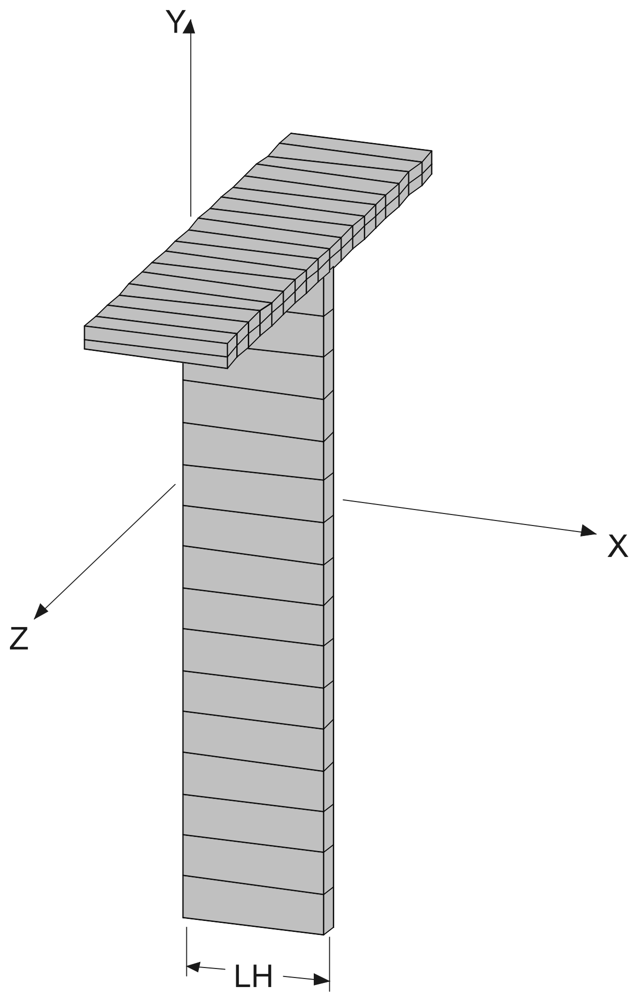


Figure 2.5.2-1 A Tee-Shaped Plastic Hinge, Showing Fiber Divisions

The Steel Specs include the data items FY, ESH, EH, ESU, and FSU, and define the stress-strain properties of the steel of rolled shape plastic hinges (wide flange, tee, channel, pipe, and tube) and the primary reinforcing steel in R-C plastic hinges. The assumed uniaxial stress-strain curve for steel, due to Balan et.al. (88), is shown in Figure 2.5.2-2 below; the Steel Specs data items are labeled. The default values for the Steel Specs data items are summarized in Table 2.5.2-2.

Steel specs Data Item	Default Value
E	29000 ksi
FY	60 ksi
ESH	(32 - .45FY)(FY/29000) (FY in ksi)
EH	EH = 0.5*(FSU - FY)/(ESY - ESH)
ESU	0.05
FSU	1.5FY

Table 2.5.2-2 Default Values for Steel Spec Data Items

Note that (ESU, FSU) represents the peak steel strain and stress respectively, and not the ultimate fracture strain and corresponding stress. It is assumed that stress and strain values beyond FSU and ESU are highly sample-dependent and not useful; therefore, for strains that exceed ESU, steel stress is taken as 0.0.

Steel stress-strain properties also may be defined using the CURVE option to specify the name of a custom piece-wise linear stress-strain curve that was previously defined and stored using the STORE STRESS STRAIN CURVE command (see Section 2.5.2.2.1 below).

The steel stress-strain behavior is assumed to be identical for both tension and compression and also nonlinear elastic at this time (loading and unloading follows the same stress-strain curve path) implying that the constitutive behavior does not include the effects of plastic unloading. This generally leads to conservative pushover analysis predictions of collapse loads (actual collapse loads being greater than predicted collapse loads); however, this fact must be considered and may have important implications, particularly with respect to applied loading sequences.

The Steel Specs also include the ALPHA data item, which is used to define the effect of residual stresses on steel wide flange plastic hinges. Any positive value for ALPHA is allowed; however, in practice, ALPHA should be 0.5 or less. The assumed wide flange residual stress distribution is shown in Figure 2.5.2-3.

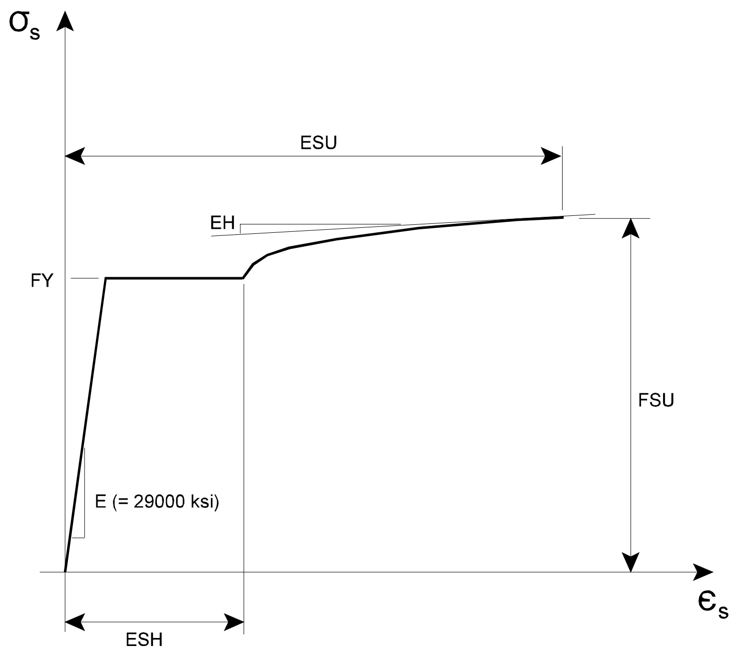


Figure 2.5.2-2 Assumed Steel Stress-Strain Curve for Plastic Hinges

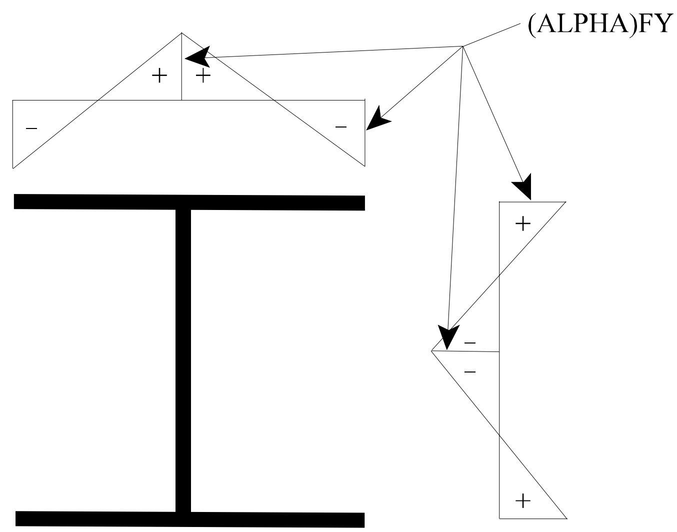


Figure 2.5.2-3 Assumed Residual Stress Distribution for Wide Flange Plastic Hinges

The R-C Specs that define the properties of reinforced concrete plastic hinges include cross section shape, cross section dimensions, principal concrete material properties, and reinforcing steel locations. If R-C Specs are not given, the plastic hinges are assumed to be steel only and the member properties must correspond to one of the supported rolled steel cross section shapes: wide flange, tee, channel, pipe, and structural tube.

The R-C Specs data that apply generally to both start and end plastic hinges include the RECTANGLE and CIRCULAR cross section data items, the concrete material data items FCP, EC0, and FYS, and the BARS selection. R-C plastic hinges may have either a RECTANGLE or CIRCULAR cross section shape where the data items B and H define the width and depth of the rectangular cross section respectively and the B data item defines the diameter of circular cross sections.

The compression stress-strain behavior of the concrete is based on the confined concrete compression stress-strain model due to Mander et.al. (89), which is illustrated in Figure 2.5.2-3 on the next page, where the input data items FCP refer to f'_{c0} and EC0 refers to $\in'_{c0}$. Tension stress-strain behavior is assumed to be linear up to the modulus of rupture, $75\sqrt{f'_{c0}}$ psi, with an elastic modulus of $60200\sqrt{f'_{c0}}$ psi, which is also assumed to be the initial elastic modulus in compression. FCP is taken as 4000 psi by default and EC0 is taken as 0.002. The concrete stress-strain properties also may be defined by using the CURVE option to specify the name of a custom piece-wise linear stress-strain curve that was previously defined and stored using the STORE STRESS STRAIN CURVE command (see Section 2.5.2.2.1 below). When defining the stress-strain curve to be used with CURVE option, remember that compression stress-strain behavior is defined with negative stress and strain values.

The enhanced concrete compressive behavior due to confinement is tied to the specification of a value for FYS and HOOP/TIES bar data. If these data are not specified, unconfined concrete behavior is assumed. FYS is taken as 0.0 by default. The effects of confinement are not automatically considered when the CURVE option is used to define the concrete stress-strain properties.

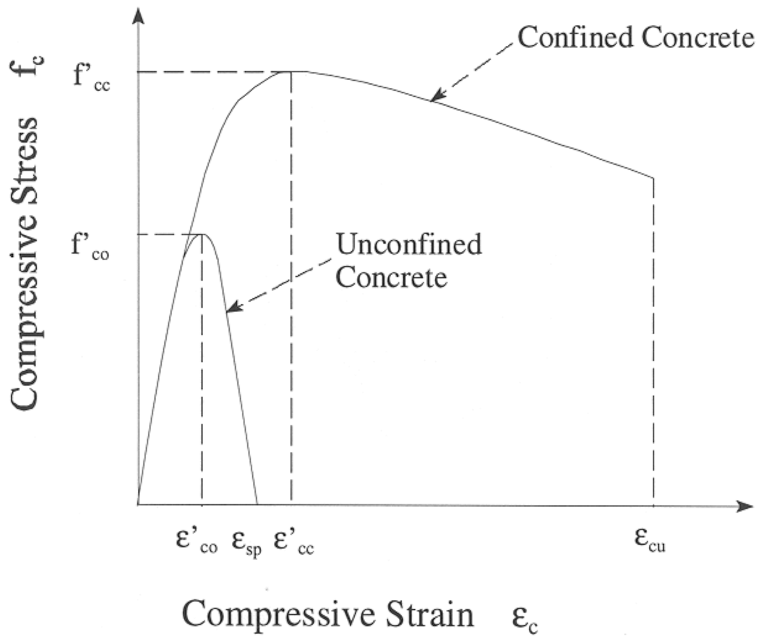


Figure 2.5.2-3 Confined Concrete Compression Stress-Strain Model

The optional BARS specification is used to select the table from which reinforcing bar properties are taken, where ASTM is the default table which contains the standard USA bar sizes. CANADIAN, UNESCO, and KOREAN bar tables also may be selected.

The BARS specification is followed by the START/END specification which refers to the member START and END plastic hinges to which the subsequent bar size and location data apply. The word START/END is followed by TOP, BOTTOM, and SIDE bar data for rectangular plastic hinges, CIRCULAR bar data for circular plastic hinges, and transverse HOOP/TIE bar data for rectangular and circular plastic hinges. The TOP/CIRCULAR, BOTTOM, and SIDE bar data consist of two values: the first value is the integer number of equally-spaced bars while the second value is the integer size of a single bar from the specified table. Figure 2.5.2-4 shows the arrangement of bars as defined by these data values for both rectangular and circular plastic hinge cross sections.

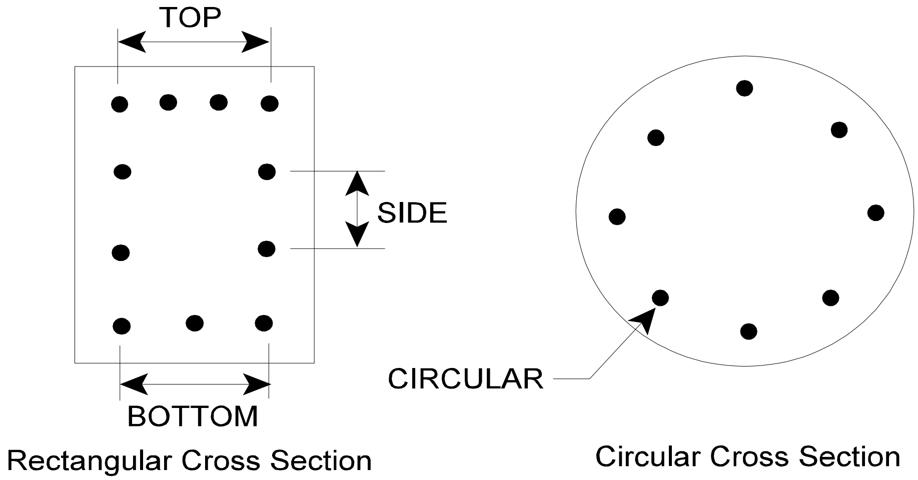


Figure 2.5.2-4 TOP, BOTTOM, SIDE and CIRCULAR BAR Data for Rectangular and Circular Plastic Hinge Cross Sections

The HOOP and TIES data define the existence of secondary reinforcing that is used for the computation of the effects of confinement. The HOOP data applies to circular cross sections and consists also of two values. The first value is the integer bar size and the second is the decimal on-center spacing, or in the case of spiral reinforcing, the spiral pitch. Hoop and spiral reinforcing is treated identically in this context. TIES data applies to plastic hinges with rectangular cross sections and consists of four values. The first value is the integer tie bar size. The second and third values are the integer numbers of tie legs parallel to the local member/plastic hinge y and z axes respectively. And the fourth value is the decimal on-center spacing between tie leg groups. If HOOP/TIES bar data are not specified, then the effect of confinement on concrete compressive stress is not considered. Note that HOOP/TIES bar data plus a non-zero value for FYS must be present in order for the effect of confinement on compressive stress to be considered.

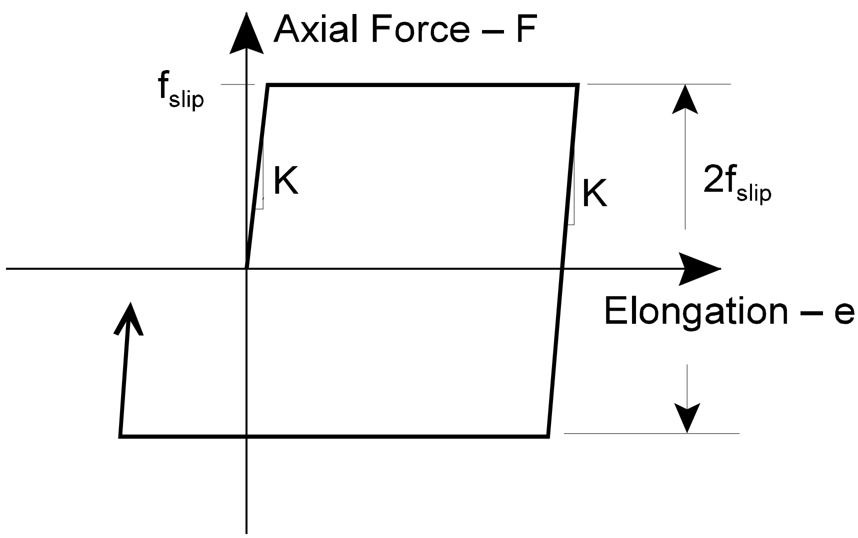
COVER is the final reinforcing bar data item and specifies the decimal clear cover.

If R-C plastic hinges are specified for both the start and end of a member, then the START and END options both must be specified, followed by the Bar Specs data, even if the Bar Specs data are identical for both ends of the member. If a R-C plastic hinge is defined only at the start or end of a member, then the Bar Specs data need be specified only for that end.

If Bar Specs data are not specified, then the effect of reinforcing steel on plastic hinge stiffness and strength is neglected.

Fiction Damper Specs

The Friction Damper Specs are used to specify friction damper behavior for a plane or space truss element. Friction damper behavior is characterized as bi-linear, hysteretic, maximum tension and compression axial force-elongation behavior as illustrated in Figure 2.5.2-5 below.



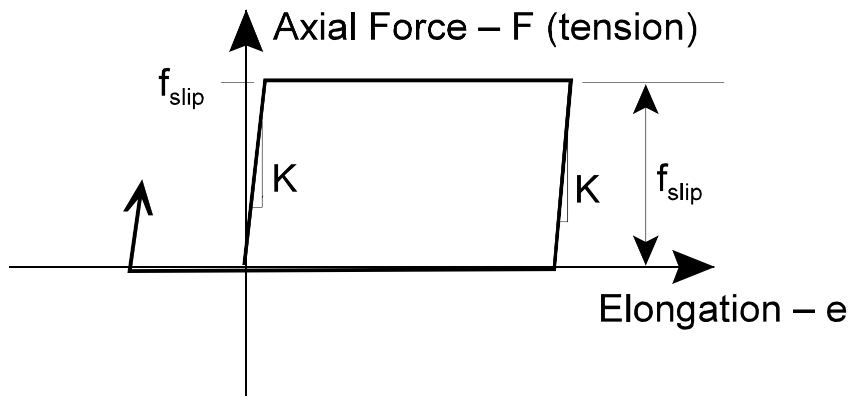
$$-f_{slip} \leq F = AE/Le \leq f_{slip}$$

A = Cross-section area of truss member,
E = Young's modulus,
L = Length of truss member.

Figure 2.5.2-5 Friction Damper Axial Force-Elongation Behavior

The truss member tension or compression force is computed as AE/Le up to a maximum of $\pm f_{slip}$. Loading-unloading cycles are hysteretic.

Nonlinear GEOMETRY is also in effect automatically for the friction damper effect. The FRICTION DAMPER effect may not be used in conjunction with the PLASTIC HINGE/SEGMENT or NLS CONNECTION effect. However, the TENSION/COMPRESSION ONLY and the FRICTION DAMPER effects may be used together. The FRICTION DAMPER EFFECT in conjunction with the TENSION ONLY effect is illustrated in Figure 2.5.2-6.



$0 \leq F = AE/Le \leq f_{slip}$

A = Cross-section area of truss member,
E = Young's modulus,
L = Length of truss member.

Figure 2.5.2-6 Friction Damper-Tension only Axial Force-Elongation Behavior

The FRICTION DAMPER effect is valid for both static and dynamic nonlinear analysis.

Modifications:

The NONLINEAR EFFECTS command functions the same in the ADDITIONS and the CHANGES mode. In the DELETIONS mode, the member $list_{DEL}$, is all that is specified. In the DELETIONS mode, all previously specified nonlinear effects are deleted for the members in $list_{DEL}$. The tabular nonlinear member specs should not be given in the DELETIONS mode.

If existing nonlinear effects data are to be modified, then the complete data description must be specified for the members of interest if the changes are to be properly effected. Any data items not specified revert to default values.

Examples:

The following examples illustrate the various options of the NONLINEAR EFFECTS command:

1. In the following sequence of commands, all members are first specified in the ADDITIONS mode as having geometric nonlinear effects. In addition, members 3, 5, 7, 9, and 11 are to be considered as tension only members. Later, all nonlinear effects (geometry and tension only) are deleted for members 3, 5, 7, 9, and 11. Finally, in the CHANGES or ADDITIONS mode, compression only effects are specified for members 3, 5, 7, 9, and 11.

```
NONLINEAR EFFECTS
GEOMETRY ALL MEMBERS
TENSION ONLY MEMBERS 3 5 7 9 11
        •
        •
        •
DELETIONS
NONLINEAR EFFECTS MEMBERS 3 TO 11 BY 2
        •
        •
        •
CHANGES (or ADDITIONS)
NONLINEAR EFFECTS
COMPRESSION ONLY MEMBERS 3 TO 11 BY 2
        •
        •
        •
```

2. In the next example, the NONLINEAR EFFECTS command specifies nonlinear geometry and plastic hinge data for steel wide flange members:

```
MATERIAL STEEL
MEMBER PROPERTIES
```

```
        1 2 TABLE 'M/S/HP9 '  'HP14X73'
UNITS INCHES KIPS
NONLINEAR EFFECTS
GEOMETRY MEMBERS 1 2
PLASTIC HINGE  -
        FIBER GEOMETRY NTF 1 NTW 1 NBF 8 ND 8 LH 3.0 -
        STEEL FY 50.0 ALPHA 0.5 MEMBER 1 2
```

Under the NONLINEAR EFFECTS command, both geometry and plastic hinge nonlinearities are specified for members 1 and 2. Plastic hinges are assumed to form at both the start and end of the members, and because only STEEL data items are specified, the plastic hinges are assumed to be steel only and must have a wide flange, tee, channel, structural tube or pipe shape as specified by the MEMBER PROPERTIES command. In this case the previous MEMBER PROPERTIES command for members 1 and 2 specifies that the properties are from the HP14X73 profile of the M/S/HP9 table. The plastic hinges therefore have the wide flange shape and cross section dimensions of this profile. The flanges have 1 fiber division (NTF 1) through the thickness and eight fiber divisions (NBF 8) across the width. The web has 1 fiber division (NTW 1) across the thickness and eight fiber divisions (ND 8) through the depth. The hinge length is 3.0 inches. The steel has a yield stress of 50.0 ksi and the hinge has a residual stress distribution with a maximum stress in compression and tension of 25.0 ksi [ALPHA(FY) = .5FY]. Other steel data item default values come from Table 2.5.2-2 above.

3. The following example illustrates the NONLINEAR EFFECTS command for R-C plastic hinges.

```
UNITS  INCH  LBS   DEG   FAH
MEMBER DIMENSIONS RECT B 24.0 H 40.0
          1 2
UNITS INCHES KIPS
NONLINEAR EFFECTS
PLASTIC HINGE START END FIBER GEOM NB 10 NH 20 LH 2.0 -
        STEEL FY 60.0 R-C RECT B 24.0 H 40.0 FCP 4.0 -
        BARS ASTM -
        START BOTTOM 10 9 TOP 10 9 TIES 3 2 2 2.0 COV 4.061 -
        END     BOTTOM 10 9 TOP 10 9 TIES 3 2 2 2.0 COV 4.061 -
        MEMBER 1 2
```

The plastic hinges are defined for both the start and end of members 1 and 2. Note that the same Bar Specs data are given for both the START and END plastic hinges. Note also that although the TIES bar data are given, the FYS value is not specified, implying that secondary reinforcing will not produce confinement effects in the concrete compressive stress.

4. The next sequence of commands illustrates the specification of NLS connections:

```
UNITS INCHES KIPS INCHES DEGREES
NONLINEAR SPRING PROPERTIES
    CURVE  'MPy'  MOMENT SYMMETRIC
           0.0      0.0  -
        4080.0      7.8
    PRINT ALL
END

NONLINEAR EFFECTS
    GEOMETRY MEMBERS 1 4
    NLS CONNECTION MEMBER 2 START MOMENT Y 'MPy'
    NLS CONNECTION MEMBER 3 END   MOMENT Y 'MPy'
```

The NONLINEAR SPRING PROPERTIES command (Section 2.5.3.3 of Volume3 of the User Reference Manual) is used first to define the moment-rotation characteristics of curve MPy. The NLS CONNECTION options of the NONLINEAR EFFECTS command are then used to define the same NLS moment y connection at the start of member 2 and the end of member 3.

5. The final sequence of commands illustrates the specification of friction damper nonlinearity.

```
UNITS INCHES KIPS
NONLINEAR EFFECTS
    FRICTION DAMPER FSLIP 30.0 MEMBERS 256 TO 260
    TENSION ONLY MEMBERS 256 TO 260
```

This example of the NONLINEAR EFFECTS command defines the nonlinearity of members 256 to 260 to be a combination of the friction damper type, with a slip force of 30.0 kips, and tension only. These members must be plane or space truss members and they will exhibit friction damper nonlinear behavior, but only when the member axial forces are tension.

2.5.2.2 Piece-wise Linear Stress-Strain Curves

Sections 2.5.2.2.1-3 below describe the STORE STRESS STRAIN CURVE, DELETE STRESS STRAIN CURVE, and PRINT STRESS STRAIN CURVE commands. These commands are used to define and manage custom, piece-wise linear stress-strain curves that can be use to describe the steel and concrete material properties for plastic hinges.

2.5.2.2.1 STORE STRESS STRAIN CURVE Command

General form:

$$\underline{\mathrm{STO}}\mathrm{RE}\ \ \underline{\mathrm{STRE}}\mathrm{SS}\ \ (\underline{\mathrm{STRA}}\mathrm{IN})\ \ \underline{\mathrm{CUR}}\mathrm{VE}\ \left\{\begin{matrix} i_{curve} \\ 'a_{curve}' \end{matrix}\right\}\ \ (\underline{\mathrm{SYM}}\mathrm{METRIC})$$

$$v_{stress}\quad v_{strain}\quad v_{stress}\quad v_{strain}\ldots$$

$$v_{stress}\quad v_{strain}\quad v_{stress}\quad v_{strain}\ldots$$

$$\ldots$$

$$\underline{\mathrm{END}}$$

Elements:

i_{curve} = unique integer name for the material stress-strain curve. The name may have up to eight digits and must be unique among all currently stored stress-strain curves.

a_{curve} = unique alpha-numeric name for the material stress-strain curve. The name may have up to eight characters and must be unique among all currently stored stress-strain curves.

v_{stress} = the stress value for a specified stress-strain curve point in active force and length units. Tension is positive.

v_{strain} = the corresponding strain value for a specified stress-strain point. Tension is positive.

Explanation:

The STORE STRESS STRAIN CURVE command is used to input the stress and strain data points for the piece-wise linear stress-strain curves that may be used to define the material stress-strain properties of plastic hinges using the STEEL CURVE and RC CURVE options of the NONLINEAR EFFECTS/PLASTIC HINGE/SEGMENT command described in Section 2.5.2.1 (vol. 3, GTStrudl Reference Manual). A curve may define any tension and compression stress-strain properties for a material, including tension-only, compression-only, and symmetric tension and compression.

The following rules apply when specifying the stress-strain data point:

1 All data point sets must begin with a 0-strain data point. The corresponding stress value need not be 0.
2. If both tension and compression points are specified, the tension points shall be specified first, followed by the compression points. Both the tension and the compression point sets shall begin with a 0-strain data point.
3. The absolute value of the strain values shall be specified in ascending order in both the tension and compression point sets.
4. If the SYMMETRIC option is given, then only the tension points or the compression points shall be specified. The appropriate set of symmetric compression or tension points will be computed and stored automatically.
5. The data point values are specified in pairs: stress first, followed by strain. Any number of pairs may be given on any one input line.
6. Any number of tension and compression data points may be specified.

The END command is used to terminate the lines of point data input.

Modifications:

The STORE STRESS STRAIN CURVE commands operates identically in both ADDITIONS and CHANGES modes, with the exception of the following caviats:

1. Under ADDITIONS mode, all curve names are assumed to be new and refer to new data. If an existing curve name is given, an error condition is generated.
2. New curves having new names may be defined under CHANGES mode. If an existing curve name is given under CHANGES mode, the stress-strain data currently stored for that curve are replaced by the newly specified data.

Under DELETIONS mode, the STORE STRESS STRAIN CURVE command is ignored and a message so indicating is printed. The DELETE STRESS STRAIN CURVE command, described in Section 2.5.2.3 below, shall be used to delete existing stress-strain curves.

Examples:

The following example illustrates the STORE STRESS STRAIN command input for a compressive stress-strain model for confined concrete:

```
UNITS INCHES KIPS
STORE STRESS-STRAIN CURVE ‘CCONC1'
      0.01    0.0
     -3.603 -0.0009964
     -5.593 -0.0019944
     -6.566 -0.0029935
     -7.002 -0.0039931
     -7.16  -0.0049929
     -7.173 -0.0059929
     -7.11  -0.006993
     -7.007 -0.007993
     -6.885 -0.0089931
     -6.754 -0.0099933
     -6.621 -0.0109934
     -6.49  -0.0119935
     -6.362 -0.0129936
     -6.239 -0.0139938
     -6.121 -0.0149939
     -6.008 -0.015994
     -5.9   -0.0169941
END
```

This stress-strain curve is illustrated in Figure 2.5.2.2 below. The absolute value of the stress and strain data point values are shown.

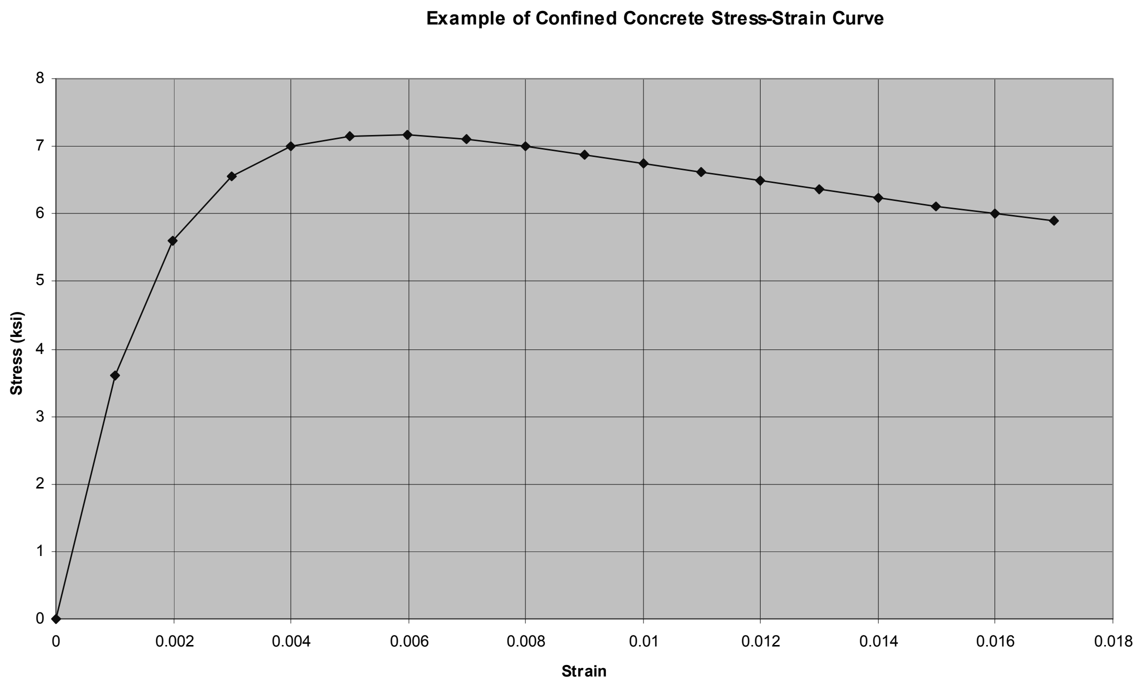


Figure 2.5.2. Illustration of STORE STRESS-STRAIN CURVE Input Data

This following example illustrates the STORE STRESS STRAIN CURVE command under CHANGES mode:

```
UNITS INCHES KIPS
CHANGES
STORE STRESS-STRAIN CURVE 'CCONC1'
   0.0  0.0
  -7.0 -0.002
  -7.0 -0.02
END
ADDITIONS
```

The STORE STRESS STRAIN CURVE command above names the previously stored stress-strain curve CCONC1 under CHANGES mode. The previously stored data points for curve CCONC1 are thus replaced with the three new points that define a compression-only, bi-linear curve.

Error and Warning Conditions:

The following error and warning messages indicate exception conditions that can be encountered during the execution of the STORE STRESS STRAIN CURVE command:

1. Previously specified curve name in ADDITIONS mode

```
{    24} > STORE STRESS-STRAIN CURVE 'Test2' SYMMETRIC
{    25} >   0.0 0.0

**** WARNING_STRSSC -- Stress-strain curve name Test2    already used.
                       Choose a different, unique name.

{    26} >   -60.0 -0.00207
CI-w-cmdmiss, ERROR: Command name missing from line.
{    27} >   -61.0 -0.09
CI-w-cmdmiss, ERROR: Command name missing from line.
{    28} > END

**** END_INFO - END is not recognized in this context.
                Use the FINISH command to leave GTSTRUDL.
```

This output from a GTStrudl execution illustrated what happens when the STORE STRESS STRAIN CURVE command is given in additions mode and the specified curve name has previously been stored.

2. No curve name given

```
{    29} > STORE STRESS-STRAIN CURVE
CI-w-rqcdmis, ERROR: A required condition is unsatisfied.
{    30} >   0.0 0.0

**** WARNING_STRSSC -- No valid stress-strain curve name given.
                       Command ignored.

CI-w-errncmd, ERROR: The previous command completed with one or more
errors.
{    31} >   -60.0 -0.00207
CI-w-cmdmiss, ERROR: Command name missing from line.
{    32} >   -61.0 -0.09
CI-w-cmdmiss, ERROR: Command name missing from line.
{    33} > END

**** END_INFO - END is not recognized in this context.
                Use the FINISH command to leave GTSTRUDL.
```

This GTStudl output illustrates the warning condition caused when a curve name is not given in the STORE STRESS STRAIN CURVE command header.

3. STORE STRESS STRAIN CURVE command under DELETIONS mode

```
{    34} > DELETIONS
{    35} > STORE STRESS-STRAIN CURVE 'Test2' SYMMETRIC
{    36} > ADDITIONS

**** WARNING_STRSSC -- DELETIONS mode, command ignored.
```

This GTStudl output illustrates the warning condition is caused when the STORE STRESS STRAIN CURVE command is given under DELETIONS mode.

2.5.2.2.2 DELETE STRESS STRAIN CURVE Command

General form:

$$\underline{\text{DEL}}\text{ETE } \underline{\text{STRE}}\text{SS } (\underline{\text{STRAI}}\text{N}) \ \underline{\text{CUR}}\text{VE} \begin{Bmatrix} i_{curv} \\ 'a_{curv}' \end{Bmatrix}$$

Elements:

i_{curve} = integer name of a previously stored stress-strain curve.
a_{curve} = alpha-numeric name of a previously stored stress-strain curve.

Explanation:

Give this command to delete previously stored stress-strain curves. It is not necessary to be under DELETIONS mode when this command is used. If the specified curve name does not exist, a warning message is issued.

Error and Warning Conditions:

1. Successful deletion

```
{    47} > DELETE STRESS-STRAIN CURVES 'Test1'

   **** WARNING_STDSSC -- Stress-strain curve Test1    deleted!
```

This warning message is printed when the specified stress-strain curve is found and successfully deleted.

2. Specified stress-strain curve not found

```
{    47} > DELETE STRESS-STRAIN CURVES 2

   **** WARNING_STPSSC -- Data for stress-strain curve 2        not found!
```

This warning message is printed when the specified stress-strain curve could not be found. The curve had not been previously stored or had been previously deleted.

2.5.2.2.3 PRINT STRESS STRAIN CURVE Command

General form:

$$\underline{\text{PRI}}\text{NT } \underline{\text{STRE}}\text{SS } (\underline{\text{STRA}}\text{IN}) \ \underline{\text{CUR}}\text{VE} \begin{Bmatrix} \underline{\text{ALL}} \\ \text{list}_{\text{curve}} \end{Bmatrix}$$

Elements:

$\text{list}_{\text{curve}}$ = list of previously stored stress-strain curves.

Explanation:

This command is used to print the stored stress-strain data points for the specified stress-strain curves. ALL causes the data to be printed for all stored stress-strain curves. The $\text{list}_{\text{curve}}$ option can be used to print the data for a selected subset of all store curves. No data is printed for a specified curve that does not exist.

Example:

Below is an example of the printed output from the PRINT STRESS STRAIN CURVE command for a curve named ‘Test1':

```
{   44} > PRINT STRESS-STRAIN CURVE 'Test1'

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

JOB ID - NLXXX        JOB TITLE - ST4544 -- STORE STRESS-STRAIN CURVE data

ACTIVE UNITS -  LENGTH        WEIGHT        ANGLE       TEMPERATURE      TIME
                  INCH          LB            RAD           DEGF          SEC


------------------------------
Stress-strain Curve: Test1
------------------------------

     Stress        Strain           Stress        Strain           Stress        Strain

   0.000000E+00  0.00000E+00       44000.0      0.15000E-02       45000.0      0.90000E-01
```

5.3.10 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:

ELEMENT PROPERTIES
list (NLS4PH specs)
.
.
.
list (NLS4PH specs)

where,

NLS4PH specs = TYPE '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.

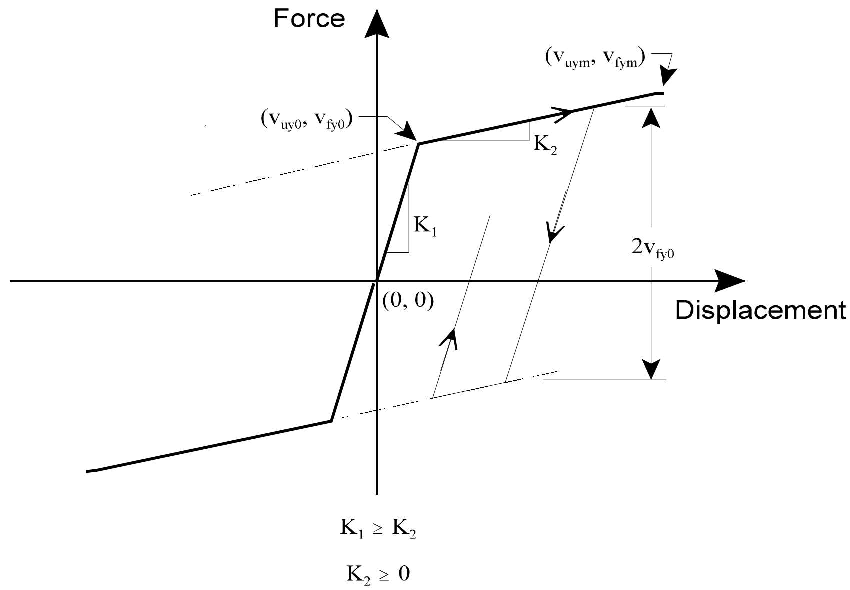


Figure 2.5.3.5-1 NLS4PH Force-Displacement Spring Curve with Hysteretic Behavior

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.

5.3.11 Nonlinear Analysis Output Commands

New nonlinear PRINT and LIST commands have been implemented as described below. The section is numbered as it will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.5.7 Nonlinear Analysis Output Commands

This section describes the GTSTRUDL commands that apply specifically to data and results related to nonlinear analyses. The nonlinear analysis output commands include the PRINT NONLINEAR EFFECTS, LIST PLASTIC HINGE DISPLACEMENTS, and the LIST PLASTIC HINGE STATUS commands which are described in the following sections.

2.5.7.1 The PRINT NONLINEAR EFFECTS Command

General form:

$$\underline{\text{PRI}}\text{NT } \underline{\text{NONL}}\text{INEAR } \underline{\text{EFF}}\text{ECTS } (\underline{\text{MEM}}\text{BERS } \left\{ \begin{matrix} \underline{\text{ALL}} \\ \text{list} \end{matrix} \right\})$$

Elements:

list = optional list of members.

Explanation:

The PRINT NONLINEAR EFFECTS command prints a list of members with corresponding nonlinearity for verification purposes. Nonlinear behavior which is identified in the printed output includes tension/compression only, nonlinear geometry, nonlinear member end connections, and plastic hinges for truss and frame members. Cable elements and nonlinear spring elements are also identified.

The nonlinear effects data are printed for all members and elements named in the member list. Members having no nonlinear effects are not printed. If the member list is not given, then the output is produced for all members, active and inactive.

Example:

An example of the output produced by the PRINT NONLINEAR EFFECTS command is shown in Figure 2.5.7-1 below.

```
{   60} > PRINT NONLINEAR EFFECTS

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


Nonlinear Effects
=================

   Member             Active Nonlinear Effects
   ------             ------------------------

  1                    Nonlinear Geometry
  2                    Nonlinear Geometry
                       Plastic Hinge
  3                    Nonlinear Geometry


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

Figure 2.5.7-1 Example Output from the PRINT NONLINEAR EFFECTS Command

2.5.7.2 The LIST PLASTIC HINGE DISPLACEMENTS Command

General form:

$$\underline{\text{LIST}}\ \underline{\text{PLA}}\text{STIC}\ (\underline{\text{HIN}}\text{GE})\ \underline{\text{DIS}}\text{PLACEMENTS}\ \left(\left\{\begin{array}{l}\underline{\text{ALL}}\ \ (\underline{\text{MEM}}\text{BERS})\\ \underline{\text{MEM}}\text{BERS}\ \ \text{list}\end{array}\right\}\right)$$

Elements:

list = list of member identifiers.

Explanation:

The LIST PLASTIC HINGE DISPLACEMENTS command prints a list of plastic hinge/segment and NLS connection displacements at the start and end of all members for which plastic hinge/segment and NLS connection nonlinearities have been specified and for which nonlinear analysis (including pushover analysis) results are available. Results are limited to the loading conditions identified by the latest LOAD LIST command. The relevant displacements for plastic hinges include axial displacement, bending about the plastic hinge y axis, and bending about the plastic hinge z axis. The relevant displacements for plastic segments include all six translations and rotations. The relevant displacements for NLS connections include any of the three displacement and three rotation connection degrees-of-freedom for which a NLS connection spring has been specified. Plastic hinge displacements are relative displacements defined with respect to the local coordinate system of the member as U_{end} - U_{start}, where U_{end} represents the displacements at the end node of the plastic hinge/segment/NLS connection, and U_{start} represents the displacements at the plastic hinge/segment/NLS connection start node.

The plastic hinge displacements are printed for all members named in the member list, and for which plastic hinge displacement results exist. If a member list is not given, then the output is produced for all plastic hinge/segment and NLS connection members, active and inactive.

Errors:

The following informational message is printed if no plastic hinge/segment displacement results exist. This will be the case if no nonlinear analysis has been performed or no members with plastic hinge/segment and/or NLS connection nonlinearity exist.

**** INFO_STLPHD – Plastic hinge displacement results do not exist. Command ignored.

Example:

Figure 2.5.7-2 below gives a sample of the output from the LIST PLASTIC HINGE DISPLACEMENTS command.

```
{   92} > LIST PLASTIC HINGE DISPL

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


ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
    LENGTH         WEIGHT         ANGLE          TEMPERATURE         TIME
     INCH           KIP            DEG               DEGF            SEC


Plastic Hinge Displacements
===========================

                                                          Plastic Hinge Displacements Start/End
 Member       Load                          TX            TY            TZ            RX            RY            RZ
 ------       ----                   ------------------------------------------------------------------------------

 1         PA2__001        Start      -0.0015698                                                 0.0000000   -0.0004978
                           End        -0.0015683                                                 0.0000000   -0.0002487
 2         PA2__001        Start      -0.0015683                                                 0.0000000   -0.0002487
                           End        -0.0015677                                                 0.0000000    0.0000000
 1         PA2__002        Start      -0.0015762                                                 0.0000000   -0.0009985
                           End        -0.0015699                                                 0.0000000   -0.0004978
 2         PA2__002        Start      -0.0015699                                                 0.0000000   -0.0004978
                           End        -0.0015678                                                 0.0000000    0.0000000
```

Figure 2.5.7-2 Example of Output from the LIST PLASTIC HINGE DISPLACEMENTS Command

2.5.7.3 The LIST PLASTIC HINGE STRESSES Command

General form:

$$\underline{\text{LIST}}\ \underline{\text{PLA}}\text{STIC}\ (\underline{\text{HIN}}\text{GE})\ \underline{\text{STR}}\text{ESSES} \left\{ \begin{array}{l} \left\{ \begin{array}{l} \underline{\text{MAX}}\text{IMUM} \\ \underline{\text{MIN}}\text{IMUM} \end{array} \right\} \\ \underline{\text{FIB}}\text{ER}\ \ i_f \end{array} \right\} \ \left(\left\{ \begin{array}{l} \underline{\text{STE}}\text{EL} \\ \underline{\text{CON}}\text{CRETE} \end{array} \right\} \right)\ -$$

$$(\underline{\text{MEM}}\text{BERS} \left\{ \begin{array}{l} \rightarrow \underline{\text{ALL}} \\ \text{list} \end{array} \right\})$$

Elements:

i_f = integer number of the fiber for which plastic hinge stress results are to be printed.

list = optional list of member names for which the specified plastic hinge stress results are to be printed.

Explanation:

The LIST PLASTIC HINGE STATUS command prints a table of plastic hinge fiber normal stress results for the plastic hinges at the start and/or end of the specified members and for the active loading conditions. If a list of members, or the MEMBERS ALL option, is not specified, then MEMBERS ALL is assumed and the plastic hinge stress results are printed for all active and inactive members having plastic hinge/segment nonlinearity.

The command provides the special output control options MAXIMUM/-MINIMUM, FIBER, and STEEL/CONCRETE. The MAXIMUM/MINIMUM option indicates that only the stress results data for the single fiber having the MAXIMUM (largest positive) or MINIMUM (largest negative) stress are to be printed. The FIBER i_f option indicates that only the stress results data for single fiber number i_f are to be printed. The STEEL/CONCRETE option is used to select either steel or concrete fibers for which the stress results are to be printed. If STEEL or CONCRETE is not selected, then both STEEL and CONCRETE fibers are processed for the printed stress output.

Example:

The Figures 2.5.7-3 through 2.5.7-5, over the next few pages, illustrate the plastic hinge stress output for several of the options provided by the command. The printed output contains the fiber normal stress and strain for each fiber printed, plus the fiber number, fiber cross-section local y and z coordinates, and fiber area. The y and z coordinate values are referenced to the centroid of the cross-section.

1. LIST PLASTIC HINGE STRESSES STEEL MEMBER 13

```
{   156} > LIST PLASTIC HINGE STRESSES STEEL MEMBER 13

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


ACTIVE UNITS   (UNLESS INDICATED OTHERWISE):
   LENGTH          WEIGHT     ANDLE        TEMPERATURE      TIME
     INCH            KIP       DEG            DEGF          SEC


Plastic Hinge Stresses/Strains, Load = PA2_024
==========================================================================================

Member    Start/End    Fiber    Stress     Strain         Matr1        Y        Z        Ax
------    ---------    -----    ------     ------         -----      -----   ------    -----

13        End          1281      82.426     0.0093331     Steel      0.000  -29.278    2.250
                       1282      81.965     0.0064855     Steel      4.580  -28.918    2.250
                       1283      79.214     0.0037080     Steel      9.048  -27.846    2.250
                       1284      31.004     0.0010691     Steel     13.292  -26.087    2.250
                       1285     -39.624    -0.0013663     Steel     17.209  -23.687    2.250
                       1286     -75.009    -0.0035383     Steel     20.703  -20.703    2.250
                       1287     -81.773    -0.0053934     Steel     23.687  -17.209    2.250
                       1288     -82.032    -0.0068859     Steel     26.087  -13.292    2.250
                       1289     -82.210    -0.0079789     Steel     27.846   -9.048    2.250
                       1290     -82.316    -0.0086457     Steel     28.918   -4.580    2.250
                       1291     -82.352    -0.0088699     Steel     29.278    0.000    2.250
                       1292     -82.316    -0.0086457     Steel     28.918    4.580    2.250
                       1293     -82.210    -0.0079789     Steel     27.846    9.048    2.250
                       1294     -82.032    -0.0068859     Steel     26.087   13.292    2.250
```

Figure 2.5.7-3 Example 1 of Output from the LIST PLASTIC HINGE STRESSES Command

The plastic hinge fiber normal stress and strain are printed for all STEEL fibers only, for all active loads, and for member 13 only.

2. LIST PLASTIC HINGE STRESSES FIBER 1311 MEMBER 13

```
{ 154} > LOAD LIST 'PA2_024'
{ 155} > LIST PLASTIC HINGE STRESSES FIBER 1311 MEMBER 13

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

ACTIVE UNITS      (UNLESS INDICATED OTHERWISE):
 LENGTH   WEIGHT   ANGLE        TEMPERATURE             TIME
 INCH      KIP      DEG            DEGF                  SEC

Plastic Hinge Stresses/Strains, Load = PA2_024
==============================================

Member    Start/End    Fiber   Stress      Strain      Matr1        Y         Z          Ax
------    ---------    -----   --------    ------      -----       -----     -----      -----
13        End          1311    85.502      0.0290127   Steel      -29.278   0.000      2.250
```

Figure 2.5.7-4 Example 2 of Output from the LIST PLASTIC HINGE STRESSES Command

Only the plastic hinge stress results for fiber 1311, member 13 are printed. Fiber 1311 is a steel fiber. The active loading conditions are PA2_024.

3. LIST PLASTIC HINGE STRESSES MIN CONCRETE MEMBER 13

```
{ 151} > LOAD LIST 'PA2_024'   'PA2_025'
{ 152} > LIST PLASTIC HINGE STRESSES MIN CONCRETE MEMBER 13

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


ACTIVE UNITS      (UNLESS INDICATED OTHERWISE):
 LENGTH          WEIGHT              ANGLE          TEMPERATURE             TIME
 INCH              KIP                DEG           DEGF                     SEC

Plastic Hinge Stresses/Strains, Load = PA2_024
==============================================

Member    Start/End    Fiber   Stress      Strain        Matr1       Y          Z         Ax
------    ---------    -----   --------    ------        -----     -----      -----      ----
13        End          1094    -5.810      -0.0062491    Conc      25.063     -13.397    4.611


Plastic Hinge Stresses/Strains, Load = PA2_025
==============================================

Member    Start/End    Fiber   Stress      Strain        Matr1     Y           Z          Ax
------    ---------    -----   --------    ------        -----     -----      -----      ----
13        End           999    -5.810      -0.0063884    Conc      24.831     -7.532     4.210
```

Figure 2.5.7-5 Example 2 of Output from the LIST PLASTIC HINGE STRESSES Command

This command requests the MINIMUM or smallest negative stress from all CONCRETE fibers in member 13. The active loading conditions are PA2_024 and PA2_025.

2.5.7.4 The LIST PLASTIC HINGE STATUS Command

General form:

$$\underline{\text{LIST}}\ \underline{\text{PLA}}\text{STIC}\ (\underline{\text{HIN}}\text{GE})\ \underline{\text{STA}}\text{TUS}\ \left(\underline{\text{MEM}}\text{BERS}\ \left\{\begin{matrix}\underline{\text{ALL}}\\ \text{list}\end{matrix}\right\}\right)$$

Elements:

list = list of member identifiers

Explanation:

The LIST PLASTIC HINGE STATUS command prints a list containing the extent of plastic hinge formation at the start and end of specified members for which plastic hinge/segment nonlinearity has been specified and for all currently active independent loading conditions. The extent of plastic hinge formation is indicated by a percentage of "yielding," which is calculated as 100 × (ratio of total area of plastic hinge fibers "yielded" to total area of plastic hinge fibers). A plastic hinge fiber has yielded if, for steel, the yield strain in the fiber has been exceeded, and for concrete, if the ultimate compressive strain has been exceeded.

The plastic hinge status is printed for all members named in the member list, and for which plastic hinge/segment results exist. If a member list is not given, then the output is produced for all plastic hinge/segment members, active and inactive.

Errors:

The following informational message is printed if no plastic hinge status results exist. This will be the case if no nonlinear analysis has been performed or no members with plastic hinge nonlinearity exist.

****INFO_STLPHD - Plastic hinge status data do not exist. Command ignored.

Example:

Figure 2.5.7-6 below gives a sample of the output from the LIST PLASTIC HINGE STATUS command.

```
{ 205} > LIST PLASTIC HINGE STATUS

 Plastic Hinge Status
 ====================


                             % Plastic Hinge Formation
Member       Load          Member Start      Member End
------       ----          ------------      ----------

2            1                   0                85
```

Figure 2.5.7-6 Example of Output from the LIST PLASTIC HINGE STATUS Command

This page intentionally left blank.

5.3.12 Pushover Analysis

A new pushover analysis feature has been implemented as described in the section below. This section is numbered as it will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.6.6 Pushover Analysis

Pushover analysis is an automated nonlinear incremental load analysis which also contains a procedure that automatically searches for the load level at which structural instability or collapse occurs. The two commands which are used to perform the pushover analysis are the PUSHOVER ANALYSIS DATA command and the PERFORM PUSHOVER ANALYSIS command, described together in Section 2.6.6.1. The PRINT PUSHOVER ANALYSIS DATA command, described in Section 2.6.6.2, is used to verify the parameter values specified by the PUSHOVER ANALYSIS DATA command. Supplementary results output commands include LIST PUSHOVER DUCTILITY RATIO LIST PLASTIC HINGE DUCTILITY RATIO, and LIST PUSHOVER ANALYSIS LIMIT LOAD. The LIST PUSHOVER DUCTILITY RATIO and LIST PLASTIC HINGE DUCTILITY RATIO commands are used to list the ductility ratio results from a pushover analysis and are described in Sections 2.6.6.4. The LIST PUSHOVER ANALYSIS LIMIT LOAD command, described in Section 2.6.6.5, is used to list the intermediate pushover analysis incremental storage load at which a specified plastic hinge limit state strain is first equaled or exceeded for the plastic hinges in each of a specified set of members.

2.6.6.1 The PUSHOVER ANALYSIS DATA and the PERFORM PUSHOVER ANALYSIS Commands

PUSHOVER ANALYSIS DATA -- General form:

$$\underline{\text{PUSH}}\text{OVER } (\underline{\text{ANAL}}\text{YSIS } \underline{\text{DAT}}\text{A})$$

$$\left\{\begin{array}{l}
\underline{\text{INCR}}\text{EMENTAL } (\underline{\text{LOA}}\text{D}) \begin{Bmatrix} i_{IL} \\ 'a_{IL}' \end{Bmatrix} \\
\underline{\text{CONS}}\text{TANT } (\underline{\text{LOA}}\text{D}) \begin{Bmatrix} i_{CL} \\ 'a_{CL}' \end{Bmatrix} \\
\underline{\text{LOA}}\text{DING } (\underline{\text{RAT}}\text{E}) \ v_{LRate} \\
\underline{\text{CONV}}\text{ERGENCE } \underline{\text{RAT}}\text{E} \ v_{CRate} \\
\underline{\text{CONV}}\text{ERGENCE } (\underline{\text{TOL}}\text{ERANCE}) \ \underline{\text{COL}}\text{LAPSE} \ v_{CTol} \\
\underline{\text{MAX}}\text{IMUM } (\underline{\text{NUM}}\text{BER } \underline{\text{OF}} \ \underline{\text{LOA}}\text{D}) \ \underline{\text{TRI}}\text{ALS} \ i_{LT} \\
\underline{\text{MAX}}\text{IMUM } (\underline{\text{NUM}}\text{BER } \underline{\text{OF}} \ \underline{\text{LOA}}\text{D}) \ \underline{\text{INC}}\text{REMENTS} \ i_{LI} \\
\underline{\text{MAX}}\text{IMUM } (\underline{\text{NUM}}\text{BER } \underline{\text{OF}}) \begin{Bmatrix} \underline{\text{CYC}}\text{LES} \\ \underline{\text{ITER}}\text{ATIONS} \end{Bmatrix} i_{CYC} \\
\underline{\text{CONV}}\text{ERGENCE } (\underline{\text{TOL}}\text{ERANCE}) \begin{Bmatrix} \underline{\text{EQUIL}}\text{IBRIUM} \\ \underline{\text{DISPL}}\text{ACEMENT} \end{Bmatrix} i_{ETol}
\end{array}\right\}$$

$$\underline{\text{END}} \ (\underline{\text{PUSH}}\text{OVER } \underline{\text{DAT}}\text{A})$$

PERFORM PUSHOVER ANALYSIS – General form:

$$\underline{\text{PER}}\text{FORM } \underline{\text{PUSH}}\text{OVER } (\underline{\text{ANA}}\text{LYSIS}) \left(\begin{Bmatrix} \underline{\text{NJP}} \ i_{NJP} \\ \underline{\text{WITH}}\text{OUT } \underline{\text{RED}}\text{UCE } (\underline{\text{BAND}}) \end{Bmatrix} \right)$$

Elements:

i_{IL}/a_{IL} = integer or alphanumeric id of the required independent loading condition which is treated as the incremental loading during the pushover analysis procedure.

i_{CL}/a_{CL} = integer or alphanumeric id of the optional independent loading condition which remains constant during the pushover analysis procedure.

v_{LRate} = decimal number specifying the initial scaling factor which is applied to i_{IL}/a_{IL} to create a loading increment.

v_{CRate} = decimal number specifying the scaling factor which is applied to v_{LRate} when convergence has failed or instability has been detected for a particular load increment. The default is 0.25. This value must be positive and less than 1.0.

v_{CTol} = decimal number specifying the convergence tolerance for the collapse load search procedure. The default is 0.01.

i_{LT} = integer number specifying the maximum number of collapse load search trials that may be executed for any one load increment. The default is 10.

i_{LI} = integer number specifying the maximum permissible number of loading increments that may be generated by the pushover analysis procedure. The default is 10.

i_{CYC} = integer number specifying the maximum number of equilibrium correction iterations that may be performed for any one load increment. The default is 1.

v_{ETol} = decimal number specifying the equilibrium or displacement convergence tolerance for the equilibrium correction iterations. The default is 0.01.

i_{NJP} = number of joints per global stiffness sub-matrix partition.

Explanation:

The PUSHOVER ANALYSIS DATA and PERFORM PUSHOVER ANALYSIS commands are used together to perform a pushover analysis. The PUSHOVER ANALYSIS DATA command is used to specify the values for a series of parameters that control the pushover analysis procedure and must be given first. The PERFORM PUSHOVER ANALYSIS command follows and is used to execute the pushover analysis procedure.

The pushover analysis procedure and the control parameters from the PUSHOVER ANALYSIS DATA command are described as follows:

1) The pushover analysis consists of a series of steps, wherein each step, a new total applied load is computed by adding a load increment to the previous applied load total. A nonlinear analysis is subsequently performed for the

new total applied load. The initial loading increment used to compute the sequence of increasing applied loading totals is computed by multiplying the applied load components of loading i_{IL}/a_{IL} by the loading rate factor v_{LRate}.

2) The pushover analysis incremental loading condition (PAIL) is used to store the total applied load and the corresponding nonlinear analysis results for each incremental step of the pushover analysis. The PAIL is automatically created, where its name is constructed by concatenating the following character strings: 'PA' + the first three characters of i_{IL}/a_{IL} + '000'. For example, if i_{IL} = 1, then the name of the pushover analysis incremental loading will be PA1__000. Note that the underscore character "_" is used as the space placeholder.

3) The initial applied load is computed by adding the loading components from the constant load i_{CL}/a_{CL} to the loading components of the initial load increment. The result is stored in the PAIL.

4) A nonlinear analysis is performed using the PAIL as the active independent load. The nonlinear analysis is performed as described in Section 2.5, Volume 3 of the GTSTRUDL User Reference Manual, using the values of i_{CYC} and v_{ETol} as the maximum number of cycles and convergence tolerance control parameters respectively. The following steps are taken, depending on the outcome of the nonlinear analysis:

   A. If the nonlinear analysis converged successfully, both the applied loading components and the results of the analysis – displacements, member and element forces, element stresses, joint loads, and support reactions – are copied into an automatically created intermediate storage loading condition. The name of this load is created like that of the PAIL, with the exception that the "000" portion of the name is replaced by a three-digit incremental loading sequence number, beginning with 001. Note that these storage loadings are fully functional independent loadings, containing both applied loading components plus the corresponding nonlinear analysis results.

   B. If the nonlinear analysis failed to converge, or a structural instability was detected, a new loading rate v_{LRate} is computed as $(v_{LRate})(v_{CRate})$. A new load increment is computed as in Item 1, using the new value of v_{LRate}. The PAIL is then updated by adding the new load increment to the applied loads used for the last successful nonlinear analysis.

This process is performed until the nonlinear analysis converges successfully or the maximum number of collapse load search trials, i_{LT}, is reached, whichever comes first. If the nonlinear analysis converges successfully, the procedure of 4-A is followed. If i_{LT} is reached, the pushover analysis is terminated. If the value of i_{LT} is sufficiently large, 10 or more, this termination may indicate a collapse condition.

5) Following each successfully convergent nonlinear analysis, a total load factor, f_{Load}, is computed as the sum of the successive v_{LRate} values from all previous load increments for which the nonlinear analysis converged successfully. Also following each nonlinear analysis, regardless of convergence success, the collapse load convergence is checked by comparing the current value of v_{LRate} to f_{Load} as follows:

$$v_{LRate}^{Current} \leq v_{CTol}(f_{Load})$$

If the inequality is satisfied, then the pushover analysis is terminated, also indicating the possibility that a collapse condition has been achieved.

6) The pushover analysis is also terminated if the maximum number of loading increments, i_{LI}, is reached before the collapse load convergence criterion of Item 5 is satisfied. In general, a state of collapse is not indicated by this situation.

7) Following the completion of the pushover analysis, all sequentially created intermediate storage loads – as in Item 4-A – are stored in a load group named "IncrLds."

The pushover analysis can be performed for any combination of nonlinear effects including nonlinear geometry, tension/compression only, plastic hinges, nonlinear member end connections, nonlinear spring elements, and cable elements.

Modifications:

The PUSHOVER ANALYSIS DATA command functions the same in ADDITIONS, CHANGES, and DELETIONS modes. Existing data is always replaced with new data.

Successive executions of the PERFORM PUSHOVER ANALYSIS command causes the automatic deletion of the existing PAIL, the intermediate storage loading conditions, and the load group “IncrLds.”.

Errors:

The following error messages indicate error conditions which may be detected when the PUSHOVER ANALYSIS DATA command is executed:

**** ERROR_STPOAD – Incremental load name not specified for pushover analysis.
Loading probably not defined.
Scan mode entered.

This error message is printed if the loading name specified in the INCREMENT LOAD option has not been previously defined.

**** ERROR_STPOAD – Constant load name not specified for pushover analysis.
Loading probably not defined.
Scan mode entered.

This error message is printed if the loading name specified in the CONSTANT LOAD option has not been previously defined.

**** ERROR_STPOAD – Incremental load 1-A is not an independent load.
Scan mode entered.

This error message is printed if the loading named in the INCREMENTAL LOAD option – in this case loading 1-A – is not an independent load, i.e. a loading combination.

*** ERROR_STPOAD – Constant load 4-A is not an independent load.
Scan mode entered.

This error message is printed if the loading named in the CONSTANT LOAD option – in this case loading 4-A – is not an independent load, i.e. a loading combination.

When any of the above error conditions is encountered, scan mode is entered and must be turned off before any analysis can be executed.

The following messages indicate error, warning, or informational conditions which may be detected when the PERFORM PUSHOVER ANALYSIS command is executed.

**** ERROR_STPACP – Loading 1 may not be both the constant load and the incremental load.
Scan mode entered and analysis terminated.

This error message is printed when both the constant load, i_{CL}/a_{CL}, and the incremental load, i_{IL}/a_{IL}, are the same loading conditions. These two loading conditions must have different names.

**** WARNING_STPACP – Convergence problems! Specified loading rate = 300.00000 possibly too large. Analysis terminated.

This message is printed if the nonlinear analysis for the first load increment fails to converge. A strong possible cause of this problem is that the loading rate v_{LRate} is too large, causing too much load to be applied in the first increment. The current value of v_{LRate} is printed and the analysis is terminated.

**** INFO_STPACP – Pushover analysis has not converged after 4 load increments.

This message means that the maximum number of load increments were executed without collapse load convergence. The analysis is terminated and the current load factor is computed. Also, all intermediate incremental storage loads are stored in the load group “IncrLds.” This is an informational message because this may be the desired outcome of the pushover analysis.

**** WARNING_STPACP -- Pushover load increment 24 has not converged after 2 load rate adjustment trials.

The convergence failure of a nonlinear analysis for a particular load increment initiates a series of collapse load rate adjustment trials as described by Item 4-B above. This message indicates that the specified maximum number of these trials, i_{LT}, is not sufficient to find a new load rate that produces a new applied load for which nonlinear analysis converges.

```
**** INFO_STPACP –  Pushover  analysis  has  converged  after  34 load
                    increments.

**** INFO_STPACP –  The current collapse load factor = 231.2000
                    Load  components  and  results  are  stored  in  the
                    following intermediate loads:
                       PA1__001  PA1__002  PA1__003  PA1__004
                       PA1__005  PA1__006  PA1__007  PA1__008
                       PA1__009  PA1__010  PA1__011  PA1__012
                       PA1__013  PA1__014  PA1__015  PA1__016
                       PA1__017  PA1__018  PA1__019  PA1__020
                       PA1__021  PA1__022  PA1__023  PA1__024
                       PA1__025  PA1__026  PA1__027  PA1__028
                       PA1__029  PA1__030  PA1__031  PA1__032
                       PA1__033  PA1__034

**** INFO_STPACP -- The incremental loads above are stored in load group
                    IncrLds .


    /----- Push-over Analysis Load Factor History -----/
         Load Increment                 Load Factor
           --------------                  -----------
            PA1__001                      10.000000
            PA1__002                      20.000000
            PA1__003                      30.000000
            PA1__004                      40.000000
            PA1__005                      50.000000
            PA1__006                      60.000000
            PA1__007                      70.000000
            PA1__008                      80.000000
            PA1__009                      90.000000
            PA1__010                     100.000000
            PA1__011                     110.000000
            PA1__012                     120.000000
            PA1__013                     130.000000
            PA1__014                     140.000000
            PA1__015                     150.000000
            PA1__016                     160.000000
            PA1__017                     170.000000
            PA1__018                     180.000000
            PA1__019                     190.000000
            PA1__020                     200.000000
            PA1__021                     210.000000
            PA1__022                     220.000000
            PA1__023                     230.000000
            PA1__024                     230.399994
            PA1__025                     230.479996
            PA1__026                     230.559998
            PA1__027                     230.639999
            PA1__028                     230.720001
            PA1__029                     230.800003
```

```
              PA1__030                    230.880005
              PA1__031                    230.960007
              PA1__032                    231.040009
              PA1__033                    231.120010
              PA1__034                    231.200012

 **** INFO_STPACP - Time to complete pushover analysis =     9.49 seconds.
```

This series of informational messages is produced at the conclusion of a pushover analysis and lists the intermediate storage loads created to store the incremental load levels and corresponding nonlinear analysis results and the load factor history for each of these intermediate loads. Notice the naming convention for the intermediate loads, constructed from the characters “PA”, the name of the increment load, 1, and the sequence numbers 001 to 034.

The incremental load i_{IL}/a_{IL} when multiplied by the final collapse load factor, in this case 231.2, produces the total load at which structural instability or collapse was finally detected within the collapse load tolerance v_{CTol}. Timing information is also printed.

Example:

The following example PA-1 illustrates a pushover analysis for a simple electric power transmission tower with geometric nonlinearity for all members. This is the only source of nonlinear behavior for the structure. The incremental load for the pushover analysis is the self weight of the structure applied as joint loads in the global -Y direction. A constant load is a one-pound joint load applied in the global X direction at select joints. Figure 2.6.6-1 shows an illustration of the structure created by GTMENU; the joints are labeled. Figure 2.6.6-2 lists the total command input for the structure, including the pushover analysis commands with brief commentary. Figures 2.6.6-3 and 2.6.6-4 show selected output from the pushover analysis.

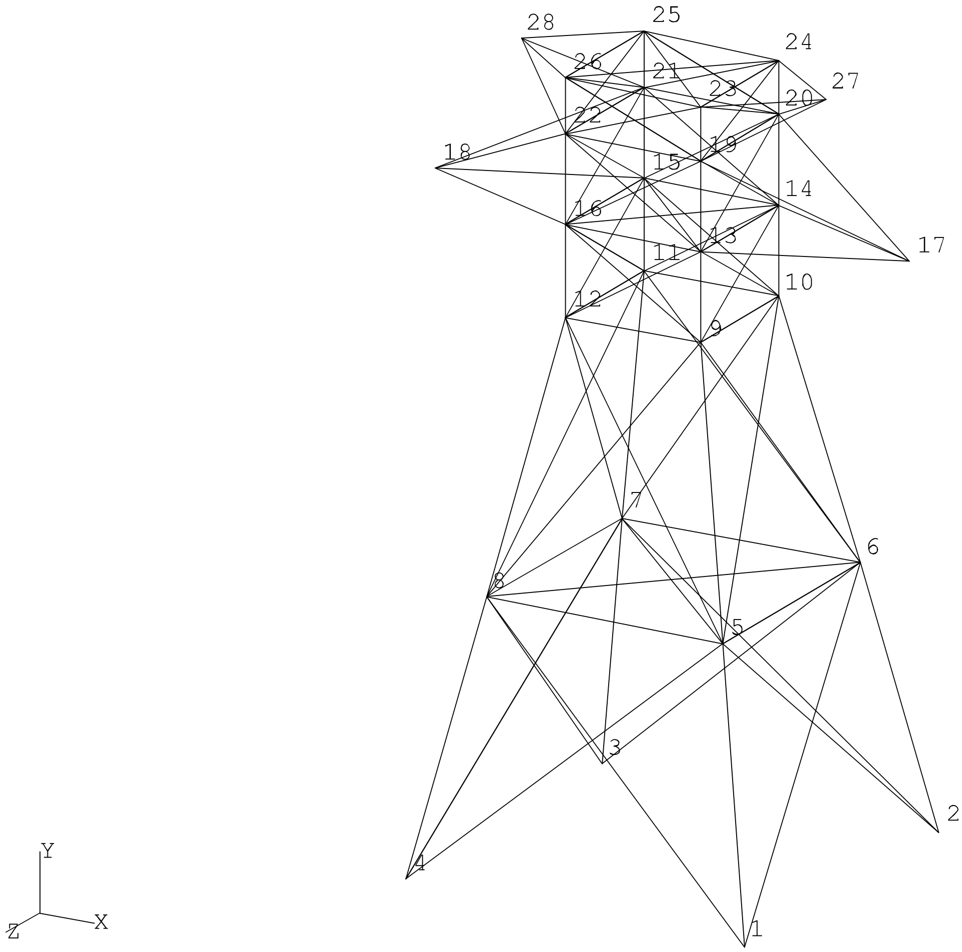


Figure 2.6.6-1 Simple Transmission Tower for Pushover Analysis Example PA-1

```
STRUDL 'PA-1' 'Example of pushover analysis of tower'
$
$  Pushover ANALYSIS of a tower where all members are designated as
$  nonlinear geometric.
$
$* **
$*********************************************************************
$*********************************************************************
$* **

UNITS FEET
$
JOINT COORDINATES
1   10.0   0.0     10.0
2   10.0   0.0     -10.0
3   -10.0  0.0     -10.0
4   -10.0  0.0     10.0
5   7.0    15.0    7.0
6   7.0    15.0    -7.0
7   -7.0   15.0    -7.0
8   -7.0   15.0    7.0
9   4.0    30.0    4.0
10  4.0    30.0    -4.0
11  -4.0   30.0    -4.0
12  -4.0   30.0    4.0
13  4.0    35.0    4.0
14  4.0    35.0    -4.0
15  -4.0   35.0    -4.0
16  -4.0   35.0    4.0
17  14.0   35.0    0.0
18  -14.0  35.0    0.0
19  4.0    40.0    4.0
20  4.0    40.0    -4.0
21  -4.0   40.0    -4.0
22  -4.0   40.0    4.0
23  4.0    43.0    4.0
24  4.0    43.0    -4.0
25  -4.0   43.0    -4.0
26  -4.0   43.0    4.0
27  9.0    43.0    0.0
28  -9.0   43.0    0.0
$
STATUS SUPPORT 1 2 3 4
$
TYPE SPACE FRAME
MEMBER INCIDENCES
1  1 5
2  5 9
3  9 13
4  13 19
5  19 23
6  2 6
7  6 10
8  10 14
```

```
9  14  20
10  20 24
11  3 7
12  7 11
13  11 15
14  15 21
15  21 25
16  4 8
17  8 12
18 12 16
19  16 22
20  22 26

TYPE SPACE TRUSS
MEMBER INCIDENCES
21  5 6
22  6 7
23  7 8
24  8 5
25  9 10
26  10 11
27  11 12
28  12 9
29  13 14
30  14 15
31  15 16
32  16 13
33  19 20
34  20 21
35  21 22
36  22 19
37  23 24
38  24 25
39  25 26
40  26 23
41  13 17
42  14 17
43  15 18
44  16 18
45  19 17
46  20 17
47  21 18
48  22 18
49  23 27
50  24 27
51  25 28
52  26 28
53  19 27
54  20 27
55  21 28
56  22 28
59  1 6
60  2 5
61  2 7
```

```
62   3 6
63   3 8
64   4 7
65   4 5
66   1 8
67   5 10
68   6 9
69   6 11
70   7 10
71   7 12
72   8 11
73   8 9
74   5 12
75   9 14
76   10 13
77   10 15
78   11  14
79   12 15
80   11 16
81   9 16
82   12 13
83   13 20
84   14 19
85   14 21
86   15 20
87   16 21
88   15 22
89   13 22
90   16 19
91   19 24
92   20 23
93   20 25
94   21 24
95   22 25
96   21 26
97   19 26
98   22 23
99   23 25
100   24 26
101   13 15
102   14 16
103   5 7
104   6 8
$
UNITS FEET POUNDS
CONSTANTS
DENSITY 490.0 ALL
UNITS INCHES KIPS
CONSTANTS
E 29000.0 ALL
CTE 0.65E-5
$
MEMBER PROPERTIES
$  LEG ANGLES
```

```
1 TO 20 TABLE 'ANGLES' 'EQ303012'
$ HORIZONTAL GIRTS
21 TO 40 TABLE 'ANGLES' 'EQ303008'
$ ARM MEMBERS
41 TO 44 TABLE 'ANGLES' 'EQ323208'
$ ARM STRAPS
45 TO 48 TABLE 'ANGLES' 'UN131106'
$ STAIC ARMS
49 TO 56 TABLE 'ANGLES' 'EQ303008'
$ DIAGONAL BRACING
59 TO 98 TABLE 'ANGLES' 'EQ202008'
$ HORIZONTAL BRACING
99 TO 104 TABLE 'ANGLES' 'EQ303008'

$
$ Designate nonlinear geometric members
$
NONLINEAR EFFECTS
  GEOMETRY ALL MEMBERS

$
$ Define the pushover analysis constant and incremental
$ loads
$
DEAD LOAD  1  'Incremental load' DIRECTION -Y ALL JOINTS
$
UNITS POUNDS
LOADING  2  'Constant lateral perturbation load'
JOINT LOADS
25 26 FORCE X 1.0
21 22 FORCE X 1.0
15 16 FORCE X 1.0
11 12 FORCE X 1.0
 7  8 FORCE X 1.0
17 18 FORCE X 1.0
27 28 FORCE X 1.0

$
$ Specify pushover analysis data and execute pushover analysis
$
PUSHOVER ANALYSIS DATA
  INCREMENTAL LOAD 1
  CONSTANT LOAD 2
  MAXIMUM NUMBER OF LOAD INCREMENTS 100
  LOADING RATE 1.0
  CONVERGENCE RATE 0.2
  MAXIMUM NUMBER OF TRIALS 20
  MAXIMUM NUMBER OF ITERATIONS 50
  CONVERGENCE TOLERANCE COLLAPSE 0.002
  CONVERGENCE TOLERANCE DISPL 0.0001
END

PRINT PUSHOVER ANALYSIS DATA
```

```
PERFORM PUSHOVER ANALYSIS

$
$ List displacements at joint 16 for the loading history
$ stored in the intermediate incremental storage loads in
$ group IncrLds.
$
LOAD LIST GROUP 'IncrLds'
OUTPUT BY MEMBER
LIST DISPL JOINT 16

$ FINISH
```

Figure 2.6.6-2 Complete GTSTRUDL Input file for Example PA-1

Figure 2.6.6-3 below includes the intermediate output from the pushover analysis of Example PA-1, showing the list of the intermediate storage loads created and the load factor history.

```
**** INFO_STPACP -- Pushover analysis has converged after  38 load increments.

**** INFO_STPACP -- The current collapse load factor =  23.6896
                    Load components and results are stored in the following intermediate loads:
                       PA1__001  PA1__002  PA1__003  PA1__004
                       PA1__005  PA1__006  PA1__007  PA1__008
                       PA1__009  PA1__010  PA1__011  PA1__012
                       PA1__013  PA1__014  PA1__015  PA1__016
                       PA1__017  PA1__018  PA1__019  PA1__020
                       PA1__021  PA1__022  PA1__023  PA1__024
                       PA1__025  PA1__026  PA1__027  PA1__028
                       PA1__029  PA1__030  PA1__031  PA1__032
                       PA1__033  PA1__034  PA1__035  PA1__036
                       PA1__037  PA1__038

**** INFO_STPACP -- The incremental loads above are stored in load group IncrLds .


    /----- Push-over Analysis Load Factor History -----/
         Load Increment                Load Factor
         --------------                -----------
            PA1__001                     1.000000
            PA1__002                     2.000000
            PA1__003                     3.000000
            PA1__004                     4.000000
            PA1__005                     5.000000
            PA1__006                     6.000000
            PA1__007                     7.000000
            PA1__008                     8.000000
            PA1__009                     9.000000
            PA1__010                    10.000000
            PA1__011                    11.000000
            PA1__012                    12.000000
            PA1__013                    13.000000
            PA1__014                    14.000000
            PA1__015                    15.000000
            PA1__016                    16.000000
            PA1__017                    17.000000
```

```
            PA1__018                       18.000000
            PA1__019                       19.000000
            PA1__020                       20.000000
            PA1__021                       21.000000
            PA1__022                       22.000000
            PA1__023                       23.000000
            PA1__024                       23.200001
            PA1__025                       23.240002
            PA1__026                       23.280003
            PA1__027                       23.320004
            PA1__028                       23.360004
            PA1__029                       23.400005
            PA1__030                       23.440006
            PA1__031                       23.480007
            PA1__032                       23.520008
            PA1__033                       23.560009
            PA1__034                       23.600010
            PA1__035                       23.640011
            PA1__036                       23.680012
            PA1__037                       23.688011
            PA1__038                       23.689611

 **** INFO_STPACP - Time to complete pushover analysis =    16.04 seconds.
```

Figure 2.6.6-3 Intermediate Output from the Pushover Analysis of Example PA-1

Figure 2.6.6-4 shows the output from the LIST DISPLACEMENTS command following the pushover analysis in Example PA-1. The output for joint 16 is listed by member for the intermediate storage loads created by the pushover analysis. Note the reference to the load group "IncrLds" in the LOAD LIST command.

```
 {  221} > $
 {  222} > $ List displacements at joint 16 for the loading history
 {  223} > $ stored in the intermediate incremental storage loads in
 {  224} > $ group IncrLds.
 {  225} > $
 {  226} > LOAD LIST GROUP 'IncrLds'
 {  227} > OUTPUT BY MEMBER
 {  228} > LIST DISPL JOINT 16
1

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


     PROBLEM - PA-1      TITLE - Example of pushover analysis of tower


     ACTIVE UNITS  INCH LB   RAD  DEGF SEC

     RESULTANT JOINT DISPLACEMENTS SUPPORTS

 JOINT            LOADING      /-----------------DISPLACEMENT-----------------//-------------------ROTATION-------------------/
                                   X DISP.         Y DISP.         Z DISP.         X ROT.         Y ROT.         Z ROT.


     RESULTANT JOINT DISPLACEMENTS FREE JOINTS

 JOINT            LOADING      /-----------------DISPLACEMENT-----------------//-------------------ROTATION-------------------/
                                   X DISP.         Y DISP.         Z DISP.         X ROT.         Y ROT.         Z ROT.
 16        GLOBAL
                  PA1__001       0.0002140      -0.0050172       0.0000778       0.0000005       0.0000088       0.0000000
                  PA1__002       0.0002356      -0.0100627       0.0001558       0.0000010       0.0000171       0.0000007
```

PA1__003	0.0002573	-0.0151083	0.0002338	0.0000016	0.0000254	0.0000014
PA1__004	0.0002790	-0.0201539	0.0003117	0.0000021	0.0000335	0.0000021
PA1__005	0.0003006	-0.0251996	0.0003897	0.0000026	0.0000415	0.0000027
PA1__006	0.0003223	-0.0302453	0.0004677	0.0000031	0.0000492	0.0000034
PA1__007	0.0003440	-0.0352910	0.0005457	0.0000036	0.0000568	0.0000040
PA1__008	0.0003656	-0.0403369	0.0006237	0.0000040	0.0000642	0.0000046
PA1__009	0.0003873	-0.0453827	0.0007018	0.0000045	0.0000713	0.0000051
PA1__010	0.0004089	-0.0504286	0.0007798	0.0000050	0.0000781	0.0000056
PA1__011	0.0004306	-0.0554746	0.0008578	0.0000054	0.0000845	0.0000061
PA1__012	0.0004522	-0.0605206	0.0009358	0.0000058	0.0000904	0.0000064
PA1__013	0.0004739	-0.0655667	0.0010139	0.0000062	0.0000957	0.0000067
PA1__014	0.0004955	-0.0706129	0.0010919	0.0000066	0.0001002	0.0000069
PA1__015	0.0005171	-0.0756591	0.0011700	0.0000069	0.0001037	0.0000069
PA1__016	0.0005388	-0.0807053	0.0012481	0.0000072	0.0001058	0.0000067
PA1__017	0.0005603	-0.0857517	0.0013262	0.0000074	0.0001058	0.0000061
PA1__018	0.0005819	-0.0907981	0.0014044	0.0000075	0.0001027	0.0000051
PA1__019	0.0006034	-0.0958446	0.0014826	0.0000074	0.0000947	0.0000032
PA1__020	0.0006248	-0.1008912	0.0015609	0.0000069	0.0000778	0.0000000
PA1__021	0.0006459	-0.1059380	0.0016394	0.0000058	0.0000424	-0.0000062
PA1__022	0.0006665	-0.1109852	0.0017185	0.0000029	-0.0000429	-0.0000205
PA1__023	0.0006840	-0.1160343	0.0018004	-0.0000091	-0.0003734	-0.0000741
PA1__024	0.0006856	-0.1170451	0.0018183	-0.0000168	-0.0005835	-0.0001080
PA1__025	0.0006857	-0.1172474	0.0018221	-0.0000190	-0.0006449	-0.0001179
PA1__026	0.0006856	-0.1174498	0.0018261	-0.0000217	-0.0007165	-0.0001294
PA1__027	0.0006854	-0.1176522	0.0018301	-0.0000248	-0.0008010	-0.0001430
PA1__028	0.0006849	-0.1178547	0.0018344	-0.0000285	-0.0009023	-0.0001594
PA1__029	0.0006841	-0.1180573	0.0018388	-0.0000330	-0.0010259	-0.0001793
PA1__030	0.0006829	-0.1182600	0.0018436	-0.0000387	-0.0011800	-0.0002041
PA1__031	0.0006810	-0.1184629	0.0018489	-0.0000460	-0.0013776	-0.0002359
PA1__032	0.0006781	-0.1186661	0.0018548	-0.0000556	-0.0016399	-0.0002781
PA1__033	0.0006736	-0.1188694	0.0018618	-0.0000690	-0.0020047	-0.0003369
PA1__034	0.0006660	-0.1190726	0.0018705	-0.0000890	-0.0025466	-0.0004241
PA1__035	0.0006514	-0.1192742	0.0018822	-0.0001216	-0.0034321	-0.0005669
PA1__036	0.0006128	-0.1194590	0.0018979	-0.0001825	-0.0050836	-0.0008341
PA1__037	0.0005948	-0.1194836	0.0018999	-0.0001999	-0.0055566	-0.0009112
PA1__038	0.0005948	-0.1194917	0.0019000	-0.0002000	-0.0055569	-0.0009113

Figure 2.6.6-4 Joint Displacement Output from Example PA-1

2.6.6.2 The PRINT PUSHOVER ANALYSIS DATA Command

General form:

PRINT PUSHOVER (ANALYSIS DATA)

Explanation:

The PRINT PUSHOVER ANALYSIS DATA command prints a listing of the parameter values specified by the PUSHOVER ANALYSIS DATA command described in Section 2.6.6.1 above.

Example:

Figure 2.6.6-5 below shows and example of the output from the PRINT PUSHOVER ANALYSIS DATA command. The printed data reflects the pushover analysis data values used in Example PA-1.

```
{  219} > PRINT PUSHOVER ANALYSIS DATA

******************************
* Data from internal storage *
******************************

Pushover Analysis Data
======================

   Maximum number of collapse load trials    =   20
   Collapse convergence tolerance            =    0.002000
   Incremental load id                       =    1
   Constant load id                          =    2
   Maximum number of loading increments      =   100
   Initial incremental loading rate          =    1.0000
   Maximum number of equilibrium
     correction cycles                       =   50
   Equilibrium/displacement convergence
     tolerance                               =    0.000100
   Collapse load convergence rate            =    0.200000


*************************************
* End of data from internal storage *
*************************************
```

Figure 2.6.6-5 Output from the PRINT PUSHOVER ANALYSIS DATA Command, Example PA-1

2.6.6.3 The LIST PUSHOVER ANALYSIS DUCTILITY RATIO Command

General form:

$$\underline{\text{LIST}}\ \underline{\text{PUSH}}\text{OVER}\ (\underline{\text{ANA}}\text{LYSIS}\ \underline{\text{DUC}}\text{TILITY}\ \underline{\text{RAT}}\text{IO}) \left\{ \begin{array}{l} \underline{\text{TX}} \\ \underline{\text{TY}} \\ \underline{\text{TZ}} \\ \underline{\text{RX}} \\ \underline{\text{RY}} \\ \underline{\text{RZ}} \end{array} \right\} (\underline{\text{YIE}}\text{LD}\ (\underline{\text{STR}}\text{AIN})\ v_{YS}\ -$$

$$(\left\{ \begin{array}{l} \underline{\text{STE}}\text{EL} \\ \underline{\text{CONC}}\text{RETE} \end{array} \right\}))\quad \underline{\text{TAR}}\text{GET}\ (\underline{\text{JOI}}\text{NT}) \left\{ \begin{array}{l} i_T \\ 'a_T' \end{array} \right\}$$

Elements:

v_{YS} = magnitude of the steel or concrete plastic hinge yield strain that is used to determine the yield displacement for the computation of the pushover analysis ductility ratio. A negative value represents compression strain; a positive value represents tension strain.

i_T/a_T = integer or alphanumeric name of the target joint for which the pushover analysis ductility ratio is to be computed.

Explanation:

If a structural model contains plastic hinge/segment member nonlinearity as described in the NONLINEAR EFFECTS command (Section 2.5.2, Volume 3, GTSTRUDL User Reference Manual), the LIST PUSHOVER DUCTILITY RATIO command may be used following the successful execution of the PERFORM PUSHOVER ANALYSIS command to compute and list a global ductility ratio for a particular degree of freedom at a selected target joint. The degree of freedom is selected from the list of options TX, TY, TZ, which are the global X, Y, and Z joint translation displacements respectively, and RX, RY, and RZ, which are the global joint rotations about the global X, Y, and Z axes respectively. The TARGET JOINT option names the joint, i_T/a_T, at which the pushover analysis ductility ratio is to be computed.

The pushover ductility ratio is computed as the following ratio:

$$R_{Ductility} = \frac{U_{Ult}}{U_Y} \qquad \text{Eq. 2.6.6-1}$$

where,

$R_{Ductility}$ = the pushover ductility ratio,

U_Y = the selected target joint displacement component associated with the pushover analysis intermediate storage load at which the specified plastic hinge yield strain is first detected throughout the structure,

U_{Ult} = the selected target joint displacement component associated with the last pushover analysis intermediate storage load. The total applied load level associated with this intermediate storage load may or may not correspond to a structural collapse condition.

The optional YIELD STRAIN parameter is used to specify the value of the plastic hinge yield strain, v_{YS}, for the calculation of U_Y. U_Y at the target joint is taken from the intermediate pushover analysis storage load corresponding to the total incremental load magnitude at which v_{YS} is first equaled or exceeded in any plastic hinge throughout the structure. The optional STEEL or CONCRETE YIELD STRAIN specification is used to restrict the search for v_{YS} to either STEEL or CONCRETE fibers. If STEEL or CONCRETE is not specified, then the smallest default yield strain of any material type is used.

The value of $R_{Ductility}$ is sensitive to the fiber grid geometry defined for the plastic hinges and the load rate parameter specified in the PUSHOVER ANALYSIS DATA command above. A finer fiber grid geometry, i.e. more and smaller "fibers," and a smaller load rate parameter value will produce a more accurate ductility ratio because a smaller portion of the plastic hinge will develop at the pushover analysis load level when plastic hinge formation first occurs. However, note that greater numbers of fibers and smaller load rates generally produce longer analysis times.

Errors:

The following warning or informational messages may be issued as a result of problems encountered by the LIST PUSHOVER ANALYSIS DUCTILITY RATIO command:

**** INFO_STPADR – Plastic hinge displacement results do not exist. Command ignored.

**** INFO_STPADR – Output of pushover analysis ductility factors first requires a pushover analysis. Command ignored.

These two informational messages are printed if a pushover analysis has not been executed prior to giving the LIST PUSHOVER ANALYSIS DUCTILITY RATIO command. The LIST command is merely ignored and the condition can be corrected by executing the pushover analysis with the PERFORM PUSHOVER ANALYSIS command (Section 2.6.6.1).

**** WARNING_STPADR – Pushover analysis results have been deleted or a r e otherwise missing. Command ignored.

This warning message is printed if the results of one or more of the intermediate storage loads created by the pushover analysis have been deleted. It will have been necessary to name such loads explicitly under a DELETIONS command in order to cause this to occur.

Example:

Example PA-2 illustrates a pushover analysis for a simple steel portal frame having geometric and plastic hinge nonlinearities. Plastic hinges are permitted to form at the ends of all members. The plastic hinge data are tabulated as follows:

Yield stress	=	50.0 ksi
Hinge length	=	2.0 inches
Number of flange width fiber divisions	=	8
Number of flange thickness fiber divisions	=	1
Number of web depth fiber divisions	=	8
Number of web thickness fiber divisions	=	1

Figure 2.6.6-6 shows the cross section fiber grid for the plastic hinge model.

Figure 2.6.6-7 shows a picture of the structure created by GTMENU. Dimensions, boundary conditions, member properties and loading conditions are shown.

The constant load for the pushover analysis is the uniform gravity load of 20 kips/ft applied to beam members 2 and 3 while the incremental load is the positive global X joint load applied to joint 2.

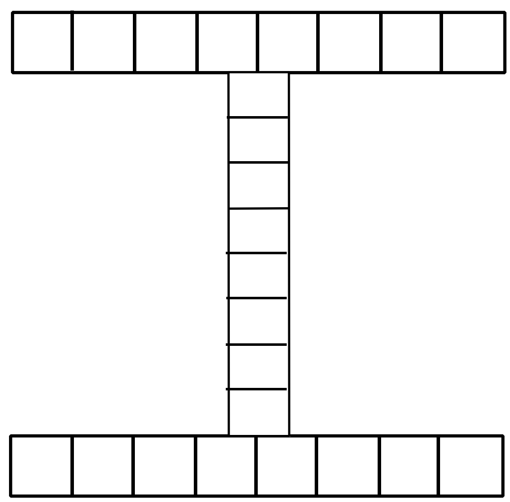


Figure 2.6.6-6 Plastic Hinge Cross Section Fiber Grid, Example PA-2

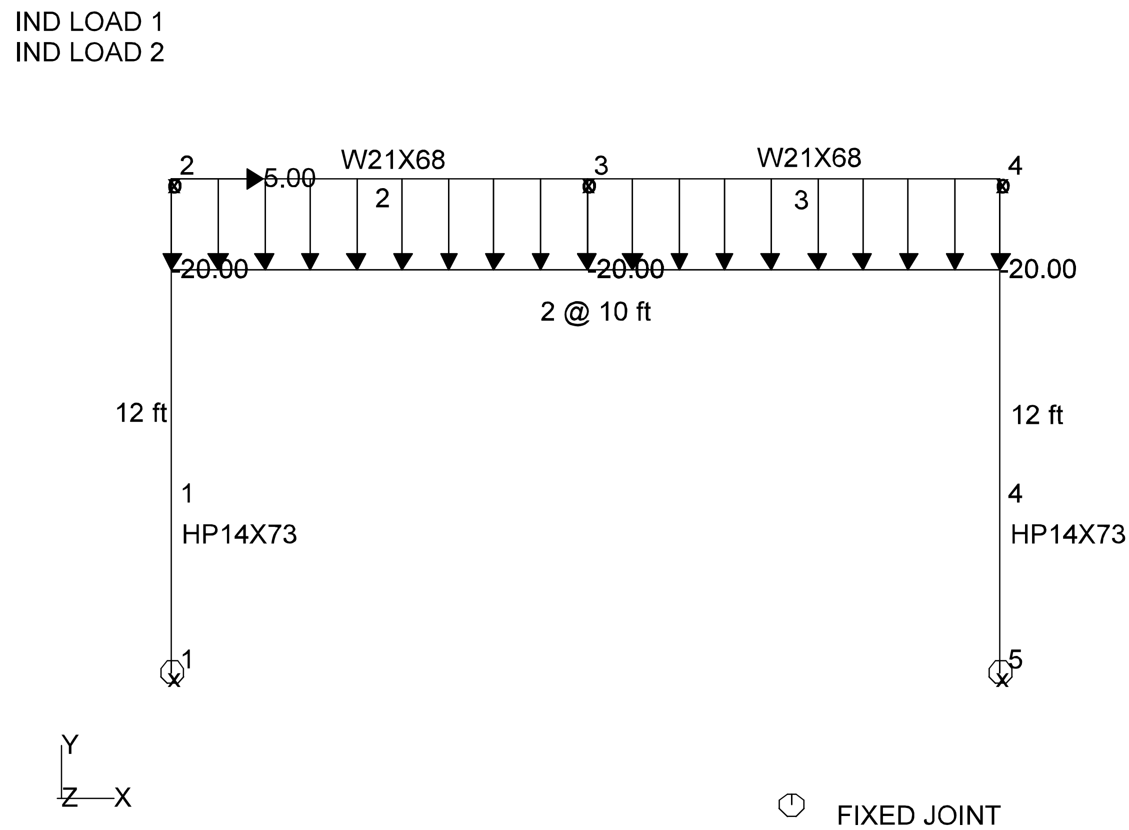


Figure 2.6.6-7 Portal Frame Model for Example PA-2

Figure 2.6.6-8 below lists the complete GTSTRUDL input file for Example PA-2. Following the pushover analysis, the LIST PUSHOVER DUCTILITY RATIO and LIST PLASTIC HINGE DUCTILITY RATIO commands compute and output these ductility ratio results at joint 2 and member 3.

```
STRUDL  'PA-2'  'Pushover analysis with plastic hinge nonlinear behavior'
$$
$$     This   GTSTRUDL file created from GTMenu on  2/ 2/99
$$
$$
UNITS  FEET  LBS   DEG   FAH
$$
$$
JOINT COORDINATES GLOBAL
   '1       '            0.0000        0.0000        0.0000
   '2       '            0.0000       12.0000        0.0000
   '3       '           10.0000       12.0000        0.0000
   '4       '           20.0000       12.0000        0.0000
```

```
   '5       '            20.0000        0.0000        0.0000
$$
$$
UNITS  FEET  LBS   DEG   FAH
$$
$$
$$
TYPE PLANE FRAME
MEMBER INCIDENCES
     '1       '      '1       '      '2       '
     '2       '      '2       '      '3       '
     '3       '      '3       '      '4       '
     '4       '      '5       '      '4       '
$$
$$
UNITS  FEET  LBS   DEG   FAH
$$
MEMBER PROPERTIES    TABLE 'M/S/HP9 '  'HP14X73 '

   '1       '      '4       '
MEMBER PROPERTIES    TABLE 'WSHAPES9'  'W21X68  '

   '2       '      '3       '
$$
STATUS SUPPORT  -
   '1       '      '5       '
$$
$$
UNITS  FEET  LBS   DEG   FAH
$$
CONSTANTS
     BETA      0.00000 ALL
$$
UNITS  FEET  LBS   DEG   FAH
$$
CONSTANTS
     E    4.1759995E+09  ALL
     G    1.5839999E+09  ALL
     POI  3.0000001E-01  ALL
     DEN  4.8954239E+02  ALL
     CTE  6.4999999E-06  ALL
$
$ Specify the plastic hinge model data
$   Nmber of flange thickness fibers (NTF) = 1
$   Number of web thickness fibers (NTW) = 1
$   Number of flange width fibers (NBF)= 8
$   Number of web depth fibers (ND) = 8
$   Plastic hinge length (LH) = 2.0 inches
$   Fy (FY) = 50.0 ksi
$
UNITS INCHES KIPS
NONLINEAR EFFECTS
  GEOMETRY MEMBERS 1 4
  PLASTIC HINGE FIBER GEOMETRY NTF 1 NTW 1 NBF 8 ND 8 LH 2.0  -
```

```
                STEEL FY 50.0  -
                MEMBERS 1 TO 4

$
$ Define the pushover analysis constant and incremental loads
$
UNITS  FEET  KIPS   DEG   FAH
LOADING  1 'Constant load'
MEMB LOADS  FOR Y GLO LIN FRA  WA   -20.0 WB   -20.0 -
   LA   0.000 LB   1.000
   2 3
LOADING 2 'Incremental lateral load'
JOINT LOADS
  2 FORCE X 5.0

$
$ Specify pushover analysis data and execute pushover analysis
$
PUSHOVER ANALYSIS DATA
  INCREMENTAL LOAD 2
  CONSTANT LOAD 1
  MAXIMUM NUMBER OF LOAD INCREMENTS 100
  LOADING RATE 2.0
  CONVERGENCE RATE 0.8
  MAXIMUM NUMBER OF TRIALS 20
  MAXIMUM NUMBER OF ITERATIONS 50
  CONVERGENCE TOLERANCE COLLAPSE 0.002
  CONVERGENCE TOLERANCE DISPL 0.0001
END

PRINT PUSHOVER ANALYSIS DATA

PERFORM PUSHOVER ANALYSIS

LIST PUSHOVER DUCTILITY RATIO TX TARGET JOINT 2
LIST PLASTIC HINGE DUCTILITY RATIO RZ MEMBERS 1 TO 4

LIST PUSHOVER ANALYSIS LIMIT STATE LOAD STRAIN -0.0164 STEEL
LIST PUSHOVER ANALYSIS LIMIT STATE LOAD STRAIN  0.0164 STEEL

FINISH
```

Figure 2.6.6-8 GTSTRUDL Input File for Example PA-2

Figure 2.6.6-9 shows the output from the LIST PUSHOVER DUCTILITY RATIO command, indicating the target joint 2, the degree of freedom, global X displacement (TX) for which the ductility ratio is computed, and the computed ductility ratio. The intermediate pushover analysis storage load and the member corresponding to when and where the default steel yield strain was first detected are also listed.

```
{  106} >
{  107} > LIST PUSHOVER DUCTILITY RATIO TX TARGET JOINT 2

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


Pushover Analysis Ductility Ratio
=================================
  Target joint = 2        DOF = TX  (Yield Loading PA2__001, Member 2   )
    Ductility Ratio = 8.679074
```

Figure 2.6.6-9 Output from LIST PUSHOVER DUCTILITY RATIO Command, Example PA-2

2.6.6.4 The LIST PLASTIC HINGE DUCTILITY RATIO Command

General form:

LIST PLASTIC (HINGE) DUCTILITY (RATIO) $\left\{\begin{matrix} \text{TX} \\ \text{RY} \\ \text{RZ} \end{matrix}\right\}$ (YIELD (STRAIN) v_{YS} -

($\left\{\begin{matrix} \text{STEEL} \\ \text{CONCRETE} \end{matrix}\right\}$)) (MEMBER list)

Elements:

v_{YS} = magnitude of the steel or concrete plastic hinge/segment yield strain that is used to determine the yield displacement for the computation of the plastic hinge ductility ratio. A negative value represents compression strain; a positive value represents tension strain.

list = list of members. If not given, all members having plastic hinge/segment nonlinearity are assumed.

Explanation:

The LIST PLASTIC HINGE DUCTILITY RATIO command is the companion to the LIST PUSHOVER DUCTILITY command, described in Section 2.6.6.3 above, and also applies to pushover analyses for structural models that contain plastic hinge/segment nonlinearity. This command may be given following the successful execution of the PERFORM PUSHOVER ANALYSIS command and lists the ductility ratio for the specified degree-of-freedom at the start and end of the specified members.

The plastic hinge ductility ratio is computed similar to the pushover ductility ratio using Equation 2.6.6-1 above, where the displacements are the member end plastic hinge displacements rather than global joint displacements.

Also similar to the LIST PUSHOVER DUCTILITY command, the optional YIELD STRAIN parameter is used to specify the value of the plastic hinge yield strain, v_{YS}, for the calculation of plastic hinge yield displacement. The plastic hinge yield displacement is taken from the intermediate pushover analysis storage load corresponding to the total incremental load magnitude at which v_{YS} is first equaled or exceeded in each of the start and end plastic hinges in a member. The optional STEEL or CONCRETE YIELD STRAIN specification is used to restrict the search for v_{YS} to either STEEL or CONCRETE fibers. If STEEL or CONCRETE is not specified, then the smallest default yield strain of any material type is used.

Errors:

The following warning or informational messages may be issued as a result of problems encountered by the LIST PLASTIC HINGE DUCTILITY RATIO command:

**** INFO_STPHDR -- Plastic hinge displacement results do not exist. Command ignored.

**** INFO_STPHDR -- Output of plastic hinge ductility factors first requires a pushover analysis. Command ignored.

These two informational messages are printed if a pushover analysis has not been executed prior to giving the LIST PLASTIC HINGE DUCTILITY RATIO command. The LIST command is merely ignored and the condition can be correct by executing the pushover analysis with the PERFORM PUSHOVER ANALYSIS command (Section 2.6.6.1).

**** WARNING_STPHDR -- Pushover analysis results have been deleted or are otherwise missing. Command ignored.

This warning message is printed if the results of one or more of the intermediate storage loads created by the pushover analysis have been deleted. It will have been necessary to name such loads explicitly under a DELETIONS command in order to cause this to occur.

Example:

The printed results of the LIST PLASTIC HINGE DUCTILITY RATIO command in Example PA-2 (input listing shown above in Figure 2.6.6-8) are illustrated below in Figure 2.6.6-10. Note that three dashes, "- - -," indicates the absence of plastic hinge formation. In this case, plastic hinges did not form at the start and end of member 1 and the start of member 2.

```
{  108} > LIST PLASTIC HINGE DUCTILITY RATIO RZ MEMBERS 1 TO 4

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


Plastic Hinge Ductility Ratios
===============================

                           Ductility Ratios -- Displacement = RZ
 Member                   Start    Yld Ld          End     Yld Ld
 ------                  -------  --------        -------  --------

1                         ---                      ---
2                         ---                       2.42   PA2__001
3                          2.42   PA2__001          1.32   PA2__001
4                          2.57   PA2__049          9.24   PA2__001
```

Figure 2.6.6-10   Plastic Hinge Ductility Ratio Results, Example PA-2

2.6.6.5 The LIST PUSHOVER LIMIT LOAD Command

General:

LIST PUSHOVER (ANALYSIS) LIMIT (STATE LOAD) STRAIN $\left\{\begin{matrix}\text{STEEL}\\ \text{CONCRETE}\end{matrix}\right\}$ v_{LS} –

(MEMBERS list)

Elements:

v_{LS} = the decimal value of the STEEL or CONCRETE limit strain that is used to determine the limit state loading condition for each specified plastic hinge. A negative value represents compression strain; a positive value represents tension strain.

list = list of members. If not given, all members having plastic hinge/segment nonlinearity are assumed.

Explanation:

The LIST PUSHOVER LIMIT STATE LOAD command is used to print a list of the pushover analysis intermediate storage loads at which the specified STEEL or CONCRETE limit state strain is first equaled or exceeded for all plastic hinges in the optional list of specified members. Members that do not have plastic hinge/segment nonlinearity are ignored. If a particular magnitude of steel or concrete strain is defined as a limit state of interest, then this command can be used to determine that point on the pushover analysis load path, identified by a particular intermediate storage load, when that strain is first equaled or exceeded in the plastic hinge(s) of each of the set of specified members given in the member list.

Example:

The printed results of the LIST PUSHOVER LIMIT STATE LOAD commands in Example PA-2 (input listing shown above in Figure 2.6.6-8) are illustrated below in Figure 2.6.6-11. Two commands are given specifying the compression(-) and tension(+) values of the same steel limit state strain. Note that three dashes, "—," in the output indicates that the limit state strain was not equaled or exceeded for the plastic hinge at the start and/or end of the listed member. The limit state strain was equaled or exceeded only for the compression limit state strain at intermediate storage load PA2__007 in the plastic hinge at the end of member 4. The pushover analysis did not reach the tension limit state strain. The absence of a specific member list means that all members with plastic hinge/segment nonlinearity were tested.

```
{  110} > LIST PUSHOVER ANALYSIS LIMIT STATE LOAD STRAIN -0.0164 STEEL

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

Pushover Analysis Limit Point Loads:  Strain =        -0.0164000, Material = Steel
==================================================================================

  Member        Limit Ld Start       Limit Ld End
  ------        --------------       ------------

 1                   ---                 ---
 2                   ---                 ---
 3                   ---                 ---
 4                   ---               PA2__007

{  111} > LIST PUSHOVER ANALYSIS LIMIT STATE LOAD STRAIN  0.0164 STEEL

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

Pushover Analysis Limit Point Loads:  Strain =         0.0164000, Material = Steel
==================================================================================

  Member        Limit Ld Start       Limit Ld End
  ------        --------------       ------------

 1                   ---                 ---
 2                   ---                 ---
 3                   ---                 ---
 4                   ---                 ---
```

Figure 2.6.6-11 Example of Output from the List PUSHOVER ANALYSIS LIMIT STATE LOAD Command

This page intentionally left blank.

5.3.13 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:

DYNAMIC PARAMETERS
BLOCK (SIZE) i_{BS}
UPDATE (STIFFNESS EVERY) i_U (TIME STEPS)
MAXIMUM (NUMBER OF EQUILIBRIUM CYCLES) i_{MAX}
CONVERGENCE (TOLERANCE) ENERGY v_{TOL}
INITIAL (STRESS LOAD) $\left\{ \begin{array}{c} \rightarrow \text{OFF} \\ i_{ISL} \\ 'a_{ISL}' \end{array} \right\}$
RESULTS (FILE NAME) 'fn'
END (OF DYNAMIC PARAMETERS)

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:

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

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, 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.

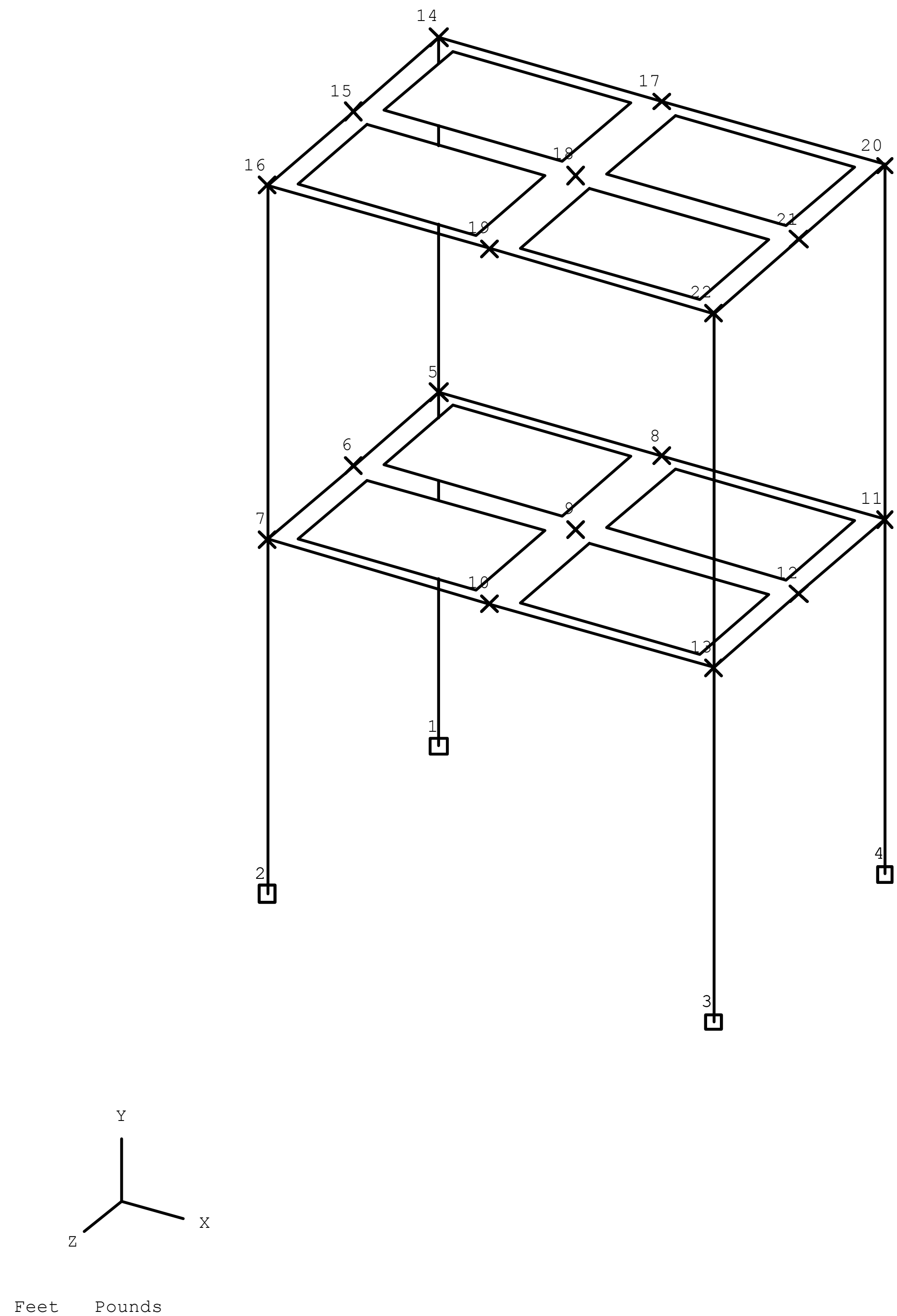


Figure 2.5.6.3-1 Nonlinear Dynamic Analysis Example Structure – Joints Labeled

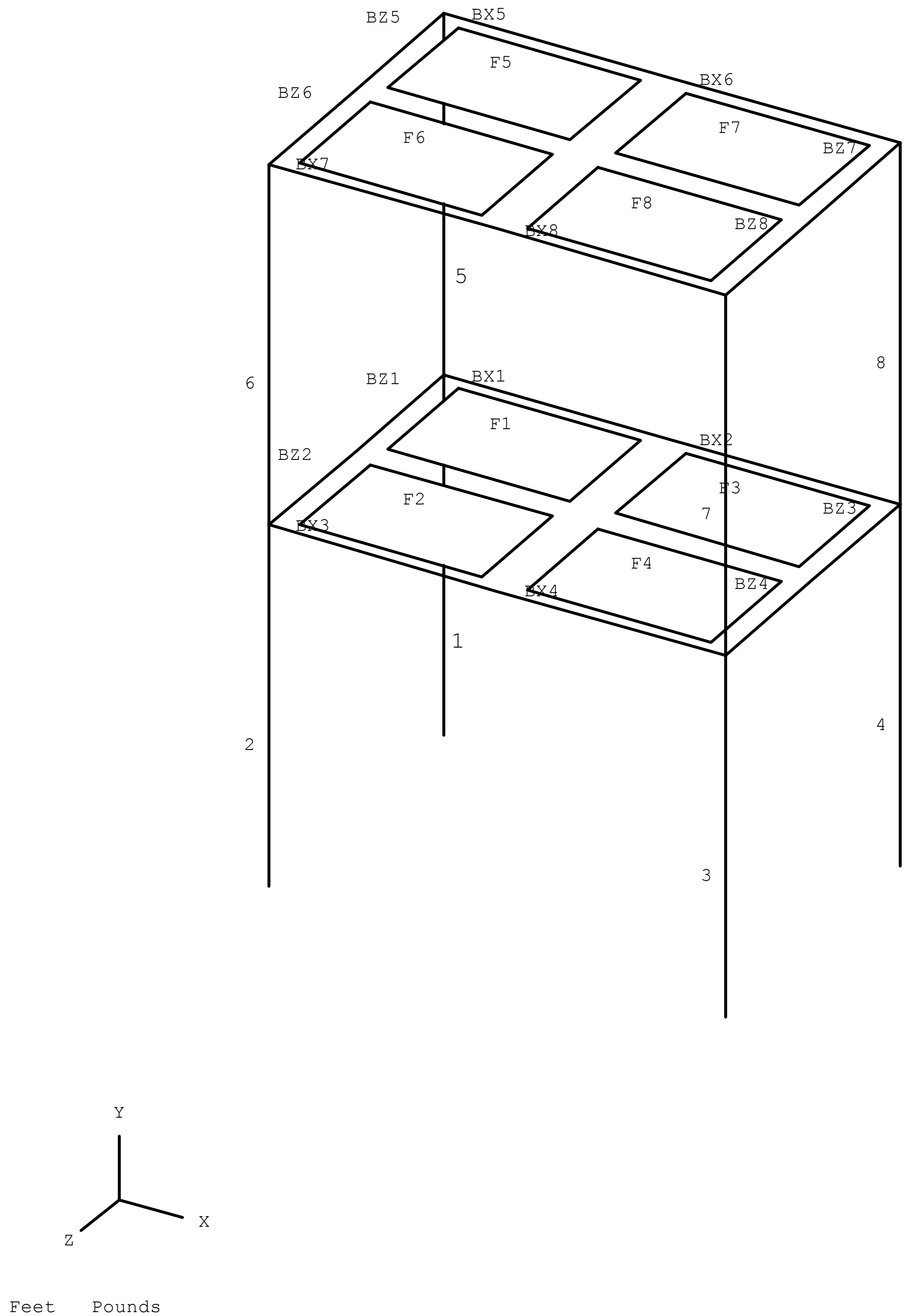


Figure 2.5.6.3-2 Nonlinear Dynamic Analysis Example Structure – Members and Finite Elements Labeled

```
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.

5.4 General Prerelease Features

5.4.1 CALCULATE SOIL SPRING VALUES Command

The CALCULATE SOIL SPRING VALUES Command will calculate elastic spring at joints based on the modulus of subgrade reaction and the tributary area of finite elements attached to the joint. The section is number as it will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.12.13 The CALCULATE SOIL SPRING VALUES Command

General Form:

CALCULATE (SOIL) SPRING (VALUES) DIRECTION $\begin{Bmatrix} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{Bmatrix}$ (v_1) -

((PLANE) TOLERANCE v_2) ((PLANE) ANGLE (TOLERANCE) v_3) -
value_sets (APPLY) (SHOW (CALCS))

Elements

v_1 = optional global coordinate value of specified global plane,

v_2 = optional value of plane tolerance; element nodes must lie within this distance to be considered planar. Default is 2 inches (5 cm).

v_3 = optional value of normal angle; planar elements or selected element face normals must lie within this many angular units of the specified global axis to be considered planar. Default is 5 degrees.

value_sets = KS v_4 ELEMENTS $list_1$ (KS v_5 ELEMENTS $list_2$ ...),

v_4, v_5 ... = the value of the modulus of subgrade reaction (in current units of force per unit length^3) for the elements in 'list'. You may have as many KS-list sets as are needed.

list = list of elements to which the specified KS is to be applied. See Section 2.1.2.2, Volume 1 of the User Reference Manual for an explanation of the list concept.

Explanation:

This command allows you to calculate elastic spring constants for joints based on a specified modulus of subgrade reaction and the tributary area of finite elements attached to the joint. The entire specified elements (2D) or faces (3D) must lie in the plane of joints - not an edge only.

DIRECTION

Choose X, Y or Z global direction for the springs. A "spring plane" (plane of joints to which soil spring values will be calculated) is determined in one of three ways:

1) if v_1 is given, only elements or faces in the specified global plane, e.g. Y = -100.0, within the plane or angle tolerance, will be considered valid.

2) If v_1 is not given and any 2D elements are referenced, the first 2D element establishes the spring plane.

3) If v_1 is not given and there are no 2D elements, the first 3D element will determine the spring plane from the element face that is the algebraic minimum in the specified global direction.

PLANE TOLERANCE

All joints must lie within this value of the spring plane. The default value is 2 inches (5 cm).

ANGLE TOLERANCE

All joints in the spring plane (all nodes for 2D elements or the appropriate face for 3D elements) must lie within this tolerance of the specified direction. The default is 5 degrees (0.087 radians). Note that the areas calculated for each element or face is a projected area in the specified global direction.

APPLY

Apply the calculated spring constants to the included joints. If this option is not specified, the result of this command is only the printed output. If APPLY is specified, the following rules are checked:

1) All joints incident to the listed 2D elements must be supports.

2) All joints incident to the appropriate 3D element faces must be supports.

3) No force release or existing spring constant in the specified direction. Other releases and spring constants are acceptable but no rotations (TH1, TH2, or TH3) are allowed.

SHOW CALCS

Print the contribution of each attached element to the spring value.

Supported elements:

2D Plate bending

CPT, BPHT, BPR, BPHQ, IPBQQ

2D Plate

SBCT, SBCR, SBHQ, SBQCSH, SBHT, SBHT6, SBHQ6

3D

TRIP, IPLS, IPQS, IPSL, IPSQ

Discussion:

When this command is issued, the list(s) of elements are concatenated and all the nodes are determined. These nodes must lie within the tolerances of the specified plane - see DIRECTION, PLANE TOLERANCE and ANGLE TOLERANCE above. Each element type is determined. If 3D elements are included, the appropriate face is chosen. For each planar element or appropriate 3D element face, a set of distribution factors is determined. These distribution factors depend on element type and nodal coordinates and correspond to the way mass is distributed for elements. Triangular elements or faces will have distribution factors of (1/3, 1/3, 1/3) regardless of nodal coordinates.

Use the SHOW CALCS option to determine how each element contributes to the total spring value at each joint to verify that the distribution factors are satisfactory for your situation.

5.4.2 Large Problem Size Command

The LARGE PROBLEM SIZE Command is used to reserve Windows Resources and is described in the section below which is numbered as it will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.3.7 The LARGE PROBLEM SIZE Command

The LARGE PROBLEM SIZE command is used to reserve Windows resources before GTSTRUDL actually uses the resources. This is done to help Windows work more efficiently and may decrease processing time for large jobs. When GTSTRUDL completes, all the Windows resources are released.

General form:

LARge (PRObLEM) SIZe i_1

Elements:

i_1 = an integer between 1 and 5 (inclusive). The largest problem size is 5 and may impact the operation of other programs running simultaneously with GTSTRUDL.

Explanation:

This command reserves resources before demands are placed on Windows, e.g. during STIFFNESS ANALYSIS, or DYNAMIC ANALYSIS. If you determine that resource reservation is desirable, it is best to put the command at the beginning of the input file to get the most efficiency improvements. You may need to experiment on your computer to find the best problem size specification for your computer resources and working environment. In general, you get the most benefit from problem size 5, but problem size 5 has the most impact on other programs running simultaneously with GTSTRUDL.

Examples:

```
{  1} >      LARGE PROBLEM SIZE 5
```

Windows has been sent a request to reserve the maximum allowable (for one program) amount of resources.

This page intentionally left blank.

5.4.3 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 alignment check because they are not used to determine the default member orientation.

Syntax:

<u>ALI</u>GN (<u>MEM</u>BERS list) (<u>REP</u>ORT (<u>ON</u>LY)) (<u>AL</u>ONG <u>Y</u> <u>AX</u>IS) -
(<u>TOL</u>ERANCE v1 (<u>RAT</u>IO)) (<u>ITE</u>MIZE (<u>WI</u>TH <u>DET</u>AIL))

Where,

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

v1 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.

5.4.4 The Locate Interference and Duplicate Joints Command

The commands described below are used to locate interference and duplicate joints. The section is numbered as it will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.12.12 The LOCATE command

General Form:

LOCATE INTERFERENCE (JOINTS) (TOLERANCE v_1)

or

LOCATE DUPLICATE JOINTS (TOLERANCE v_2) (AND) (REMOVE)

Elements:

v_1 = Joints (excluding a member's start and end joints) closer than this distance from a member are considered interference joints. The default value is 2.0 inches (5.08 cm).

v_2 = Joints within this distance are considered duplicates. The default value is 0.2 inches (5.08 mm)

Interference joints

Explanation:

This command initiates a search of the **active** members and joints to determine if any joints are within the specified tolerance distance of a member. The member's start and end joints are ignored. Finite elements, superelements and rigid bodies are not included in the search. Any joints found within the tolerance distance of a member are reported and included in the created groups **MB_Int** and **JT_Int**. This command is especially useful for finding problems in geometries created by modeling packages that convert "physical" models into "analytical" models. Extremely short members and unconnected bracing, which are two common problems with this conversion, are hard to find graphically, but become apparent by using this command.

If interfering joints are discovered, a record of the member and joint are printed in the GT STRUDL output. Note that a single member may have more than one interfering joint and that a joint may interfere with more than one member. Two groups are created with the following data:

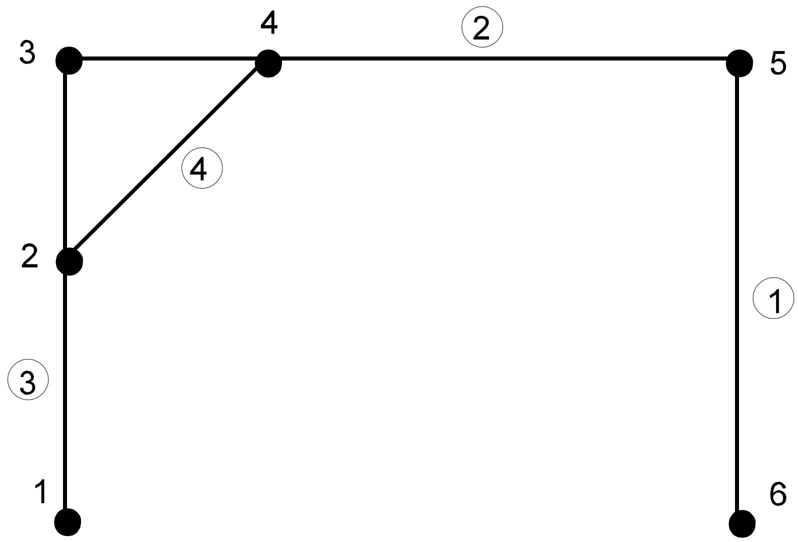
MB_Int This group is a list of all members that had one or more interfering joints. It can be useful to use the command LABEL MEMBER GRP 'MB_Int' while in the Scope environment to see which members have interfering joints. Note that the group name is case-sensitive: 'mb_int' or 'MB_INT' will not work.

JT_Int This group is a list of all joints that interfered with a member. It can be useful to use the command LABEL JOINTS GRP 'JT_Int' while in the Scope environment to see the interfering joints. Note that the group name is case-sensitive: 'jt_int' or 'JT_INT' will not work. Joints that interfere with more than one member will appear more than once in this group.

Sample Output:

```
======================================================================
      Checking joint/member interference (active members and joints only):
           Joints that lie closer than 2.00 INCH to a member and
           are neither the start nor end joint will be reported.

======================================================================

       Joint 4        is closer than the tolerance to member 2
       Joint 2        is closer than the tolerance to member 3

      **** INFO_INTFER -- The group 'MB_Int' has been created and contains
                               2 members with interfering joints.

      **** INFO_INTFER -- The group 'JT_Int' has been created and contains
                                 2 joints that interfere with members.
```

Sample Geometry:



Member incidences:			
	①	6	5
	②	5	3
	③	1	3
	④	2	4

The geometry appears valid when displayed but will generate the above output, because member 4 is not connected to the frame formed by members 1, 2, and 3.

Duplicate joints

General Form:

<u>LOC</u>ATE <u>DUP</u>LICATE <u>JOI</u>NTS (<u>TOL</u>ERANCE v_2) (<u>AND</u>) (<u>REM</u>OVE)

Explanation:

This command will find sets of joints that lie within the tolerance v_2 of each other and report each set of duplicate joints. The report includes the master joint, which is determined as the first joint with the given coordinates in the joint list, and all other joints with the same coordinates, which are considered duplicates. Each set of master joint plus duplicate(s) may have a status flag printed, indicating that the group of joints is non-homogeneous, e.g. one joint is a support but the other is not. Status flags are printed for these duplicate sets:

+ Groups that have dissimilar properties, including support status or releases.
* Groups that contain a constrained joint - JOINT TIES or RIGID BODY.
! Groups that have one or more members or nonlinear springs (NLS) between the joints in the group.

The **REMOVE** option will remove the duplicate joints from un-flagged groups by substituting the master joint for the duplicate joints in the member connectivity and group data. In addition, joint loads from the duplicate joints will be ***added*** to the master joint. Joint temperature loads from duplicate joints will ***replace*** joint temperature for the master joint if the duplicate joint temperature load is of greater magnitude than the master joint temperature load.

Sample Output:

```
 {   71} > PRINT MEM INCIDENCES MEMBERS 21 23 25

 MEMBER INCIDENCES----------/

 MEMBER   START      END
 21        11         12
 23        11         13
 25        11         15

{   72} > LOCATE DUPLICATE JOINTS AND REMOVE
```

```
 ============================================================================
 Checking for duplicate joints:
     Joints that are closer than    0.200 INCH
     to another joint will be reported.
 ============================================================================

 Duplicate joints within a tolerance of     0.2000 INCH

     5        11
     6        12
     7        13
     8        14

   Removing similar duplicate joints.
   Adding loadings from removed joint 11               to retained joint 5
   Moving temperature loads from removed joint 11      to retained joint 5
   Adding loadings from removed joint 12               to retained joint 6
   Ignoring temperature loads on removed joint 12, retained joint 6 unchanged
   Adding loadings from removed joint 13               to retained joint 7
   Adding loadings from removed joint 14               to retained joint 8

 {   73} > PRINT MEM INCIDENCES MEMBERS 21 23 25

 MEMBER INCIDENCES----------/
 MEMBER   START     END
 21         5        6
 23         5        7
 25         5        15
```

Four sets of duplicate joints, joints that are within 0.2 inches of each other, were detected. In the first duplicate set, joints 5 and 11, 5 is considered the 'master' joint, so joint 11 will be removed from the database and each occurrence of joint 11 in member and element connectivity data will be replaced with joint 5. Notice that members 21, 23 and 25 originally had joint 11 as their start joints, but after the LOCATE DUPLICATE JOINTS AND REMOVE command, their start joints are now joint 5. In addition to connectivity changes, the REMOVE option has consolidated the loadings from joint 11 to joint 5. Applied force or moment loads from 11 are ***added*** to joint 5. The joint temperature load on joint 11 was larger that the joint temperature load on joint 5, so the value from joint 11 has ***replaced*** the value on joint 5. After updating the database, joint 11 has been removed from the database.

Other output:

```
{   72} > LOCATE DUPLICATE JOINTS AND REMOVE
 ======================================================================
 Checking for duplicate joints:
     Joints that are closer than    0.200 INCH
     to another joint will be reported.
 ======================================================================

 No duplicate joints were found.
```

All the joints were searched, but there were no duplicates.

```
{   72} > LOCATE DUPLICATE JOINTS AND REMOVE
 ======================================================================
 Checking for duplicate joints:
     Joints that are closer than    0.200 INCH
     to another joint will be reported.
 ======================================================================

 Duplicate joints within a tolerance of     0.2000 INCH
      A "+" symbol indicates a group of joints within the tolerance, but
      the joints do not have similar properties (support status, releases,
      etc.)
      A "*" symbol indicates a group of joints within the tolerance, but
      at least one joint in the group is constrained (JOINT TIES or RIGID
      BODY).
      A "!" symbol indicates a group of joints within the tolerance, but
      at least one member or nonlinear spring exists between joints in the
      group.

 + ! 5        9
 +*! 6        NS27     10
 +*! 7        11
     8        12
```

Four sets of duplicate joints were found, but only one set (8, 12) could be processed by the REMOVE option, if requested. The flag type messages, for the “+”, “*” and “!” flags, are printed only if that flag type is encountered in the search. To automatically resolve duplicate joints with the REMOVE option, you will need to change the conditions that caused the flag. In many flagged cases, especially for the “!” symbol with nonlinear springs, you will want to retain the duplicate joints as a necessary part of the structure.

5.4.5 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

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

Elements:

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

r_1, r_2, r_3 = values in current angle units of the load component rotation angles 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.
```

5.4.6 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.

5.4.7 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.

5.4.8 Notes and Print Notes Command

The NOTES and PRINT NOTES commands are presented below and are numbered as they will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.12.14 The NOTES Command

A NOTE in GTSTRUDL is text information that can be assigned a type or category, and additionally, be assigned to a set of components. For example, a general note may contain the information "Model created from CAD file C:\Project\Plans.cad". Member 1 could have a note of category 'PieceMrk', with contents of "A217-3456-BFG". Notes are created with the NOTE command, and can be reviewed with the NOTE option of the PRINT command.

General Form

$$\underline{\text{NOT}}\text{ES (i) ((}\underline{\text{FOR}}\text{)}\left\{\begin{array}{l} \rightarrow\left\{\begin{array}{l}\underline{\text{GEN}}\text{ERAL}\\ \underline{\text{MOD}}\text{EL}\end{array}\right\} \\ \\ \underline{\text{JOI}}\text{NTS list} \\ \underline{\text{MEM}}\text{BERS list} \\ \underline{\text{LOA}}\text{DS list} \\ \underline{\text{GRO}}\text{UPS list}\end{array}\right\}\text{) -}$$

$$\text{(}\left\{\begin{array}{l}\underline{\text{CAT}}\text{EGORY}\\ \underline{\text{TYP}}\text{E}\end{array}\right\}\text{ category) (}\underline{\text{TEX}}\text{T) note}$$

i — note number to be deleted. See DELETIONS mode below.

category — 'a1', where a1 is 1 to 8 characters. The apostrophes are required, even if a1 is composed of all digits. "*" is not allowed to be used in a category name (this would prevent wildcards in PRINT NOTES).

note — 'text block #1' ('text block #2' ... 'text block #n')

Each block is limited to 144 characters. Trailing blanks in each block are truncated, i.e. 'abc ' becomes 'abc'. All blocks are appended into a single note. "\n" indicates a newline request. All ASCII characters and punctuation except the apostrophe (') are allowed in a note. The total number of characters in a note is limited to 8192, approximately 1.5 to 2 pages.

list A list of the appropriate components. See Section 2.1.2.2, Volume 1 in the GTSTRUDL Users Reference Manual for an explanation of the list concept.

```
                           ⎧ ⎧GENERAL⎫          ⎫
                           ⎪ ⎩MODEL  ⎭          ⎪
                           ⎪                    ⎪
PRINT NOTES  (FOR )        ⎨ JOINTS  list       ⎬ -
                           ⎪ MEMBERS  list      ⎪
                           ⎪ LOADS  list        ⎪
                           ⎩ GROUPS  list       ⎭
       ( ⎧CATEGORY⎫ category )
         ⎩TYPE    ⎭
```

category 'a1', where a1 is 1 to 8 characters. The apostrophes are required, even if a1 is composed of all digits. "*" is treated as a wildcard, matching any combination of characters. Only notes of the specified type (GENERAL, JOINTS, etc.) that match the given category will be printed. If a category is not given, all notes of the specified type will be printed.

list A list of the appropriate components. See Section 2.1.2.2, Volume 1 in the GTSTRUDL Users Reference Manual for an explanation of the list concept.

If the type of note (GENERAL, JOINTS, etc.) is not specified, all types will be printed, while still respecting the category restriction.

Note: GENERAL and MODEL are synonymous, as are CATEGORY and TYPE. It doesn’t matter which word of each pair you use.

CHANGES mode

The NOTES command acts the same in ADDITIONS or CHANGES. To change a note, first delete the note and then re-issue the NOTES command with the updated data.

DELETIONS mode

If DELETIONS mode is active, notes can be deleted by:

1) **Number**. Note the number of the note in the PRINT NOTES output. This will delete the note and all references to it.

2) **Category**. All notes of the specific category will be deleted, unless a list of components is given. In this case, all notes of the specified category applied to the specified components will be deleted.

3) **By component**. All notes for the specified components (joint, member/element, load or group) will be deleted unless a category is also specified. See #2 above.

Examples of the NOTE command:

```
$ — Sample file name
notes for member 1 text 'C:\Project\Plans.cad'

$ — Append mutiple text blocks into 1, truncating trailing blanks
notes for joints existing 1 2 category 'page 2' 'abc   ' 'def' -
'ghi  ' 'jkl' 'mno' 'pqr' 'stu' 'vwx' 'yz'

$ — Multiple text blocks with newline (\n)
note members existing type 'abc' '12345678\n' -
     'ABCDEFGHIJK\n' -
     'abcdefghijk\n' '1234567890'
$ — Allowable characters: all except ' (apostrophe / single quote)
note category 'allowed' -
     ' !"#$%()*+,-./0123456789:;<=>?@\n' -
     'ABCDEFGHIJKLMNOPQRSTUVWXYZ\n' -
     '[\]^_`\n' -
     'abcdefghijklmnopqrstuvwxyz\n' -
     '{|}~'
```

Examples of the PRINT NOTES output:

When a PRINT NOTES command is issued, a header is printed with the total number of notes, an optional category specification, and an output format description. For each note that is printed, a separator line is printed that contains the note's number following the "#". This number can used to delete the note directly - see "Deletions mode" above. Underneath the separator line two blocks of output are printed. The first block contains the category ("NotGiven" is printed for notes that do not have a category) and the contents of the note. For larger notes, the contents may be continued on more lines of output until the note is completed. The second block of output contains the note type and the components to which the note is applied. The type may be `Joints`, `Members`, `Loads`, `Groups` or `Model/General`. The list of components (except for `Model/General`, which does not accept components) follows and may print over multiple lines.

```
For note #1 below:
      Category:          NotGiven
      Note contents:     C:\Project\Plans.cad
      Type:              Members
      List of components: 1

 { 100} > PRINT NOTES


 ============================================================
 Total number of notes:   4

 ---- Output format ----
 Category   Note contents
      Type: List of components
 ============================================================

 --------   Note #1 ---------------------------------------------------
 NotGiven   C:\Project\Plans.cad
   Members: 1

 --------   Note #2 ---------------------------------------------------
 page 2     abcdefghijklmnopqrstuvwxyz
    Joints: 1        2

 --------   Note #3 ---------------------------------------------------
 abc        12345678
            ABCDEFGHIJK
            abcdefghijk
            1234567890
   Members: 1        2        3

 --------   Note #4 ---------------------------------------------------
 allowed     !"#$%()*+,-./0123456789:;<=>?@
            ABCDEFGHIJKLMNOPQRSTUVWXYZ
            [\]^_`
            abcdefghijklmnopqrstuvwxyz
            {|}~
   Model/General note:


  { 101} > PRINT NOTES FOR JOINT 1

 ============================================================
 Total number of notes:   4

 ---- Output format ----
 Category   Note contents
      Type: List of components
 ============================================================

 --------   Note #2 ---------------------------------------------------
 page 2     abcdefghijklmnopqrstuvwxyz
```

```
   Joints: 1


{ 102} > PRINT NOTES CAT  'ab*'

============================================================
   Specified category: ab*
Total number of notes:   4

---- Output format ----
Category   Note contents
     Type: List of components
============================================================

--------   Note #5 ----------------------------------------------------
abc        12345678
           ABCDEFGHIJK
           abcdefghijk
           1234567890
  Members: 1        2        3
```

5.4.9 Reference Coordinate System Command

General form:

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

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

Explanation:

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

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

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

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

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

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

5.4.9-1 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.

5.4.10 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.

End of Document.