GT STRUDL ®

GT STRUDL
Release Guide Version 26

Volume 1 of 2

February 2002

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 26, with a release date in the GTSTRUDL title block of February 2002.

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

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

Commercial Software Rights Legend

Any use, duplication, or disclosure of this software by or for the US Government shall be restricted to the terms of a license agreement in accordance with the clause at DFARS 227.7202-3 (April 1998).

GTSTRUDL® is a registered service mark of the Georgia Tech Research Corporation, Atlanta, Georgia.
© Windows NT, Windows 95, Windows 98, Windows ME, Windows 2000, and Windows XP are registered trademarks of Microsoft Corporation, Redmond Washington.

Table of Contents

Notices ii

Disclaimer ii

Commercial Software Rights Legend ii

CHAPTER 1 INTRODUCTION 1

CHAPTER 2 NEW FEATURES IN Version 26 2-1
- 2.1 GTSTRUDL Startup 2-1
- 2.3 DBX 2-24
- 2.4 Dynamics 2-24
- 2.5 Finite Element 2-25
- 2.6 General 2-25
- 2.7 GTMenu 2-31
- 2.8 Nonlinear 2-46
- 2.9 Offshore 2-46
- 2.10 Reinforced Concrete 2-47
- 2.11 Steel 2-47
- 2.12 Steel Tables 2-52

CHAPTER 3 ERROR CORRECTIONS 3-1
- 3.1 Dynamic Analysis 3-1
- 3.2 Elastic Buckling 3-1
- 3.3 Finite Elements 3-1
- 3.4 General 3-2
- 3.5 GTMenu 3-3
- 3.6 Nonlinear Analysis 3-5
- 3.7 Offshore 3-5
- 3.8 Rigid Bodies 3-5
- 3.9 Steel Design 3-6
- 3.10 Windows and Office XP 3-6

CHAPTER 4 KNOWN DEFICIENCIES 4-1
4.1 Dynamics 4-1
4.2 Finite Elements 4-1
4.3 General Input/Output 4-2
4.4 GTMenu 4-4
4.5 Rigid Bodies 4-6
4.6 Scope Environment 4-6

CHAPTER 5 PRERELEASE FEATURES 5-1
5.1 Introduction 5-1
5.2 ACI Code 318-99 5-2-
5.3 Deflection Check and Design 5-5
5.4 GTSTRUDL Indian Standard Design Code IS800 5-11
5.5 GTSTRUDL Profile Tables for the Design based on the IS800 Code 5-19
5.6 Nonlinear Effects Command (Revised) 5-25
5.7 Nonlinear Analysis Output Commands 5-47
5.8 Pushover Analysis 5-53
5.9 The CALCULATE ERROR ESTIMATE Command 5-83
5.10 Dynamic Analysis External File Solver - Improve Efficiency of Dynamic Results Computation 5-87
2.4.5.3 Specification of Miscellaneous Dynamic Parameters 5-87
5.11 Nonlinear Dynamic Analysis 5-91
2.5.6.1 Extensions to the DYNAMIC PARAMETERS Command 5-91
2.5.6.2 DYNAMIC ANALYSIS NONLINEAR Command 5-95
2.5.6.3 Nonlinear Dynamic Analysis Example 5-97
5.12 The LOCATE INTERFERENCE JOINTS Command 5-103
5.13 FORM STATIC LOAD Command -- Automatic Generation of Static Equivalent Earthquake Loads 5-109
5.14 Reference Coordinate System Command 5-119
5.14-1 Printing Reference Coordinate System Command 5-122
5.15 Rectangular and Circular Concrete Cross-Section Tables 5-123
5.16 Hashing Algorithm to Accelerate Input Processing 5-125

CHAPTER 1

INTRODUCTION

Version 26 covers GTSTRUDL operating on PC's under the Windows XP, Windows 2000, Windows NT, Windows ME, Windows 98, and Windows 95 operating systems. Chapter 2 presents the new features and enhancements which have been added since the Version 25 release. Chapter 3 provides you with details regarding error corrections that have been made since the Version 25 release. Chapter 4 describes known problems with Version 26. 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 following features are included in Version 26 as prerelease features:

1. ACI Code 318-99
2. Steel Deflection Check and Design
3. Steel Design by Indian Standard Code IS800
4. GTSTRUDL Profile Tables for the Design based on the IS800 code
5. Nonlinear Effects Command (revised)
6. Nonlinear Analysis Output Commands
7. Pushover Analysis
8. Calculate Error Estimate Command
9. Dynamic Analysis External File Solver to Improve Efficiency of Dynamic Analysis Results Computation
10. Nonlinear Dynamic Analysis
11. The Locate Interference and Duplicate Joint Command
12. Automatic Generation of Static Equivalent Earthquake Loads
13. Reference Coordinate System Command
14. Rectangular and Circular Concrete Cross-Section Tables
15. Hashing Algorithm to Accelerate Input Processing

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

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

This page intentionally left blank.

CHAPTER 2

NEW FEATURES IN Version 26

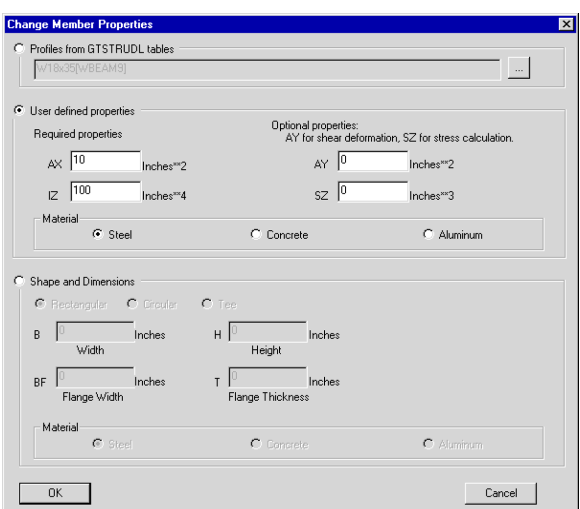
This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GTSTRUDL in Version 26. This release guide is also available online upon execution of GTSTRUDL under Help/Reference Documentation/GTSTRUDL 26 Release Guide. Other documentation has also changed in Version 26. You should also review the following online documentation which is available under Help:

- GTMenu
- Getting Started
- Analysis Guide
- Design Guide

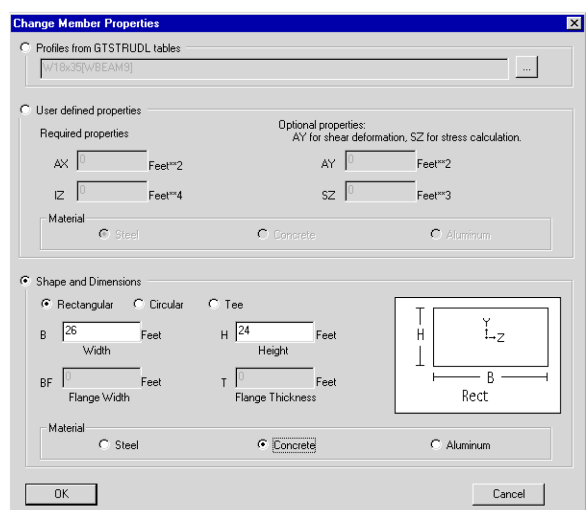
2.1 GTSTRUDL Startup

Model Wizard

New options to specify materials and member cross sectional properties are now available under the Continuous Beam and Plane Frame options in the Model Wizard. For instance, in the Plane Frame option in the Model Wizard when you are specifying the properties for beams and click on the Browse button, you may now specify user defined prismatic properties or member cross-section dimensions for rectangular, circular, and tee cross-sections. Examples of the new dialog for user defined and cross-section dimensions are shown below:



User Defined Properties



Shape and Dimensions

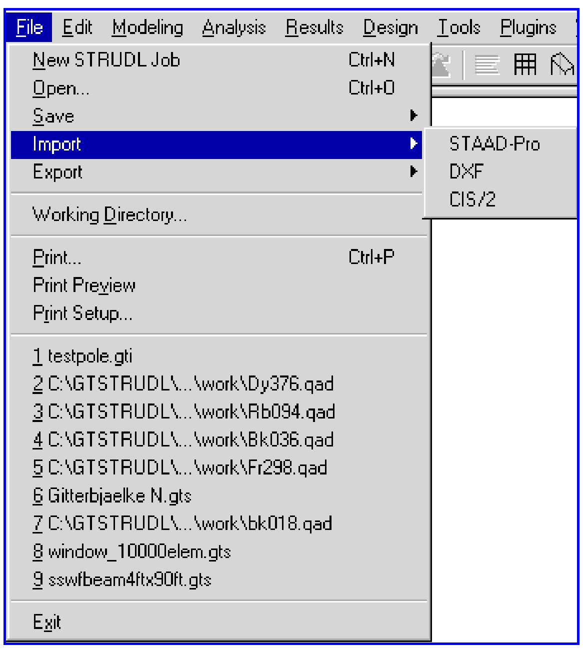
You may also specify the type of material as being steel, concrete, or aluminum in this new dialog.

2.2 GTSTRUDL Output Window

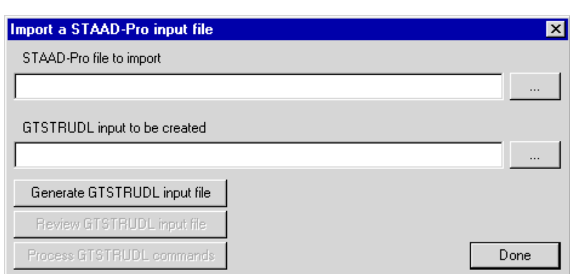
Numerous new features and enhancements have been added to the GTSTRUDL Output Window. These additions are presented below by the menu headings located at the top of the GTSTRUDL Output Window.

File Pulldown

You may now Import STAAD-Pro, DXF, and CIS/2 files into GTSTRUDL as shown in the File Pulldown Import option below:

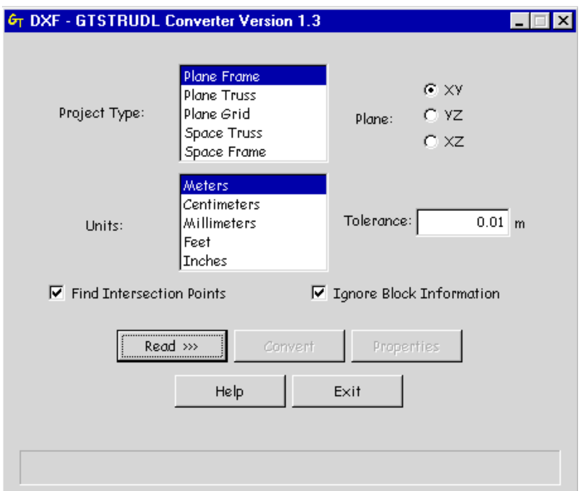


The STAAD-Pro option will take a text input file from STAAD-Pro and convert it into a GTSTRUDL input file. The following dialog appears when you have selected the Import STAAD-Pro option:

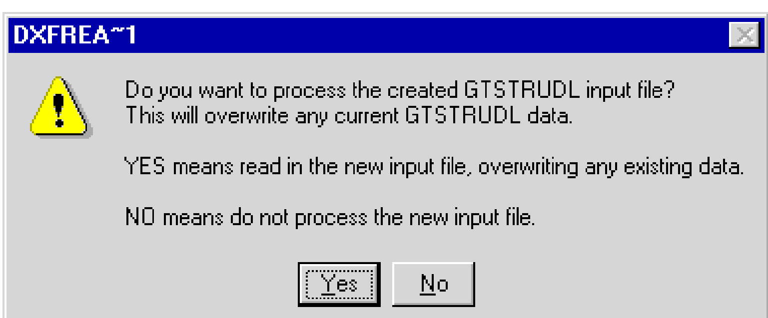


As you can see from this dialog, you may specify the file to be imported and the GTSTRUDL file to be created. You may also review the generated input file and have the generated input file processed in GTSTRUDL. Please note that not all STAAD commands are recognized so the user should carefully review the converted GTSTRUDL input file.

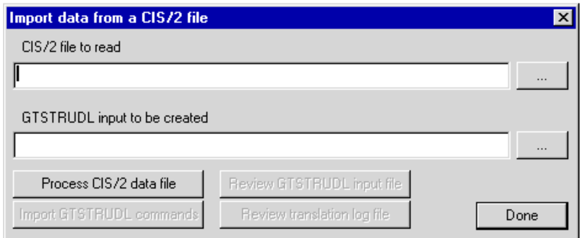
The DXF option will take a DXF file which contains lines and polylines and convert them into a GTSTRUDL input file containing the structural type (Plane Frame, Space Frame, etc.), Units, Joint Coordinates, and Member Incidences. Intersecting lines will be automatically split. An example of the Import DXF dialog is shown below:



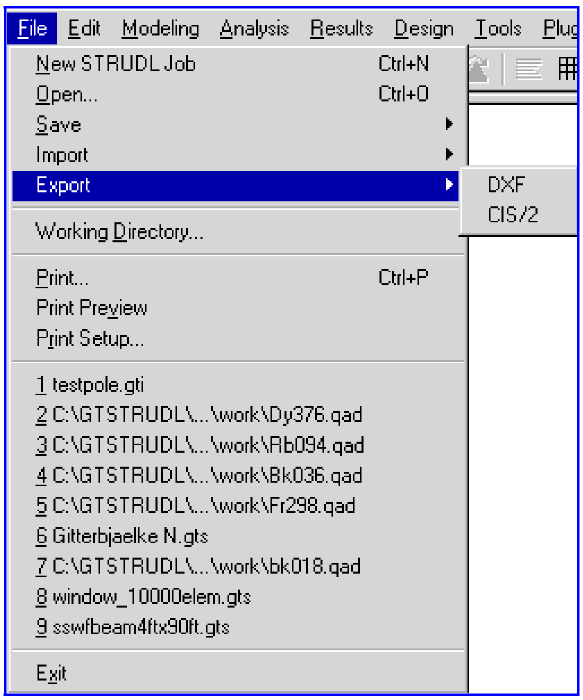
After reading, converting, and exiting the above dialog, you then have the option to process the generated input file in GTSTRUDL as shown below:



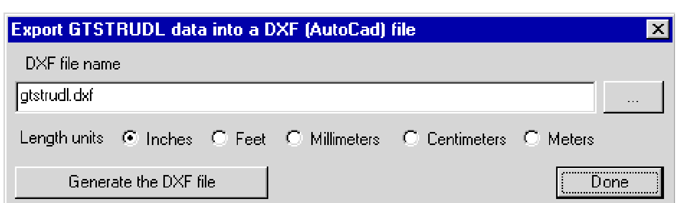
The CIS/2 option will take a CIS/2 file and convert the geometry including steel profile shapes and loadings into a GTSTRUDL input file. After processing the input file, you then have the file options to review the GTSTRUDL input file and import the file containing the GTSTRUDL commands. An example of the Import CIS/2 dialog is shown on the next page:



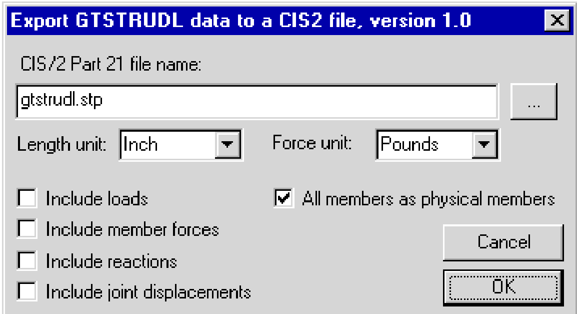
You may now Export GTSTRUDL data to DXF and CIS/2 files using the Export dialog shown below:



The Export DXF option will export geometry to a DXF file using the following dialog:

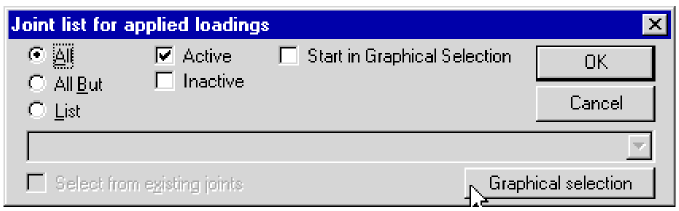


The Export CIS/2 option will export geometry including steel profile data and joint displacements and reactions and member end forces using the following dialog:

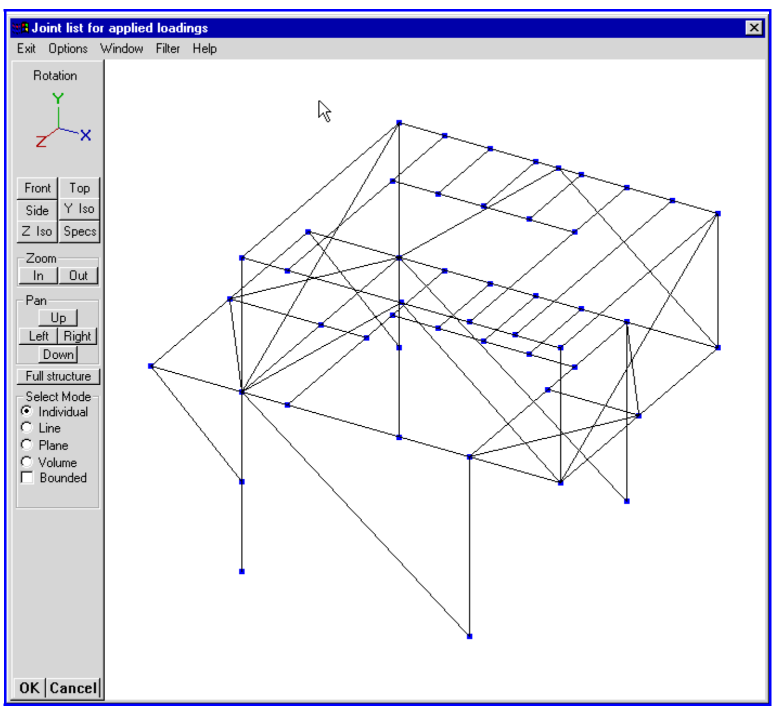


New Graphical Selection Feature

A new feature which has been added to all of the dialogs where users are asked to list joints or members is the ability to graphically select joints or members. An example of this dialog for listing joints to which joint loads are applied is shown below which illustrates the new Graphical selection button:



When the Graphical selection button is selected, a graphical display of the structure is produced as shown in the example below:



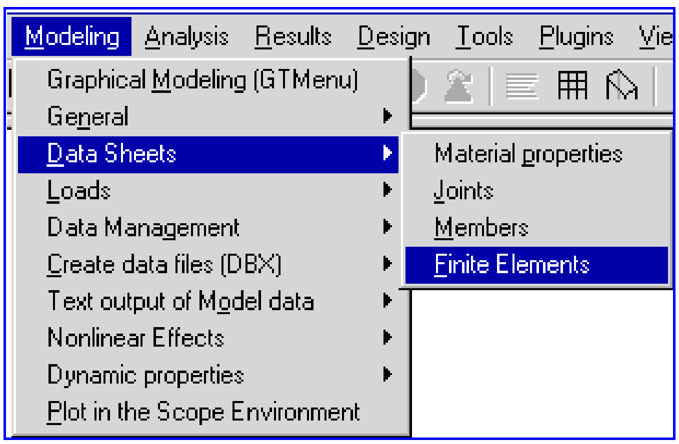
This new feature contains a number of prestored views, allows you to zoom and pan, and contains a variety of selection modes (Individual, Line, Plane, etc.) for selecting joints and members. A filter option allows you to display only certain planes in the structure. Joints and members may be de-selected by clicking on them a second time or clicking the Undo button if available.

Modeling Pulldown

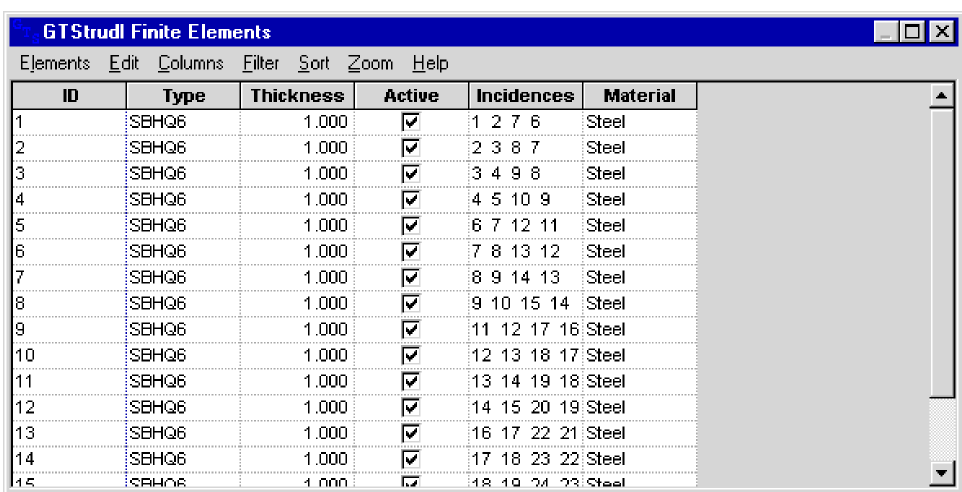
New features have been added to many of the options under the modeling pulldown and will be discussed below:

Data Sheet - Finite element

A new finite element data sheet is now available under the Data sheet option as shown below:

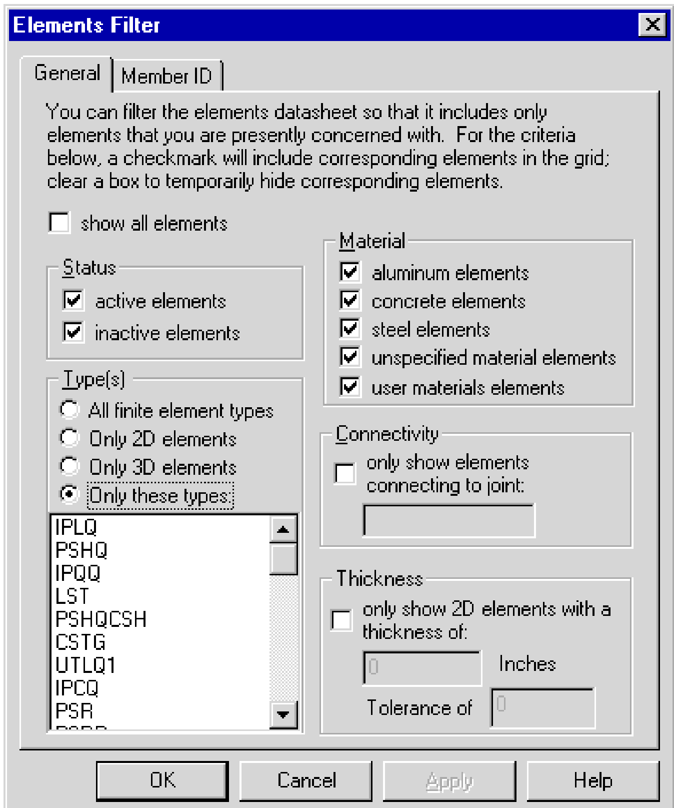


An example of the new finite element data sheet is shown below:



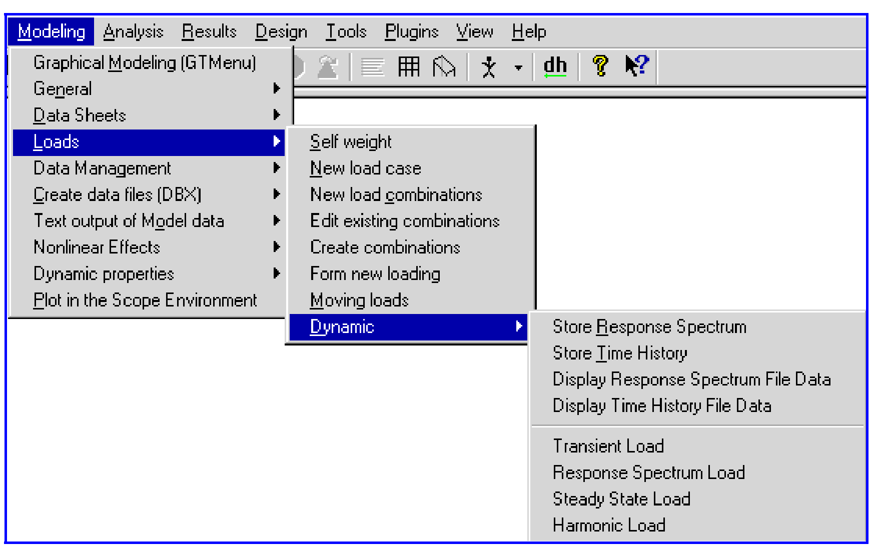
ID	Type	Thickness	Active	Incidences	Material
1	SBHQ6	1.000	☑	1 2 7 6	Steel
2	SBHQ6	1.000	☑	2 3 8 7	Steel
3	SBHQ6	1.000	☑	3 4 9 8	Steel
4	SBHQ6	1.000	☑	4 5 10 9	Steel
5	SBHQ6	1.000	☑	6 7 12 11	Steel
6	SBHQ6	1.000	☑	7 8 13 12	Steel
7	SBHQ6	1.000	☑	8 9 14 13	Steel
8	SBHQ6	1.000	☑	9 10 15 14	Steel
9	SBHQ6	1.000	☑	11 12 17 16	Steel
10	SBHQ6	1.000	☑	12 13 18 17	Steel
11	SBHQ6	1.000	☑	13 14 19 18	Steel
12	SBHQ6	1.000	☑	14 15 20 19	Steel
13	SBHQ6	1.000	☑	16 17 22 21	Steel
14	SBHQ6	1.000	☑	17 18 23 22	Steel
15	SBHQ6	1.000	☑	18 19 24 23	Steel

This data sheet has many of the features available in the Joint and Member Data Sheets. You may Change Selected Data, you may select which columns to have visible in the data sheet and you may filter the data sheet based on a broad range of data such as element types, material, thickness, and connectivity as shown in the Elements Filter dialog below:



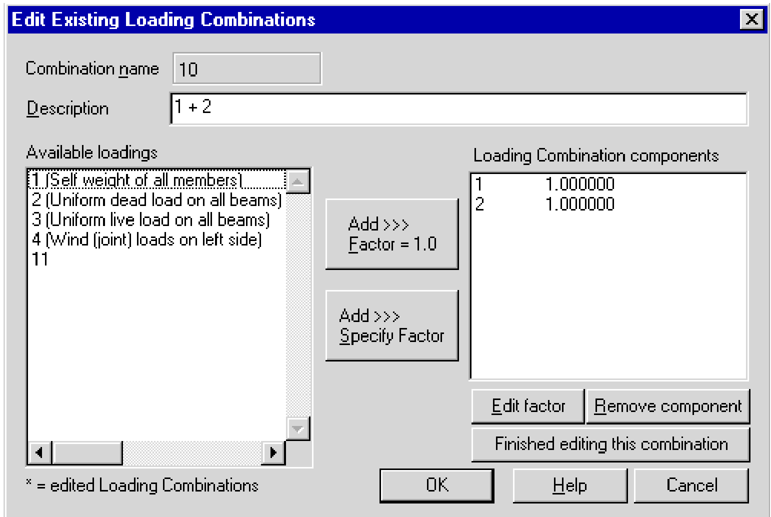
Loads

The modified Loads Pulldown is shown below:



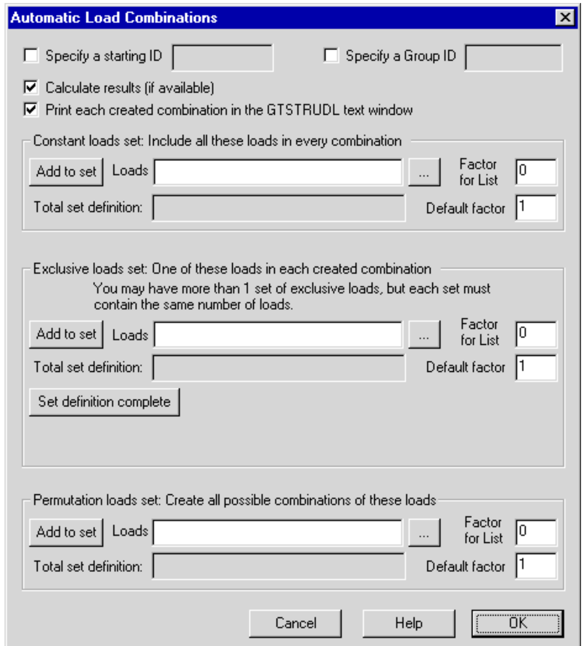
The "Edit existing combinations" and "Create combinations" options are new as well as the "Display Response Spectrum File Data" and "Display Time History File Data" options under the Dynamic pulldown. The Store Response Spectrum and Store Time History dialogs have also been modified to allow for the graphical display of data.

The new Edit Existing Loading Combinations dialog is shown below:

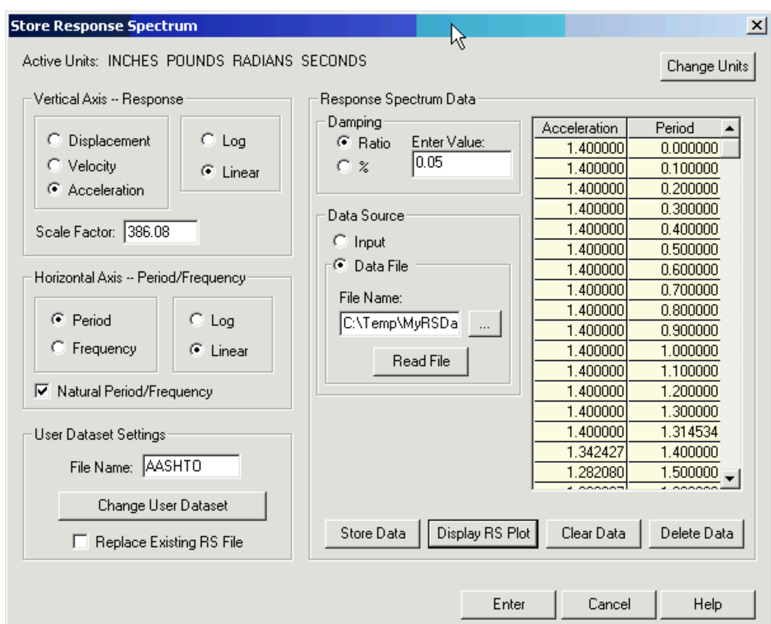


This dialog allows you to add, delete or change factors for the components of existing load combinations.

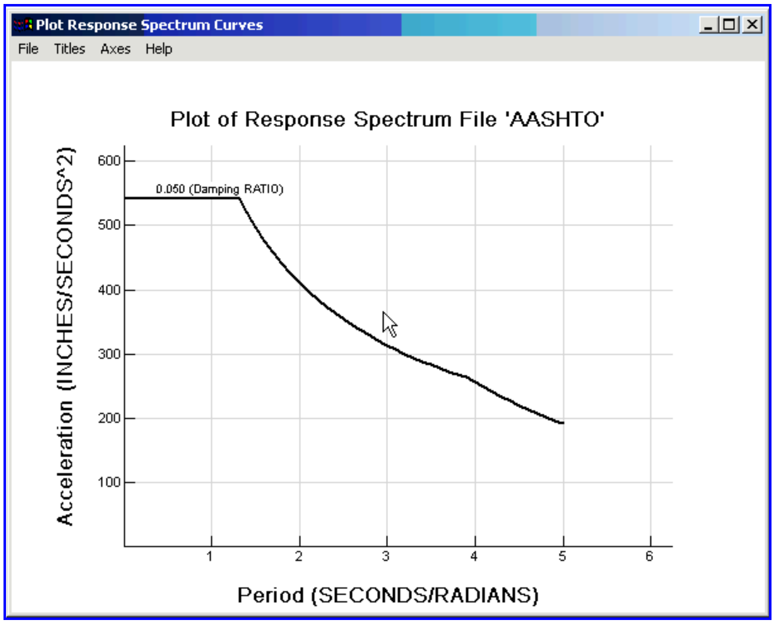
The Create Combinations option allows you to specify sets of loadings to combine one-to-many, one-to-one or in permutations. The one-to-many option is very useful to add dead or permanent loadings to a set of loads created with Moving Load. The one-to-one allows you to easily combine two or more sets of moving loads. The permutation option allows you to create all possible combinations of the specified loadings. The new Automatic Load Combinations dialog is shown on the next page:



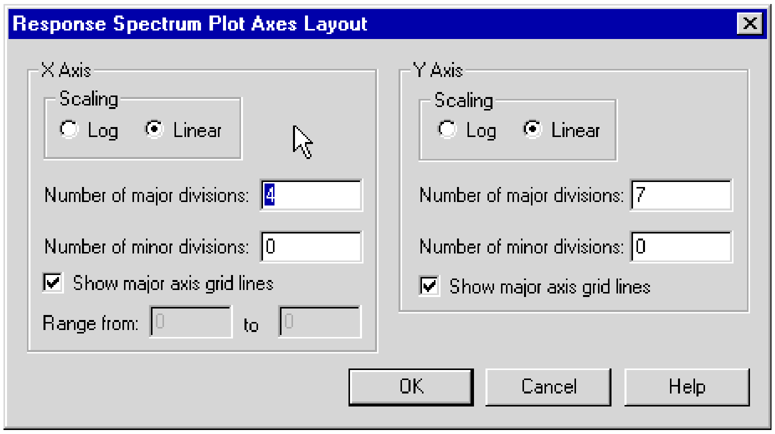
In the Store Response Spectrum and Store Time History dialogs, a new option has been added to graphically display Response Spectrum and Time History data as shown below for the Store Response Spectrum dialog (see the Display RS Plot button):



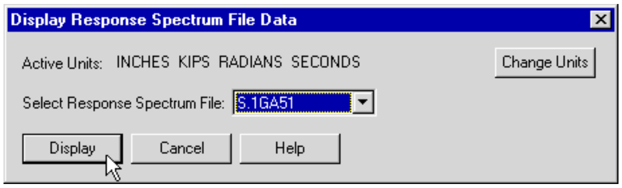
An example of the display produced for the data in the above figure is shown below:



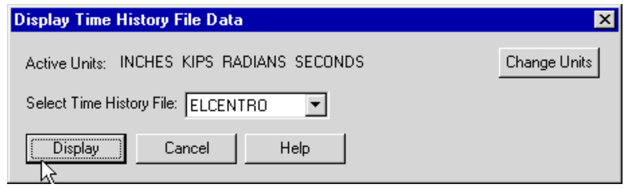
This dialog displays a plot of the selected response spectrum or time history file data and contains the menu items File, Titles, Axes, and Help that allow you to edit and manipulate the plot contents. An example of the options available under the Axes menu item is shown below:



Also available under the Dynamic option pulldown as shown above are options to Display Response Spectrum File Data and Display Time History File Data for response spectrum and time history files which have already been defined. Examples of these two dialogs are shown on the next page:

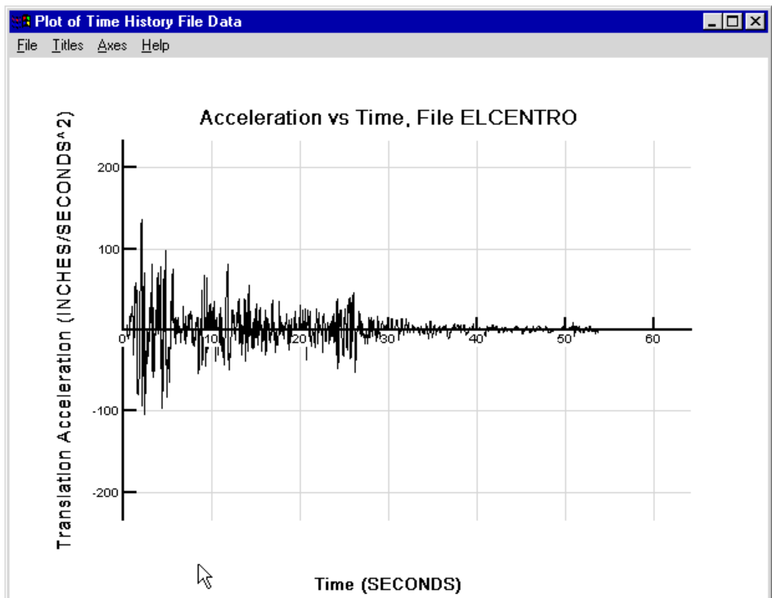


Display Response Spectra File Data



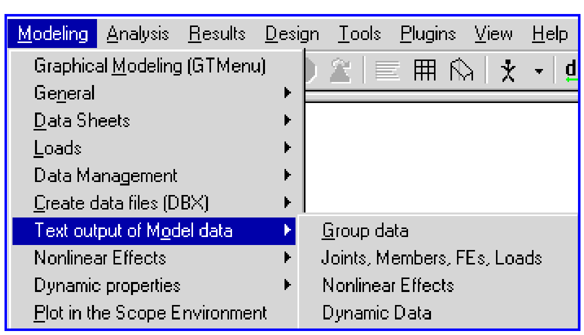
Display Time History File Data

An example of the display of the ELCENTRO time history record is shown below:

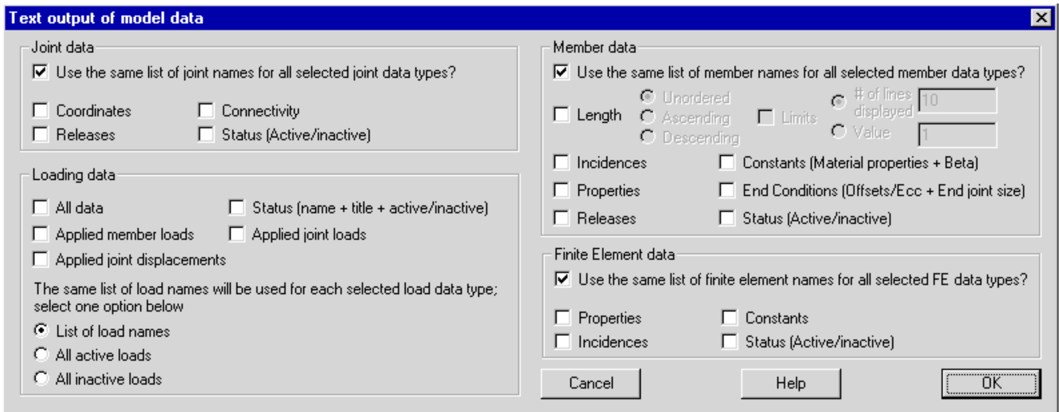


As previously presented, this dialog displays a plot of the selected time history file data and contains the menu items File, Titles, Axes, and Help that allow you to edit and manipulate the plot contents.

The "Text output of Model data" pulldown has been modified as shown below:

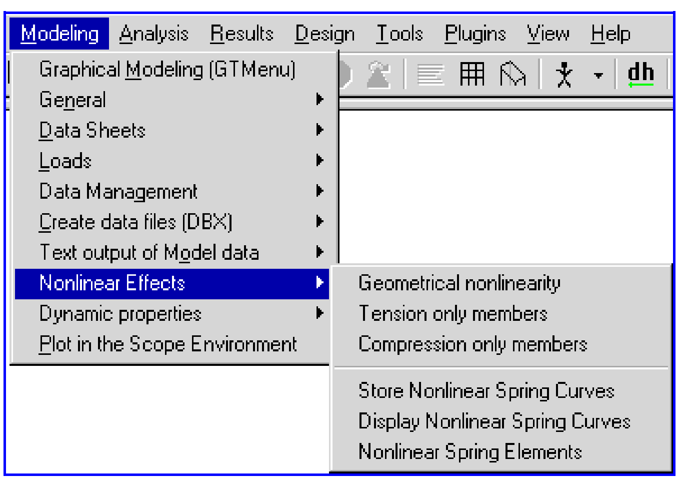


The joints, members, finite elements, and loads text output is now combined into the following dialog:

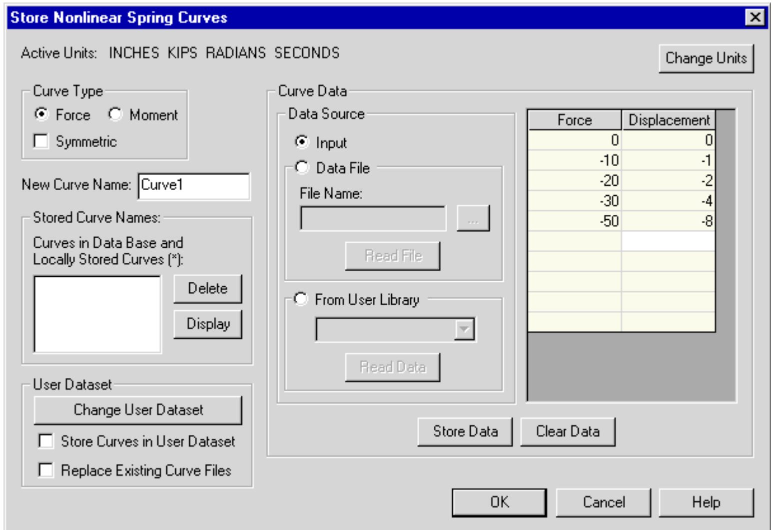


The above dialog is used to generate text output for joints, members, finite elements, and loads from the current GTSTRUDL database. If the "Use the same list..." check box is checked, then only one list per requested component (joints, member and/or finite elements) will be queried. If the box is not checked, then a separate list for each information type will be queried. For example, if the “Use the same list...” box is not checked for Joint data and you have checked the boxes for Coordinates and Releases, then you will need to specify two separate lists of joints, one for Coordinates and one for Releases.

New options to Store Nonlinear Spring Curves, Display Nonlinear Spring Curves, and specify Nonlinear Spring Elements have been added to the Nonlinear Effects pulldown as shown below:

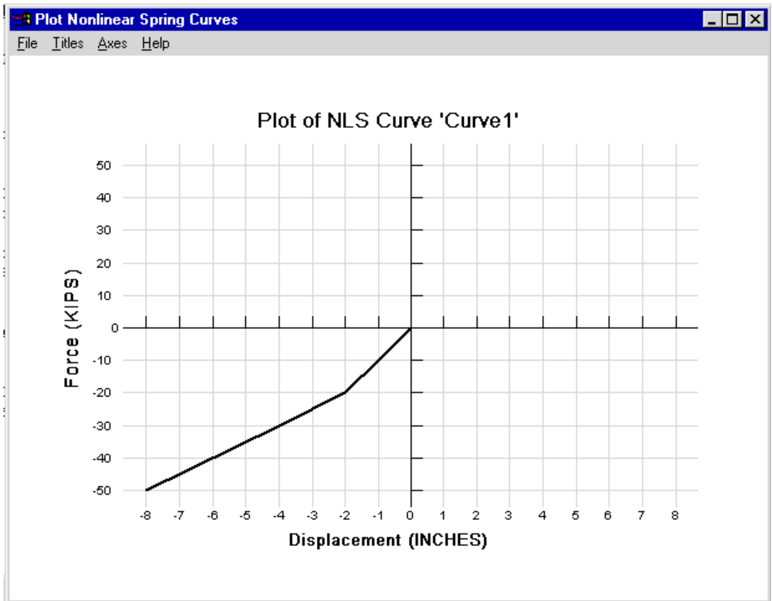


The new Store Nonlinear Spring Curves dialog is shown below:

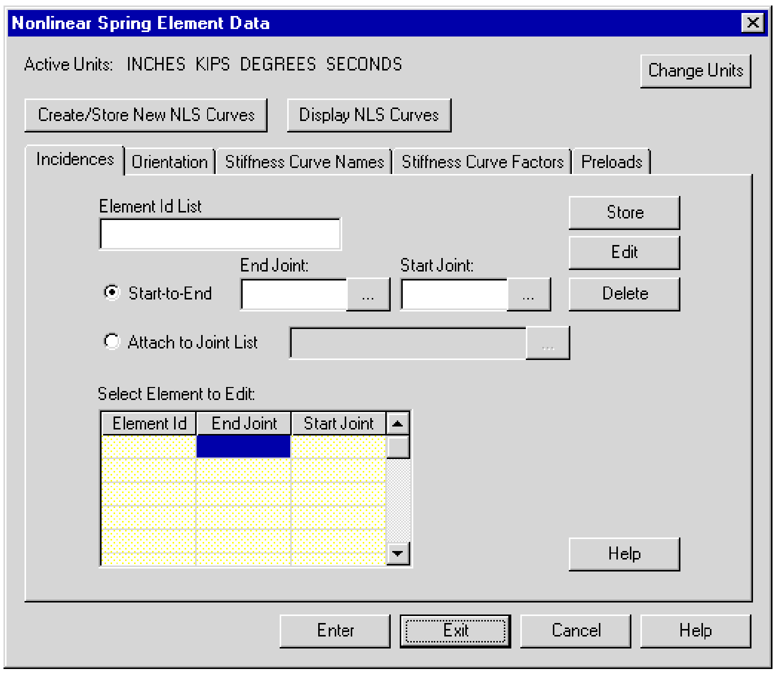


After clicking on the Store Data button, you may then display the nonlinear spring curve data. You may also use the new Display Nonlinear Spring Curves dialog shown below to display nonlinear spring curve data.

An example of the display produced by the Display Nonlinear Spring Curves dialog above and the display produced by clicking on the Display button in the Store Nonlinear Spring Curves dialog at the top of the page are shown on the next page:



The new Nonlinear Spring Element Data dialog shown below is used to create and define nonlinear spring elements in your structural model. Nonlinear spring element data include element incidences, element orientation (local coordinate system), nonlinear spring curves to use for the element stiffness properties, element stiffness factors, and element preload (initial forces).

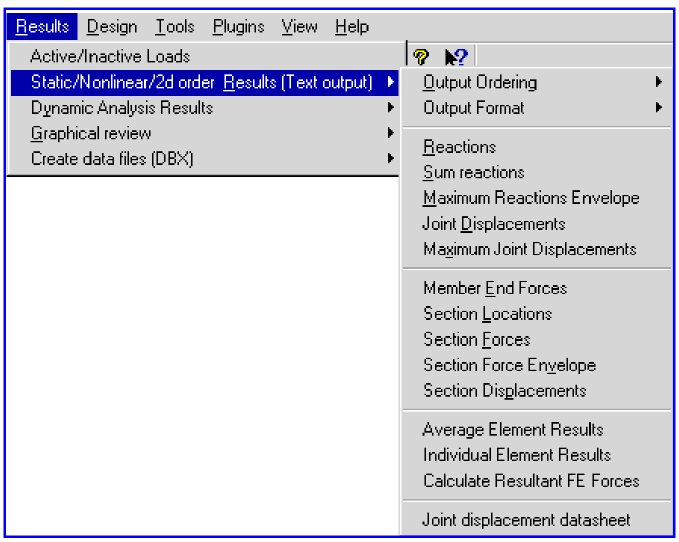


The Nonlinear Spring Element Data dialog is a property page, where the different types of required data are specified by clicking the appropriate tab. The table below summarizes the tab title and the type of data to which it refers:

Tab Title	Data Type Description
Incidences	Element start and end incidences
Orientation	Local element coordinate system orientation
Stiffness Curve Names	Names of nonlinear spring curves for element stiffness properties
Stiffness Curve Factors	Element stiffness force or moment scaling factors
Preloads	Element initial loads

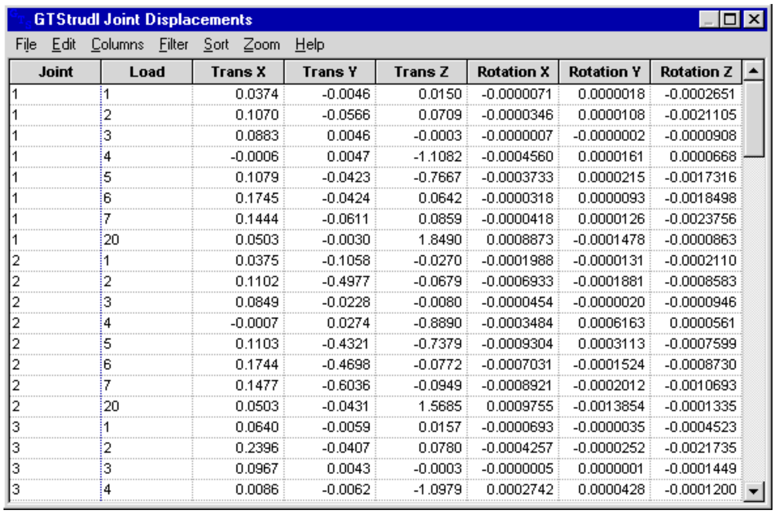
Results Pulldown

The modified Static/Nonlinear Results text output pulldown is shown below:



The Maximum Reactions Envelope option replaces the Maximum Reactions option which was available in previous versions. The Maximum Reactions Envelope option will output an envelope (algebraic max and min) of the reactions. Previously, the Maximum Reactions option would only output the reactions with the largest absolute value.

The new Joint Displacement datasheet is used to review and arrange results from a previous analysis - no editing of the display is allowed. Only active joints and loads are included in the display. An example of the Joint Displacement datasheet is shown below:



GTStrudl Joint Displacements

File Edit Columns Filter Sort Zoom Help

Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y	Rotation Z
1	1	0.0374	-0.0046	0.0150	-0.0000071	0.0000018	-0.0002651
1	2	0.1070	-0.0566	0.0709	-0.0000346	0.0000108	-0.0021105
1	3	0.0883	0.0046	-0.0003	-0.0000007	-0.0000002	-0.0000908
1	4	-0.0006	0.0047	-1.1082	-0.0004560	0.0000161	0.0000668
1	5	0.1079	-0.0423	-0.7667	-0.0003733	0.0000215	-0.0017316
1	6	0.1745	-0.0424	0.0642	-0.0000318	0.0000093	-0.0018498
1	7	0.1444	-0.0611	0.0859	-0.0000418	0.0000126	-0.0023756
1	20	0.0503	-0.0030	1.8490	0.0008873	-0.0001478	-0.0000863
2	1	0.0375	-0.1058	-0.0270	-0.0001988	-0.0000131	-0.0002110
2	2	0.1102	-0.4977	-0.0679	-0.0006933	-0.0001881	-0.0008583
2	3	0.0849	-0.0228	-0.0080	-0.0000454	-0.0000020	-0.0000946
2	4	-0.0007	0.0274	-0.8890	-0.0003484	0.0006163	0.0000561
2	5	0.1103	-0.4321	-0.7379	-0.0009304	0.0003113	-0.0007599
2	6	0.1744	-0.4698	-0.0772	-0.0007031	-0.0001524	-0.0008730
2	7	0.1477	-0.6036	-0.0949	-0.0008921	-0.0002012	-0.0010693
2	20	0.0503	-0.0431	1.5685	0.0009755	-0.0013854	-0.0001335
3	1	0.0640	-0.0059	0.0157	-0.0000693	-0.0000035	-0.0004523
3	2	0.2396	-0.0407	0.0780	-0.0004257	-0.0000252	-0.0021735
3	3	0.0967	0.0043	-0.0003	-0.0000005	0.0000001	-0.0001449
3	4	0.0086	-0.0062	-1.0979	0.0002742	0.0000428	-0.0001200

After the data is arranged in a convenient manner by using the Columns, Filter and Sort selections, you can request a text copy to be printed in the GTSTRUDL text output so you can highlight-and-print or copy-and-paste to another program. You can also put selected data on the Windows clipboard and then paste into another program, such as a spreadsheet. The various options available in this datasheet are described below:

File

> "Write display to text window" prints the display in the GTSTRUDL output text window, respecting ordering, filtering and display columns. A header is provided to indicate ordering and filtering values plus column labels.
> Define group puts the selected joint IDs from selected rows into a group. You will be prompted for a group name and optional title.

Edit

> Only the Copy option is supported. The Copy option copies the selected (highlighted) portion of the datasheet to the Windows clipboard.

Columns

> Use this menu to control how many columns of data are displayed. The X, Y and Z translations and X, Y and Z rotations may be displayed in any combination.

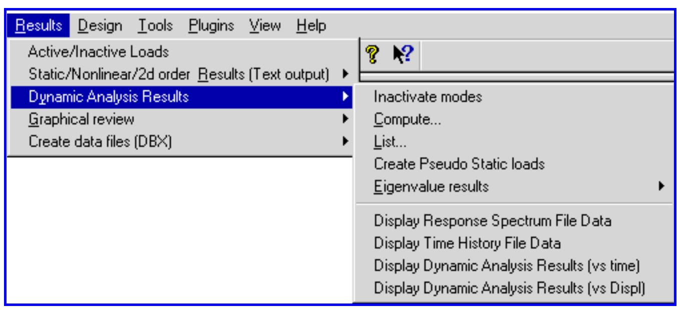
Filter

Use this menu (through the 'Filter Settings...' selection) to determine which results (Joint/Load pair) are displayed.
Show All removes all the filter settings.

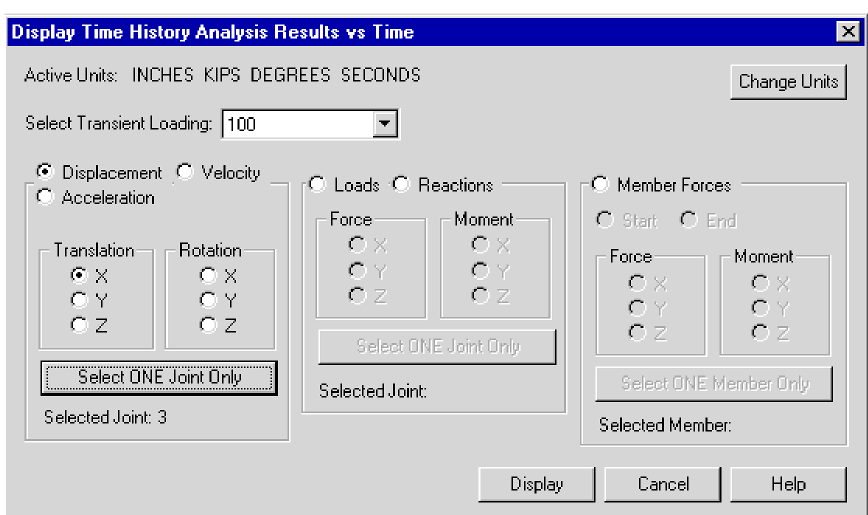
Sort

Use this menu to order the results in either ascending or descending order based on the selected data: Joint ID, Load ID or value (Trans X, Y or Z, Rot X, Y or Z). Also, values may be sorted algebraically (where -5 is less than 2) or by magnitude (where -5 is larger than 2).

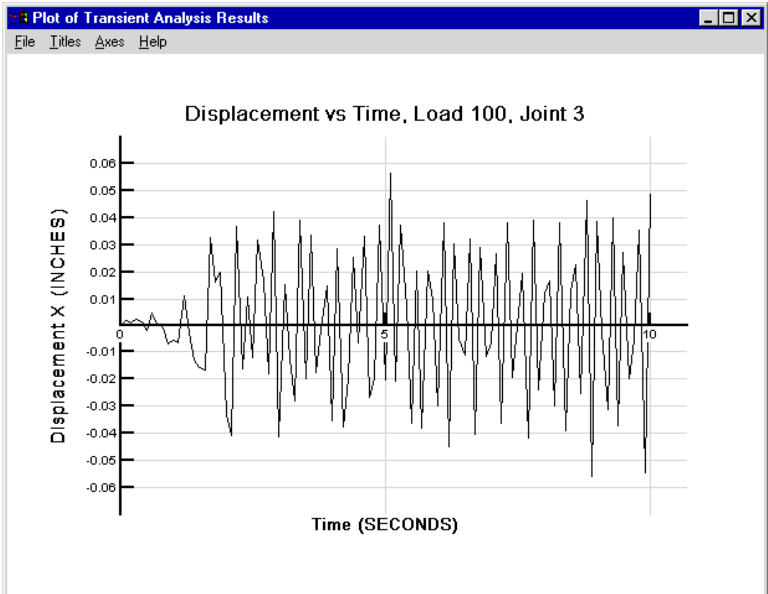
The modified Dynamic Analysis Results pulldown is shown below:



The Display Response Spectrum File data and Display Time History File Data dialogs have been discussed previously under the Modeling Pulldown. The new Display Dynamic Analysis Results vs Time dialog is shown below:

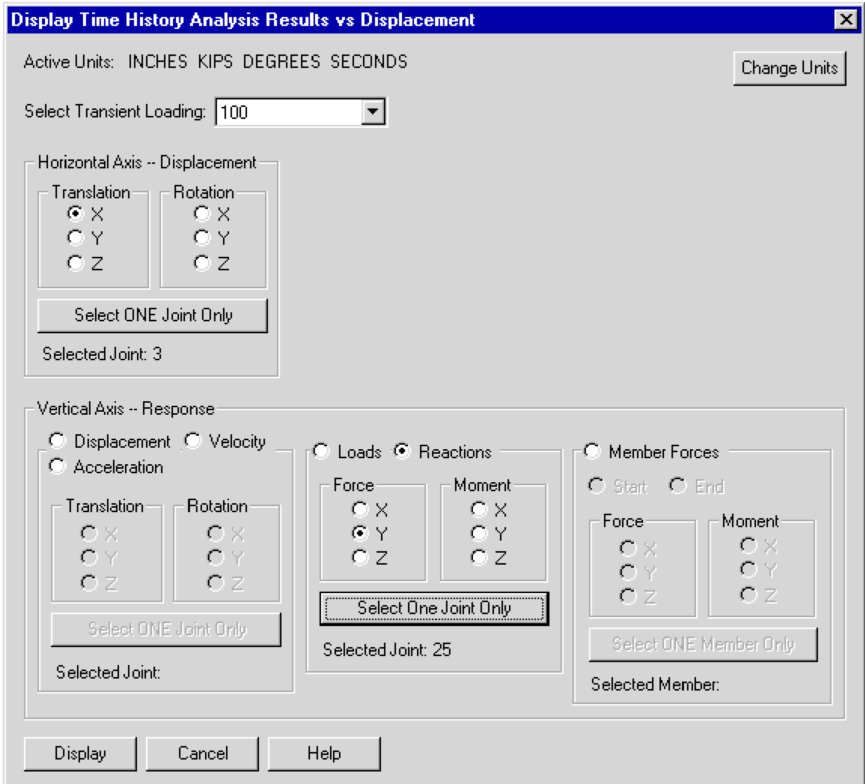


An example of the graphical display of time history joint displacement results is shown below:



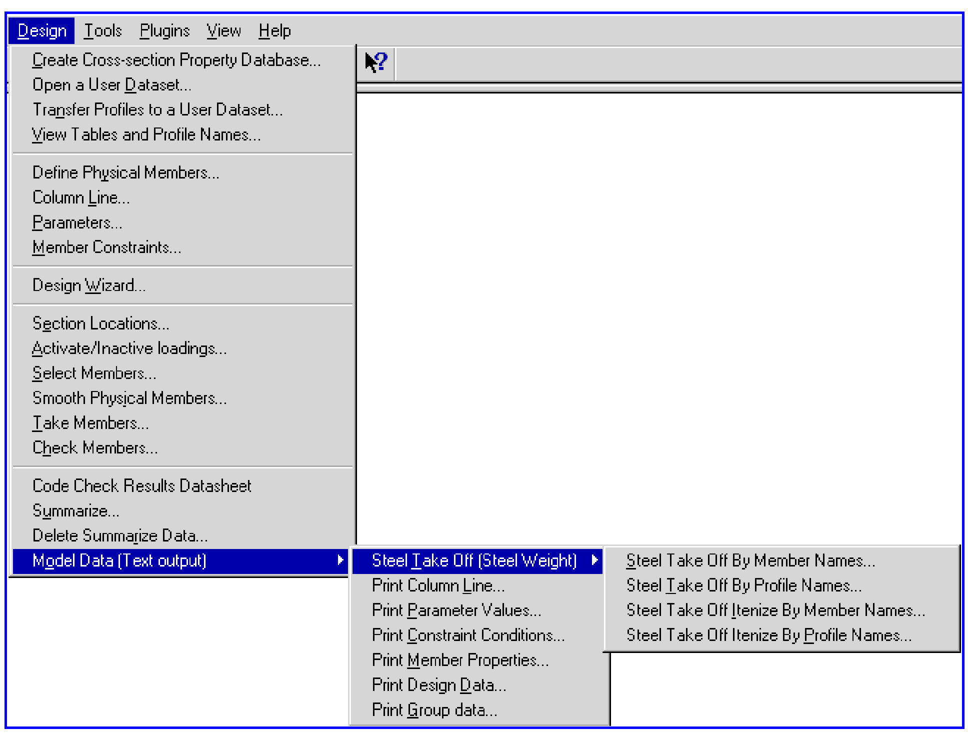
The Display Time History Analysis Results vs Time dialog is used to display a plot of selected dynamic transient analysis results vs time. The results categories include: displacements, velocities, accelerations, joint loads, support reactions, and member forces. These results can be calculated by either a linear or nonlinear dynamic analysis for transient loading conditions.

The new Display Time History Analysis Results vs Displacement dialog shown on the next page is used to display a plot of selected dynamic transient analysis results vs displacement. The horizontal axis of the resulting plot represents displacement at a selected joint. The vertical axis of the plot represents response at joints or members selected from the following categories: displacements, velocities, accelerations, joint loads, support reactions, and member forces. These results can be calculated by either a linear or nonlinear dynamic analysis for transient loading conditions; however, the dialog is most useful for nonlinear dynamic analysis results for structures that include the friction damper nonlinear effect for plane and space truss members.

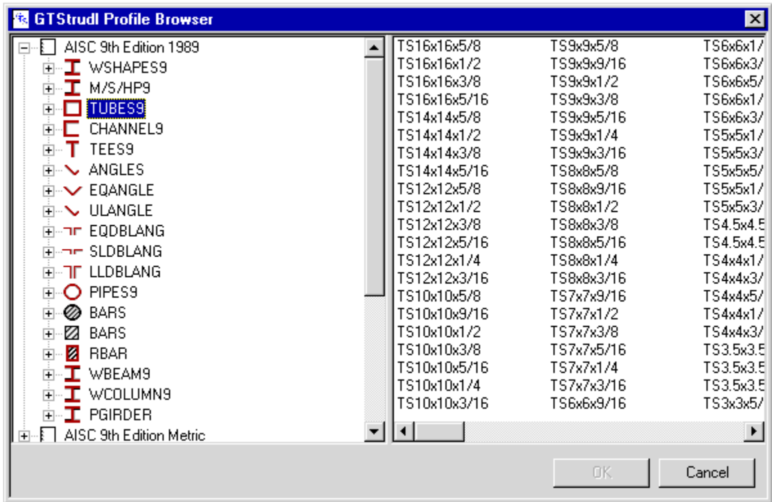


Design Pulldown

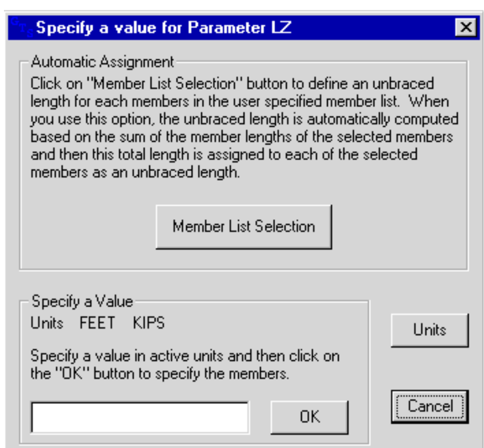
The modified Design pulldown is shown below:



The View Tables and Profile Names option is new and allows you to browse existing tables in order to locate a steel profile. The new Profile Browser dialog is shown below:

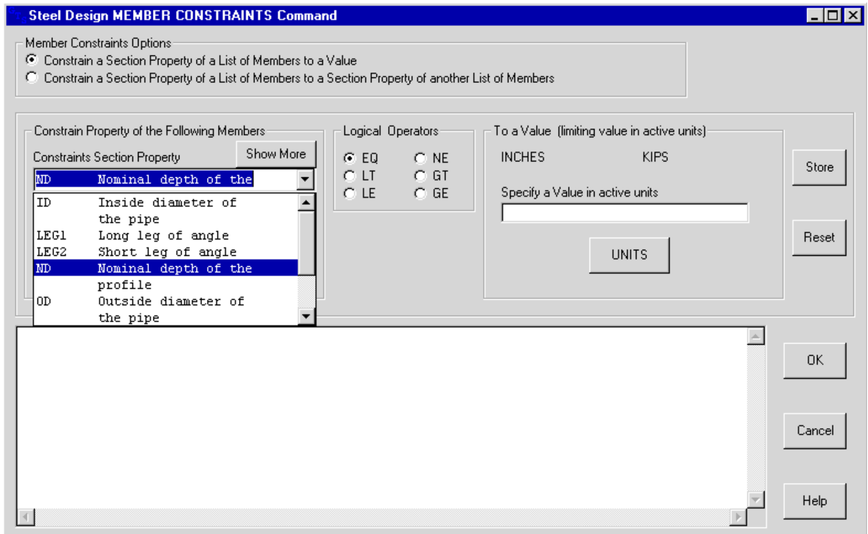


The unbraced length (LY, LZ, UNLCF, etc.) of a member or physical member can be now graphically assigned by the parameter dialog. The new Automatic Assignment option allows you to assign the physical member unbraced length by graphically selecting the members that are part of the physical member. When you click on the members that are part of a physical member, the unbraced length is automatically computed based on the sum of the member lengths of the selected members and then this total length is assigned to each of the selected members as an unbraced length. An example of the dialog to assign the LZ unbraced length is shown below:

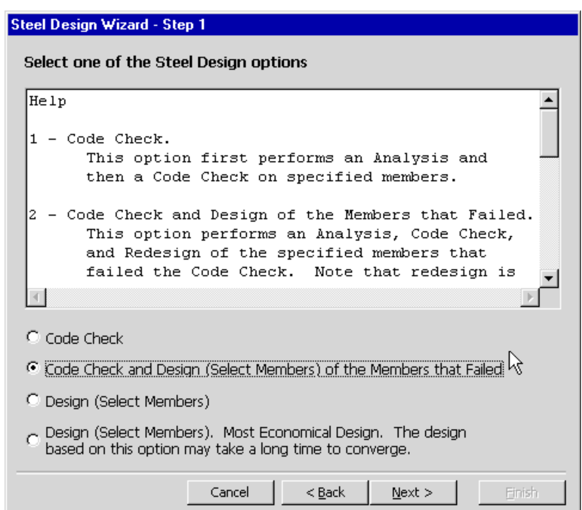


If you select the “Member List Selection” button, you then have the option to either give a list of members or graphically select the members that will be used to compute the unbraced length.

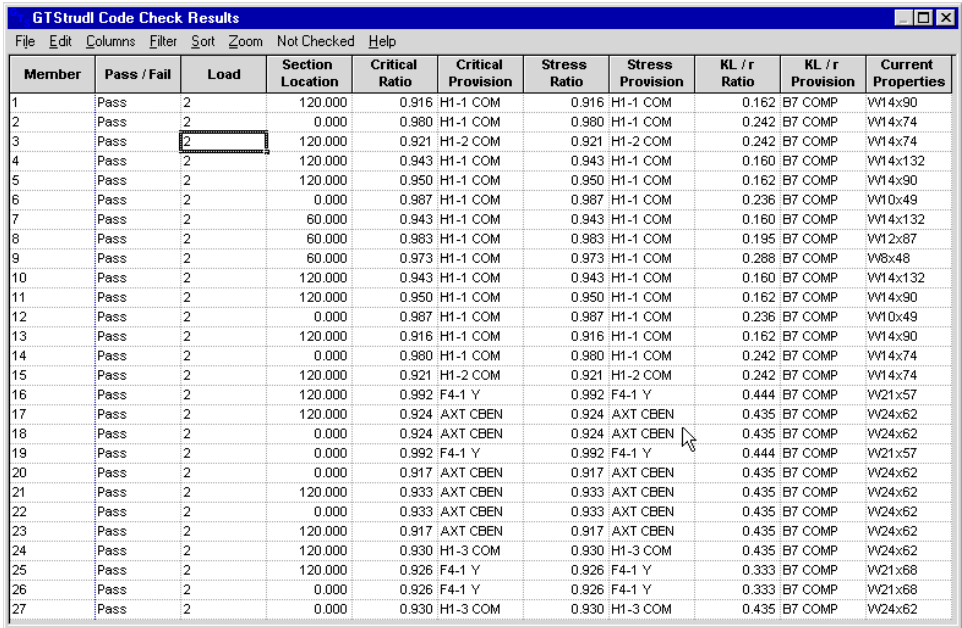
The Member Constraints option now has a feature to show what we consider to be the most commonly used constraints as shown below. To see all of the available constraints, click on the Show More button in the dialog.



The Design Wizard option has a new feature to code check and automatically design the members that failed the code check as shown below:

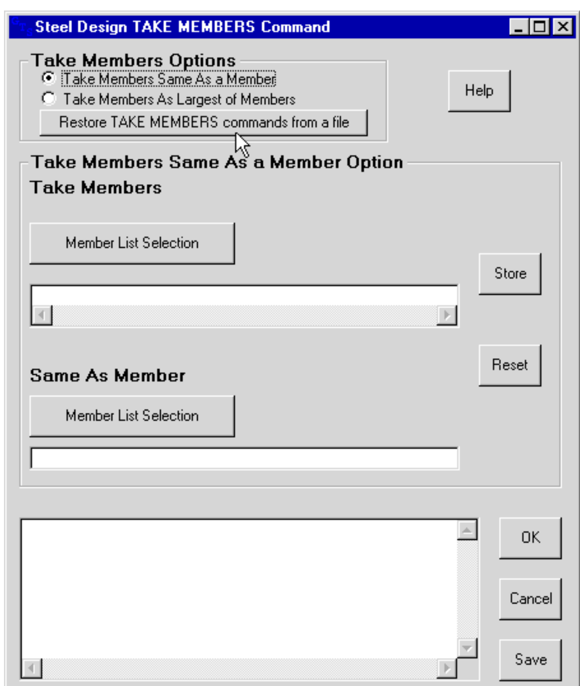


The Take Members option can now read the TAKE commands from a file as shown in the modified dialog below:



Member	Pass / Fail	Load	Section Location	Critical Ratio	Critical Provision	Stress Ratio	Stress Provision	KL / r Ratio	KL / r Provision	Current Properties
1	Pass	2	120.000	0.916	H1-1 COM	0.916	H1-1 COM	0.162	B7 COMP	W14x90
2	Pass	2	0.000	0.980	H1-1 COM	0.980	H1-1 COM	0.242	B7 COMP	W14x74
3	Pass	2	120.000	0.921	H1-2 COM	0.921	H1-2 COM	0.242	B7 COMP	W14x74
4	Pass	2	120.000	0.943	H1-1 COM	0.943	H1-1 COM	0.160	B7 COMP	W14x132
5	Pass	2	120.000	0.950	H1-1 COM	0.950	H1-1 COM	0.162	B7 COMP	W14x90
6	Pass	2	0.000	0.987	H1-1 COM	0.987	H1-1 COM	0.236	B7 COMP	W10x49
7	Pass	2	60.000	0.943	H1-1 COM	0.943	H1-1 COM	0.160	B7 COMP	W14x132
8	Pass	2	60.000	0.983	H1-1 COM	0.983	H1-1 COM	0.195	B7 COMP	W12x87
9	Pass	2	60.000	0.973	H1-1 COM	0.973	H1-1 COM	0.288	B7 COMP	W8x48
10	Pass	2	120.000	0.943	H1-1 COM	0.943	H1-1 COM	0.160	B7 COMP	W14x132
11	Pass	2	120.000	0.950	H1-1 COM	0.950	H1-1 COM	0.162	B7 COMP	W14x90
12	Pass	2	0.000	0.987	H1-1 COM	0.987	H1-1 COM	0.236	B7 COMP	W10x49
13	Pass	2	120.000	0.916	H1-1 COM	0.916	H1-1 COM	0.162	B7 COMP	W14x90
14	Pass	2	0.000	0.980	H1-1 COM	0.980	H1-1 COM	0.242	B7 COMP	W14x74
15	Pass	2	120.000	0.921	H1-2 COM	0.921	H1-2 COM	0.242	B7 COMP	W14x74
16	Pass	2	120.000	0.992	F4-1 Y	0.992	F4-1 Y	0.444	B7 COMP	W21x57
17	Pass	2	120.000	0.924	AXT CBEN	0.924	AXT CBEN	0.435	B7 COMP	W24x62
18	Pass	2	0.000	0.924	AXT CBEN	0.924	AXT CBEN	0.435	B7 COMP	W24x62
19	Pass	2	0.000	0.992	F4-1 Y	0.992	F4-1 Y	0.444	B7 COMP	W21x57
20	Pass	2	0.000	0.917	AXT CBEN	0.917	AXT CBEN	0.435	B7 COMP	W24x62
21	Pass	2	120.000	0.933	AXT CBEN	0.933	AXT CBEN	0.435	B7 COMP	W24x62
22	Pass	2	0.000	0.933	AXT CBEN	0.933	AXT CBEN	0.435	B7 COMP	W24x62
23	Pass	2	120.000	0.917	AXT CBEN	0.917	AXT CBEN	0.435	B7 COMP	W24x62
24	Pass	2	120.000	0.930	H1-3 COM	0.930	H1-3 COM	0.435	B7 COMP	W24x62
25	Pass	2	120.000	0.926	F4-1 Y	0.926	F4-1 Y	0.333	B7 COMP	W21x68
26	Pass	2	0.000	0.926	F4-1 Y	0.926	F4-1 Y	0.333	B7 COMP	W21x68
27	Pass	2	0.000	0.930	H1-3 COM	0.930	H1-3 COM	0.435	B7 COMP	W24x62

A new Code Check Results Datasheet has been implemented. This datasheet includes all of the information that is available for graphical display in GTMenu including the pass/fail status, critical load and section location, critical ratio and provision, stress ratio and provision, kl/r ratio and provision, and the current properties. An example of this new datasheet is shown below:

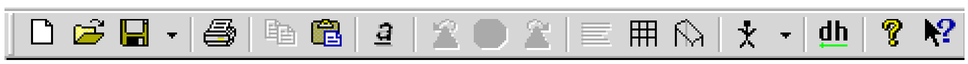


The functionality of this new datasheet is similar to that of the Joint Displacement Datasheet previously presented. The Sort and Filter options are particularly useful. The "Not Checked" option will identify which members have not yet been code checked.

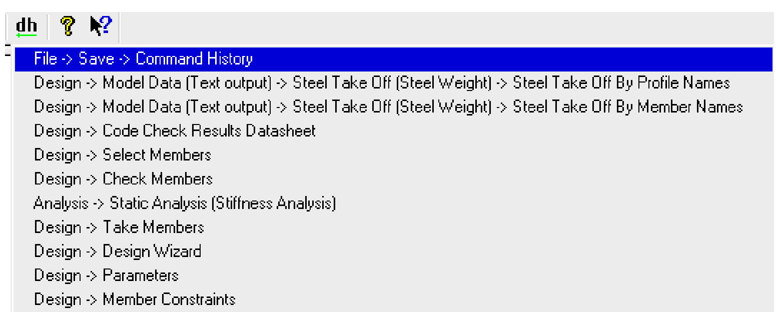
The Steel Take Off pulldown under the Model Data option is new in Version 26. Previously, some of these features were available in previous versions. New in Version 26 is the ability to perform the takeoff by profile names.

Toolbar

The modified Toolbar is shown below with the new Dialog History button,  :



The Dialog History button offers a menu of your most recent menu picks as a quick way to return to a previous dialog or command generation menu pick. The most recent selection is at the top of the menu. The menu is rebuilt each time you run GTSTRUDL, so until you make a menu pick the Dialog History will be empty. An example of the Dialog History is shown in the figure below:



The "S>" indicates a submenu or selection, so 'Check Members' is a selection from the 'Design' menu.

2.3 DBX

1. A new UNREGISTERED option has been added to the WRITE command shown below:

    WRITE UNREGISTERED data_type (‘file_name’) list

    UNREGISTERED means that the created DBX file information will not be entered into the DBX directory information. In addition, a FILE SPECS command does not need to precede a WRITE UNREGISTERED command. UNREGISTERED files are always written in ASCII80 format, in REPLACE mode.

    This means that when the UNREGISTERED option is used, the WRITE DIRECTORY command will not print the results (date written, number of items, etc.) for the unregistered file. It also means that you can give a new file name in the write command and have it created and filled with the requested data without issuing a previous FILES SPECS command. If no file name is given, the standard default name (“STDBX” + class data number) will be created. Since REPLACE mode is the standard, if ‘file_name’ already exists, it will be overwritten without warning. The current default FILE SPECS modes (ASCII, ASCII80, BINARY, SEQUENTIAL, etc.) are ignored - unregistered files are always written in ASCII80 mode.

2.4 Dynamics

1. A JOINT list option has been added to the LIST DYNAMIC MASS SUMMARY command. This will enable the computation of mass properties for substructures such as floors by specifying only the joints that are contained within the boundaries of the substructure.

2. Direct input of the global system damping matrix now allows only the rows having non-zero damping coefficient entries to be specified. In all earlier versions, all rows, even those having all 0.0 coefficient entries, up to and including the last row having at least one non-zero entry, had to be specified.

3. Damping matrices are now condensed in physical analysis (direct integration), regardless of source, if the global stiffness matrix is assembled. If the global stiffness matrix is created by input, the damping matrix cannot be condensed.

2.5 Finite Element

1. The CALCULATE AVERAGE VON MISES command has been modified in order to be able to compare directly with the yield stress. Previously, the comparison had to be made with the yield stress divided by the square root of 3. Examples of the modified command are shown on the next page:

   CALCULATE AVERAGE VON MISES COMPARE WITH YIELD TOP SURFACE

   To COMPARE WITH THE YIELD is now the default option that will be executed as in the command below:

   CALCULATE AVERAGE VON MISES TOP

   To compare with the yield divided by the square root of 3, which was the only option available in versions prior to Version 26, issue the following command:

   CALCULATE AVERAGE VON MISES COMPARE WITH YIELD/SQRT3 TOP

   The modified command is documented in Section 2.3.7.2 of Volume 3 of the GTSTRUDL Reference Manual.

2.6 General

1. The ability to specify member properties for I-shapes and plate girders by giving only the cross-section dimensions in the MEMBER PROPERTIES command has been brought to release status. An example of using this feature is shown below:

```
MEMBER PROPERTIES
1  TO  9  BY  2  I-SHAPE TOTAL DEPTH 24.5   WEB THICKNESS  -
   0.5  FLANGE WIDTH  10.75  FLANGE THICKNESS  0.875
```

   The enhanced MEMBER PROPERTIES command is documented in Section 2.1.9.2 of Volume 1 of the GTSTRUDL Reference Manual.

2. The modification to the PRINT command to output computed design properties and definitions for I-shapes, plate girder, and pipes specified using the cross-section specs option of the MEMBER PROPERTIES command is now a released feature. The new options on the PRINT command are shown on the next page:

PRINT MEMBER PROPERTIES FOR DESIGN
PRINT MEMBER PROPERTIES FOR DESIGN WITH DEFINITIONS

The revised PRINT command is presented in Section 2.1.14.2 of Volume 1 of the GTSTRUDL Reference Manual.

3. The CREATE AUTOMATIC LOAD COMBINATION with multiple exclusive list option has been brought to a release status. If multiple exclusive lists are given, one load from each list will be included in any single load combination. This feature is particularly useful when moving loads have been created for each side of a crane or each side of a vehicle. An example of this new option is shown below:

Example: Multiple EXCLUSIVE lists plus CONSTANT

```
CREATE AUTOMATIC LOAD COMBINATION START ID  101
    ALWAYS INCLUDE              'Dead' 1.0
    EXCLUSIVE                   11  TO  14
    EXCLUSIVE                   21  22  23  24
```

This command set generates these four load combinations. Note that the EXCLUSIVE lists '11 TO 14' and '21 22 23 24' both contain four loads, since multiple EXCLUSIVE list must be the same length.

```
LOAD COMB  101  SPECS  'Dead'  1.0  11  1.0  21  1.0
LOAD COMB  102  SPECS  'Dead'  1.0  12  1.0  22  1.0
LOAD COMB  103  SPECS  'Dead'  1.0  13  1.0  23  1.0
LOAD COMB  104  SPECS  'Dead'  1.0  14  1.0  24  1.0
```

The revised CREATE AUTOMATIC LOAD COMBINATION command is documented in Section 2.1.11.3.6 of Volume 1 of the GTSTRUDL Reference Manual.

4. The SELF WEIGHT command has been modified to generate loadings only on ALL ACTIVE members or JOINTS. Previously, the command generated loadings on all active and inactive members or joints. The revised command is documented in Section 2.1.11.3.1.1 of Volume 1 of the GTSTRUDL Reference Manual.

5. A new option has been added to the PRINT MEMBER LENGTH command. PRINT MEMBER LENGTH SORTED has been added to allow you to sort and limit the display of all members or a subset of members. The command is shown below:

$$\underline{\text{PRI}}\text{NT}\ \ \underline{\text{MEM}}\text{BER}\ \ \underline{\text{LEN}}\text{GTH}\ \ \underline{\text{SOR}}\text{TED}\ \left(\left\{\begin{array}{l} \rightarrow \underline{\text{ASC}}\text{ENDING} \\ \underline{\text{SMA}}\text{LLEST}\ (\underline{\text{TO}})\ (\underline{\text{LAR}}\text{GEST}) \\ \underline{\text{DES}}\text{CENDING} \\ \underline{\text{LAR}}\text{GEST}\ (\underline{\text{TO}})\ (\underline{\text{SMA}}\text{LLEST}) \end{array}\right\}\right)\ -$$

$$(\ \underline{\text{LIM}}\text{IT}\ \left\{\begin{array}{c} \text{i} \\ \text{v} \end{array}\right\})\ \ (\underline{\text{MEM}}\text{BERS list})$$

where,

ASCENDING, SMALLEST
= these are equivalent, asking for sorting to be performed starting with the shortest member in the list. This is the default order.

DESCENDING, LARGEST
= these are equivalent, asking for sorting to be performed starting with the longest member in the list. This order must be requested.

LIMIT i
= limit the display to no more than 'i' members in the requested order.

LIMIT v
= stop printing members when this limit is violated. For smallest-to-largest order, 'v' is the largest length to print. For largest-to-smallest order, 'v' is the smallest length to print. The number of members displayed is not restricted and you cannot limit size and number simultaneously.

MEMBERS list
= you may give a member list to which to restrict the sort and limit operations. By default all existing members are considered.

Examples:

PRINT MEMBER LENGTHS SORTED
Print all members with length, starting with the shortest member.

PRINT MEMBER LENGTHS SORTED LIMIT 10
Print the 10 shortest members

PRINT MEMBER LENGTHS SORTED LIMIT 10. $ Note the decimal!

Print all members whose length is 10.0 current length units or less.

The revised command is documented in Section 2.1.14.2.1 of Volume 1 of the GTSTRUDL Reference Manual.

6. The LIST MAXIMUM REACTIONS command has been extended to include an ENVELOPE option. This option prints the algebraic maximum and minimum forces and moments at each support in the joint list (all joints by default). In addition, the summary will include the overall maximum and minimum forces and moments, along with the joint where they occurred and the load that caused them.

$$\underline{\text{LIST}}\ \underline{\text{MAX}}\text{IMUM}\ \underline{\text{REA}}\text{CTIONS}\ (\underline{\text{ENV}}\text{ELOPE})\ \ (\underline{\text{F}}\text{OR})\ —$$

$$\left(\underline{\text{JOI}}\text{NTS} \begin{Bmatrix} \rightarrow \underline{\text{ALL}} \\ \text{list}_1 \end{Bmatrix} \right) (\ \underline{\text{A}}\text{ND}\ ) \left(\underline{\text{LOA}}\text{DS} \begin{Bmatrix} \rightarrow \underline{\text{ALL}} \\ \underline{\text{ACT}}\text{IVE}\ (\ \underline{\text{O}}\text{NLY}\ ) \\ \text{list}_2 \end{Bmatrix} \right)$$

Examples:

LIST MAXIMUM REACTIONS ENVELOPE

Print the max and min global X, Y, and Z forces and moments for all support joints, considering all loads.

LIST MAXIMUM REACTIONS ENVELOPE FOR JOINTS 1 TO 10 AND LOADS -
5 TO 12

Print the max and min global X, Y, and Z forces and moments for supports in the group of joints 1 to 10 inclusive, considering only loadings 5 to 12 inclusive.

The revised command is described in Section 2.1.14.4.2 of Volume 1 of the GTSTRUDL Reference Manual.

7. The file containing elastic connections, the elastic.con file, now stays in the GTSTRUDL installation directory. Previously, when users changed the location of the password file, a MEMBER RELEASE command which specified elastic connections from a file could not find the elastic.con file, unless elastic.con was moved to the new password file directory.

8. The OUTPUT LONG NAME (Section 2.1.14.3, Volume 1) is now the default naming convention used for steel profiles. This will allow the steel table profile long names to be displayed when the specified commands print steel table profile names (i.e., Print Member Properties, Steel Takeoff, Check, Select, Summarize, etc.). To return to the same naming convention as in previous versions of GTSTRUDL, you may specify the OUTPUT SHORT

NAME command (Section 2.1.14.3, Volume 1). Users should note that GTMenu still uses the “Short Name” for steel profiles.

9. The STEEL TAKE OFF command has been modified to output the results by profile name. The modified command is shown below:

$$\underline{\mathrm{STEE}}\mathrm{L}\ (\underline{\mathrm{TAKE}}\ \ \underline{\mathrm{O}}\mathrm{FF})\left\{\begin{array}{l}\rightarrow\underline{\mathrm{ALL}}\ (\mathrm{active/inactive})\ (\underline{\mathrm{MEM}}\mathrm{BERS})\\ \underline{\mathrm{MEM}}\mathrm{BERS\ list}\end{array}\right\}\ -$$

$$\left(\left\{\begin{array}{l}\underline{\mathrm{BY}}\ \underline{\mathrm{PRO}}\mathrm{FILE}\ (\underline{\mathrm{NAM}}\mathrm{ES})\\ \underline{\mathrm{ITE}}\mathrm{MIZE}\ \left(\underline{\mathrm{BY}}\left\{\begin{array}{l}\rightarrow\underline{\mathrm{MEM}}\mathrm{BER}\ (\underline{\mathrm{NAM}}\mathrm{ES})\\ \underline{\mathrm{PRO}}\mathrm{FILE}\ (\underline{\mathrm{NAM}}\mathrm{ES})\end{array}\right\}\right)\end{array}\right\}\right)$$

$$\mathrm{active/inactive}\ =\ \left\{\begin{array}{l}\rightarrow\underline{\mathrm{ACT}}\mathrm{IVE}\\ \underline{\mathrm{INA}}\mathrm{CTIVE}\end{array}\right\}\ \left(\underline{\mathrm{AND}}\left\{\begin{array}{l}\underline{\mathrm{INA}}\mathrm{CTIVE}\\ \underline{\mathrm{ACT}}\mathrm{IVE}\end{array}\right\}\right)$$

An example of the output produced by STEEL TAKE OFF BY PROFILE NAMES is shown below:

```
STEEL  TAKE  OFF  MEMBERS  EXISTING  BY  PROFILE NAMES



     <<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>
     <                                                                                   >
     <  Active Units     Weight Unit =  KIP      Length Unit =  FEET                    >
     <                                                                                   >
     <  Steel Take Off Itemize Based on the PROFILE                                     >
     <  Total Length, Volume, Weight, and Number of Members                             >
     <                                                                                   >
     <  Profile Names     Total Length    Total Volume    Total Weight    # of Members  >
     <  W10x49              4.6098E+01      4.6098E+00      2.2567E+00          2        >
     <  W12x45              2.3049E+01      2.1128E+00      1.0343E+00          1        >
     <  W14x132             1.0000E+01      2.6944E+00      1.3190E+00          1        >
     <  W14x145             1.0000E+01      2.9653E+00      1.4516E+00          1        >
     <  W14x257             1.5000E+01      7.8750E+00      3.8551E+00          1        >
     <  W14x311             1.0000E+01      6.3472E+00      3.1072E+00          1        >
     <  W14x99              6.0000E+01      1.2125E+01      5.9357E+00          2        >
     <  W18x40              5.0000E+00      4.0972E-01      2.0058E-01          1        >
     <  W21x57              1.5000E+01      1.7396E+00      8.5160E-01          1        >
     <  W21x68              1.2500E+01      1.7361E+00      8.4990E-01          3        >
     <  W21x83              1.5000E+01      2.5312E+00      1.2392E+00          1        >
     <  W24x104             3.0000E+01      6.3750E+00      3.1208E+00          1        >
     <  W24x55              1.0000E+01      1.1250E+00      5.5074E-01          2        >
     <  W24x62              1.2500E+01      1.5799E+00      7.7341E-01          3        >
     <  W24x68              2.7500E+01      3.8385E+00      1.8791E+00          5        >
```

```
<  W24x76              3.2500E+01      5.0556E+00      2.4749E+00           5      >
<  W24x84              4.0000E+01      6.8611E+00      3.3588E+00           4      >
<  W27x146             4.0000E+01      1.1917E+01      5.8337E+00           2      >
<  W30x108             1.0000E+01      2.2014E+00      1.0777E+00           2      >
<  W30x116             1.0000E+01      2.3750E+00      1.1627E+00           2      >
<  W30x191             2.0000E+01      7.7917E+00      3.8144E+00           1      >
<  W30x90              5.0000E+00      9.1667E-01      4.4875E-01           1      >
<  W33x118             3.5000E+01      8.4340E+00      4.1288E+00           3      >
<  W33x130             1.0000E+01      2.6597E+00      1.3020E+00           1      >
<  W36x160             1.5000E+01      4.8958E+00      2.3967E+00           2      >
<  W36x182             2.0000E+01      7.4444E+00      3.6444E+00           1      >
<  W40x167             5.0000E+00      1.7049E+00      8.3460E-01           1      >
<  W40x183             1.5000E+01      5.5938E+00      2.7384E+00           1      >
<  W40x215             2.5000E+01      1.0990E+01      5.3799E+00           3      >
<  W40x244             2.0000E+01      9.9583E+00      4.8750E+00           1      >
<  W40x298             2.5000E+01      1.5208E+01      7.4451E+00           2      >
<  W4x13               1.4142E+01      3.7614E-01      1.8414E-01           1      >
<  W6x15               8.7111E+01      2.6799E+00      1.3119E+00           5      >
<  W6x20               1.4142E+01      5.7649E-01      2.8222E-01           1      >
<  W6x9                6.0000E+01      1.1167E+00      5.4666E-01           6      >
<  W8x21               6.0000E+01      2.5667E+00      1.2565E+00           6      >
<  W8x28               3.0000E+01      1.7188E+00      8.4140E-01           2      >
<  W8x31               6.7082E+01      4.2532E+00      2.0821E+00           2      >
<  W8x35               6.4031E+01      4.5800E+00      2.2421E+00           2      >
<  W8x48               2.3049E+01      2.2569E+00      1.1048E+00           1      >
<                                                                                  >
<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>
```

```
<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>
<                                                                          >
<  ACTIVE UNITS     WEIGHT  KIP      LENGTH  FEET                          >
<                                                                          >
<  TOTAL LENGTH, WEIGHT AND VOLUME FOR SPECIFIED MEMBERS                   >
<                                                                          >
<  LENGTH =   1.0487E+03   WEIGHT =   8.9193E+01   VOLUME =   1.8220E+02  >
<                                                                          >
<                                                                          >
<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<<>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>
```

The modified STEEL TAKE OFF command is documented in Section 2.1.12.3 of Volume 1 of the GTSTRUDL Reference Manual.

10. Additional error checking has been added during the Consistency Check phase of all analyses. Now, if supports have not been specified, you will receive the following error message:

    ****ERROR_STCHCK - - No supports have been specified. SCAN mode is entered.

    In addition, if you are performing an analysis and the last specified structural type is either, TYPE SPACE FRAME, TYPE PLATE, or TYPE TRIDIMENSIONAL and you have not supported one of the translational degrees-of-freedom, you will receive the following warning message:

****WARNING_STCHCK - - Possible input ERROR: One or more global degrees-of-freedom are unsupported. The missing DOF are: X

11. A new command has been implemented as a prerelease feature to locate and remove duplicate joints. The LOCATE DUPLICATE JOINT command is documented in detail in Chapter 5 of this Release Guide.

2.7 GTMenu

The enhancements to GTMenu will be presented in the following order:

- Menu Bar
  Enhancements under File, Create, Edit,...,etc..
- Button Bar
  Enhancements under the items in the Button Bar
- Graphics Display Area
- Other Enhancements

Menu Bar

The Menu Bar in GTMenu has been changed as shown below:

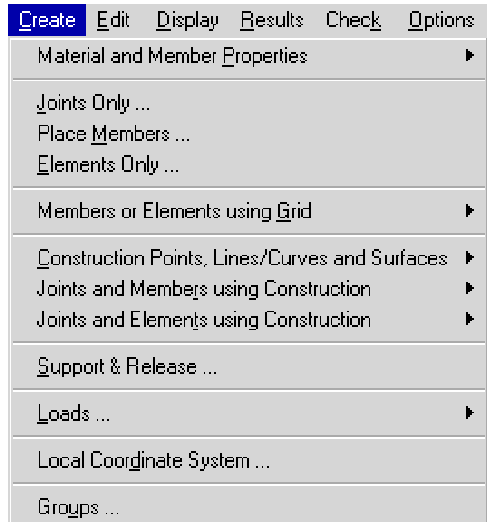
A new item, Results, has been added to the Menu Bar. The Results pulldown will be discussed later in this section.

File Pulldown from the Menu Bar

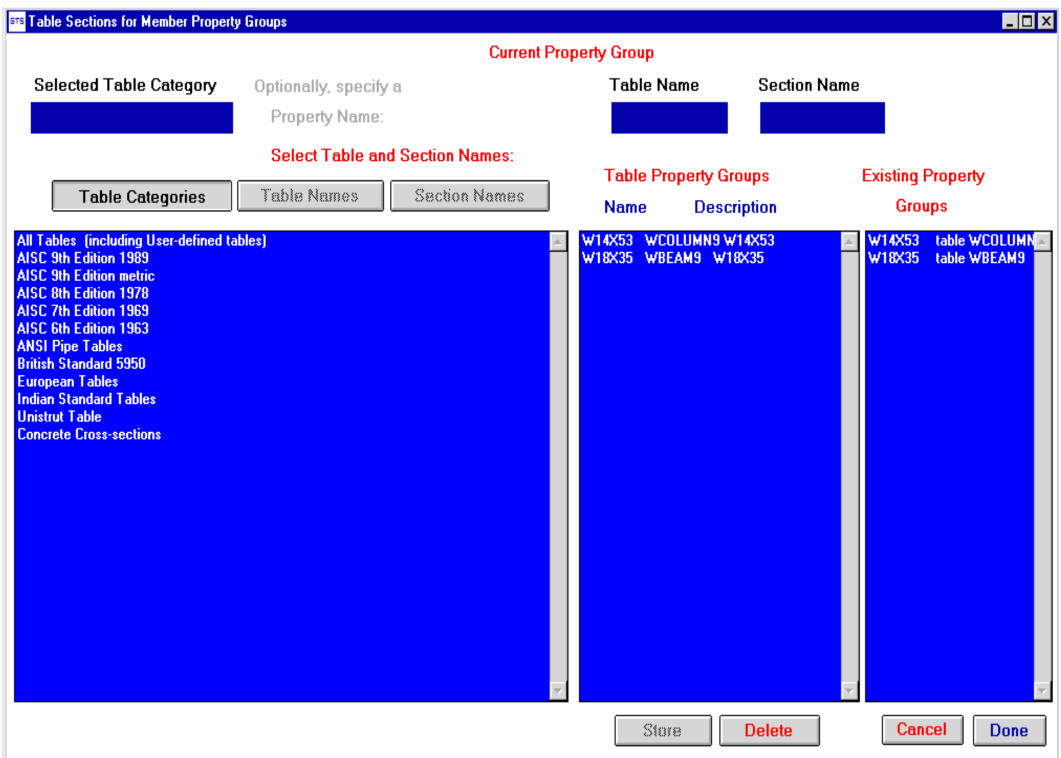
The items under the File menu have not changed. However, a change has been made to the Generate Strudl Input file option. The input file generated will now contain dynamic mass data such as the Inertia of Joints Lumped or Consistent, Inertia of Joints Weight, and Member Added Mass.

Create Pulldown from the Menu Bar

The modified Create menu is:



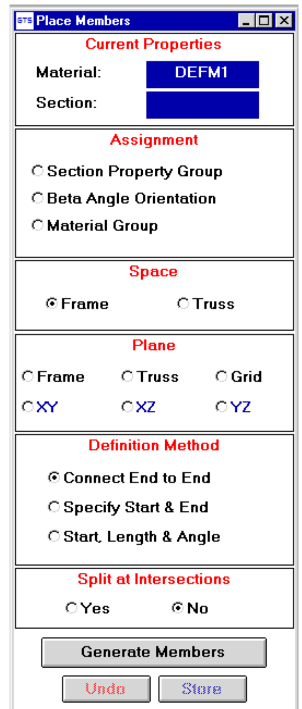
Under Create → Member Properties → Tables, the dialog has changed. Tables of cross-section shapes, profiles, are now stored by Table Categories. An example of the dialog which now appears after the user has selected Tables is shown below:



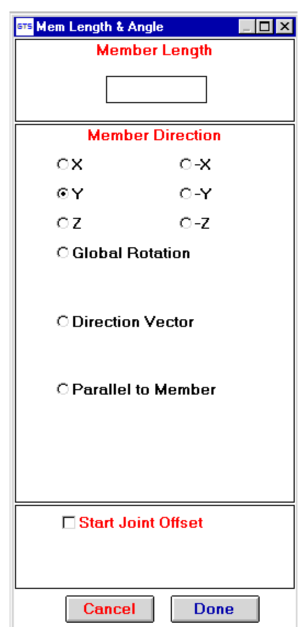
After selecting the Table Category, the user will then see the available tables in the selected category much as in previous releases except the number of available tables will be a much smaller list of tables for each category except the All Tables category.

The Members Only selection under the Create menu has been replaced by a Place Members selection.

The Place Members dialog has a new method for defining members and also the ability to specify if you would like members to be automatically split when a newly created member intersects or crosses existing members.



You may now define members by specifying the Start, Length, and Angle of the member as shown in the dialog:

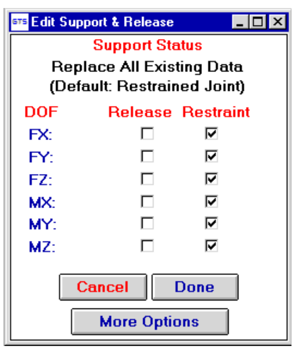


The Start of the member may be specified by selecting an existing joint or specifying a coordinate depending on if Joint or Coord is selected on the Mode Bar. You may also have the start of the member offset from an existing joint so you do not need to know the coordinates of the existing joint.

The Members or Elements Using Grid selection under the Create menu also has the new Start, Length, and Angle method for defining members and the ability to automatically split members.

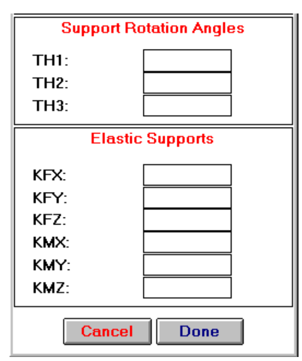
A new Support and Release option has been added under the Create menu. Previously, if a joint was created as being free (unsupported), the user would have to go to the Edit Joints dialog in order to change the joint to a supported joint and specify which forces of moments were restrained or released. The new Create Supports dialog is shown below:

After the supported joints have been selected, the Edit Support & Release dialog shown below appears:



If you select the More Option button on this dialog, you may also specify rotated support directions as well as elastic supports as shown in the dialog:

After specifying the supports and clicking on Done in the Edit Support & Release dialog, the supports will now be automatically labeled and a legend indicating the restrained degrees of freedom will be displayed in the graphical display area.

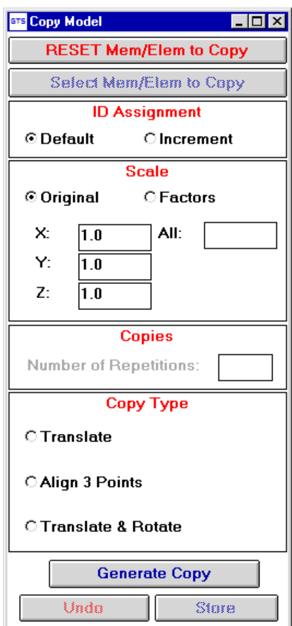


A new Groups option has been added under the Create pulldown. This is the same dialog that appears under Edit Groups and it has been added also to the Create pulldown.

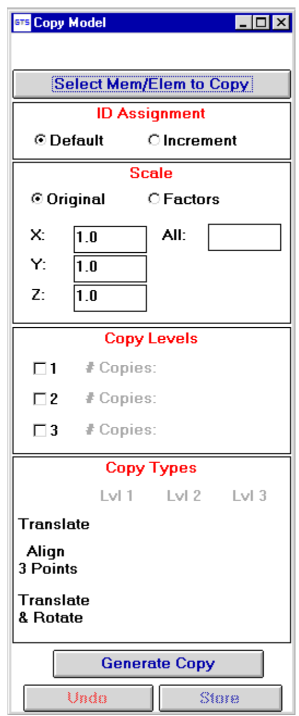
Edit Pulldown from the Menu Bar

The Edit Joints Support & Release dialog has been resized and changed as discussed under Create Support & Release above. The dialog has been resized and the Support Rotation Angle and Elastic Supports options do not appear unless the user selects the More Options button.

The Copy Model dialog has been changed. The old and the new dialogs are shown below for comparison purposes:



(Version 26)

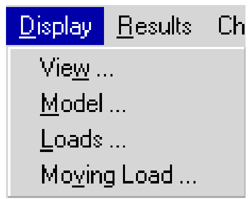


(Version 25)

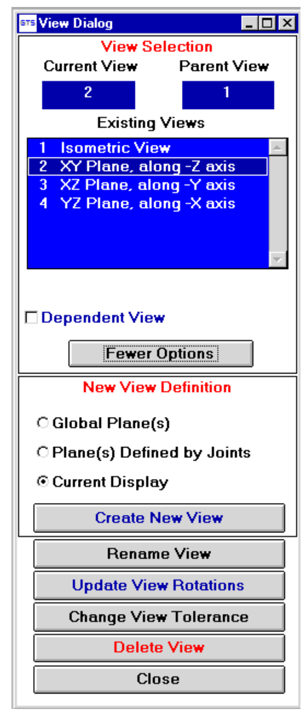
As can be seen by comparing the two dialogs, the Level (Lvl, Lvl2, and Lvl3) have been removed from Version 26. Our experience has been that this feature was rarely used and tended to cause more confusion in using the Copy Model feature than it was worth.

Display Pulldown from the Menu Bar

The modified Display menu is shown below:



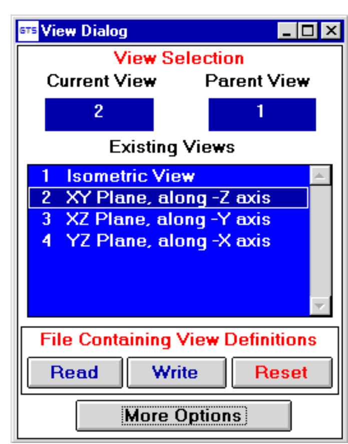
The modified Display →View dialog is:



A new View option has been added which allows you to create a view of the Current Display. If you have zoomed in on a particular area of your model and rotated the model to a particular orientation, you may now create a view which contains the current display. Only complete members or elements are stored in this view so members or elements which are clipped will not be shown when you later double click on a view created using the Current Display option.

Under New View Definition, the wording has also been changed for the second option to "Plane(s) Defined by Joints" as several users were confused with the previous wording under the old View Type heading which was "Skew Plane."

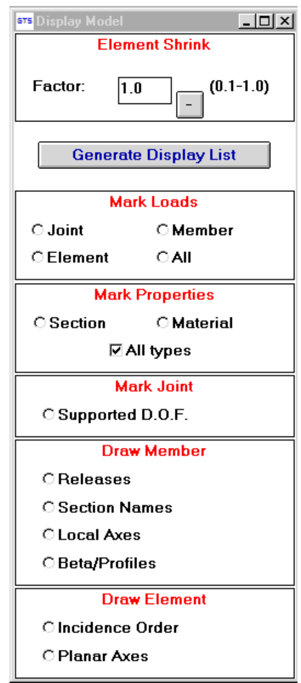
If you select Fewer Options from the modified View Dialog, the dialog shown to the right will appear:



Each time you create new views, a file called *GTMenu_Views.txt* is written to your current Working Directory which contains all of the information needed by GTMenu to define your current views. This is a text file with comments at the top which you can manually edit or delete to create additional views. If you create an input file using GTMenu and would like to recover the views which you had defined when the input file was created, you may select the Read button on the dialog and then select the file containing the views.

The number of views allowed in GTMenu has been increased from 100 to 1,000 in this release as we had several users who wanted to create over 100 views.

The modified Display Model dialog is:

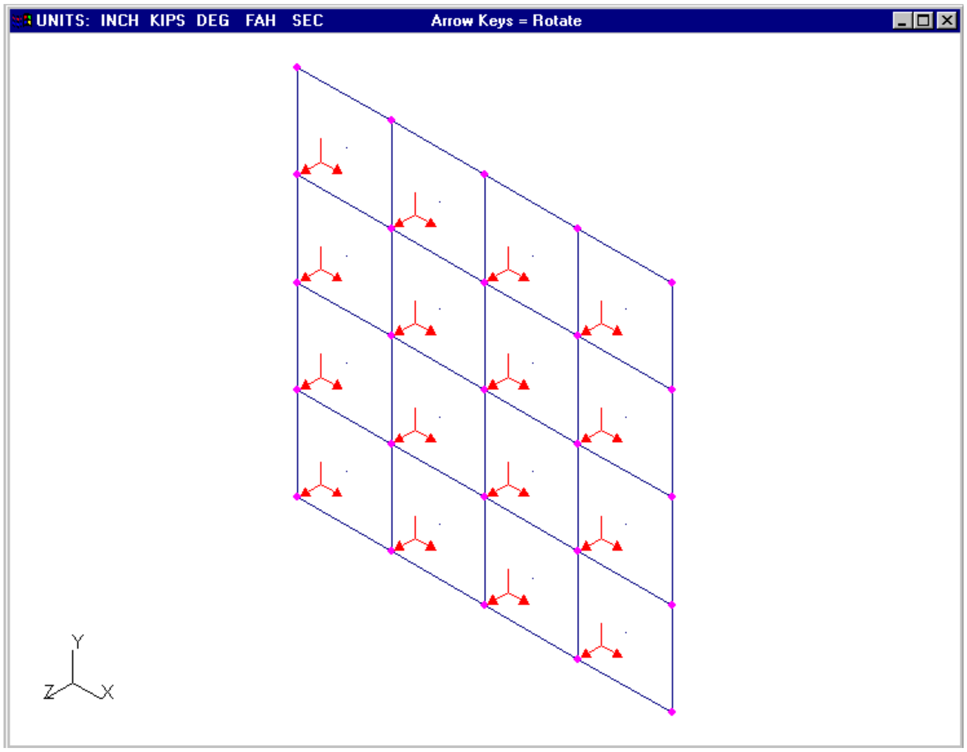


New "+" and "-" buttons have been added to set the element and member shrink factors. Changes in terminology have been made as shown below:

Version 26	Version 25
Mark Joint Supported D.O.F.	Draw Joint Supports
Draw Member Beta/Profiles	Draw Member Profiles

The Draw Member Orientation option has been removed as many users were confused by this option and did not find it useful. Members with the same Orientation were shown on the screen and the Orientation legend was displayed which indicated the rotations needed to rotate the member from being aligned parallel to the global x axis to the current position of the member.

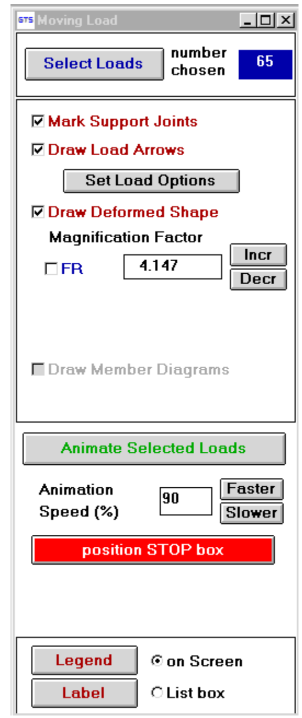
A new option has been added to display the Planar Axes of 2D finite elements. This has been added since the stress output including stress contours for all 2D finite elements is in the Planar coordinate system of the element. An example of the Planar Axes display is shown below:



The Planar X and Y axes are in the plane of the element with arrowheads on the Planar X and Z axes.

A new option has been implemented to Display Moving Loads. This feature will allow you to animate selected loading conditions to visualize the loads as they move across your structure. You may also have the deformed shape of the structure drawn as the loads move across the structure. The new Display Moving Loads dialog is shown below:

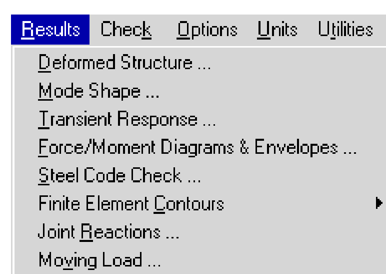
You may have your supports marked on the display and change the format of the moving loads display such as only showing vectors for the loads or showing the vectors and values of the load. Several options are available to scale the deformed shape with the default scaling set to scale the deformed shape based on the maximum displacement over all of the selected loading conditions. You may stop the animation at any time by clicking on the right mouse button. You may then label the magnitude of the deformations on the screen or in a List Output box. After stopping the animation, you may resume the animation or step the loads forward or backward.



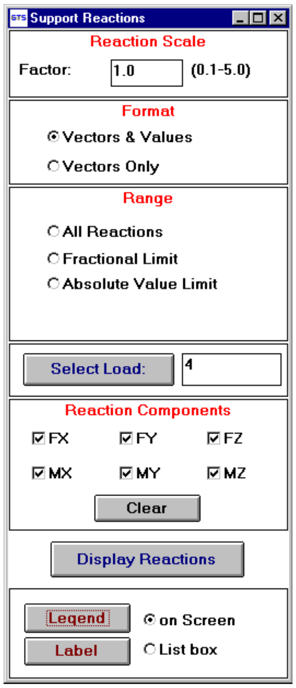
Results Pulldown from the Menu Bar

The new Results pulldown is shown below:

In Version 25 of GTSTRUDL, the ability to display results was located under the Display Model pulldown. Since Version 25, two new features have been added to the result display in GTMenu. You may now display Joint Reactions and Moving Loads. The ability to display Moving Loads is also available under the Display pulldown and has been discussed above.

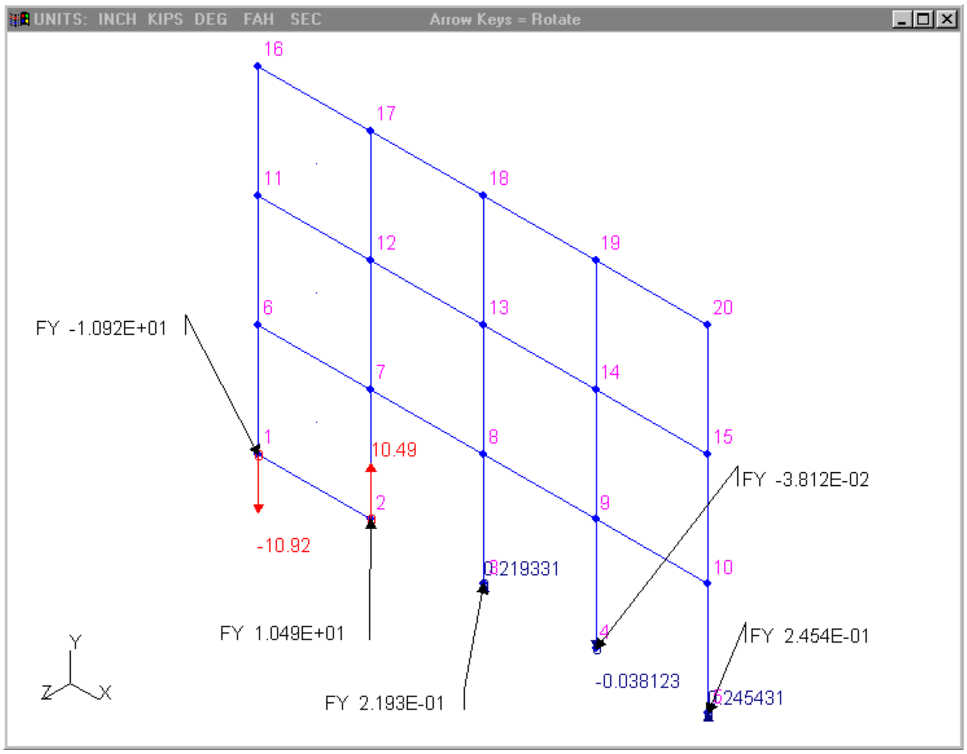


The new Support Reactions dialog is:



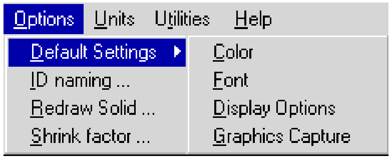
You may display the reactions as vectors with and without values on the screen. You may select which reaction components to display. You may also have the reaction values labeled on the screen.

An example of the display of the FY reactions and the label of reactions is shown below:

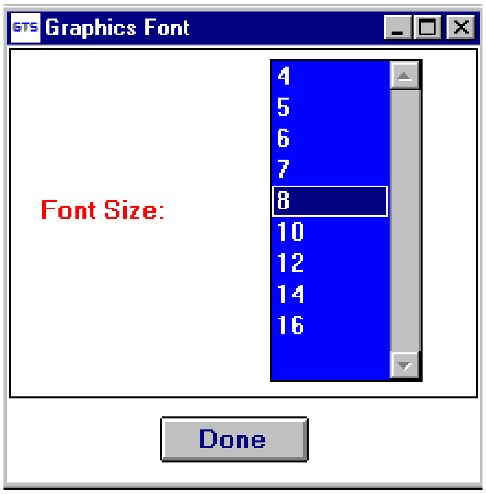


Options Pulldown from the Menu Bar

The modified Options pulldown is shown below:



Under Default Settings, you may now specify the size of the font to be used for annotation in the Graphical Display Area. The new Font dialog is shown below:



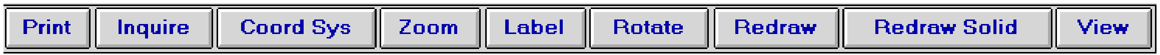
This feature was added in order to have the ability to reduce the size of the font when large models are displayed and labeled on the screen.

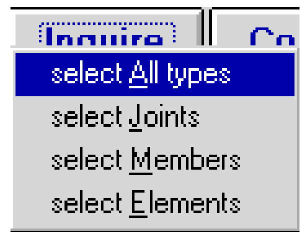
The ID naming option was available as a button on the Button Bar in previous versions. This option allows you to control the number of new joints and members/elements in your model.

The Shrink Factor option now has “+” and “-“ buttons which allow you to increase or decrease the shrink factor used to shrink elements and members.

Button Bar

The modified Button Bar is shown below:



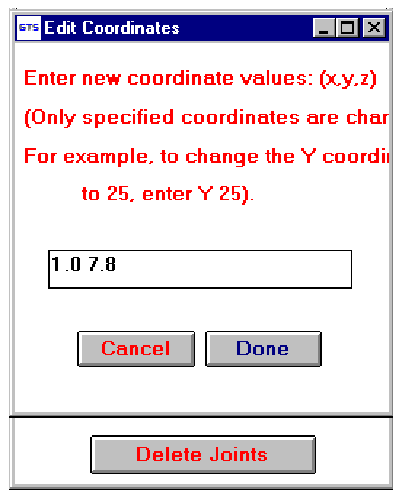


The Inquire button on the Button Bar is a new feature in this release. The Inquire pulldown menu is shown to the right:

Inquire allows you to obtain information about joints, members, and elements. After clicking on the Inquire button and clicking on one of the choices, you may then click on the selected choice (joint, member, element) in the graphics area have information output in a List Output dialog. An example of the List Output dialog obtained by selecting Joints 16 and 21 in the graphics area is shown below:

You may also activate the Edit function for any of the information in the List Output dialog by clicking on the line you wish to Edit and then clicking on the Edit button. You may also double click on the line to Edit. For example, if the coordinates of Joint 16 are to be changed, double click on the line in the List Output dialog containing the coordinates (0.00,7.50,0.00) and then click on the Edit button at the bottom of the dialog as shown on the next page:

Click on Respecify Coordinate in the pop-up dialog that appears and then enter the new coordinates in the Edit Coordinates dialog shown to the right and then click on the Done button:



You may also perform an Inquire by simply right clicking on the mouse while the mouse is located on a joint, member, or element in the graphical display area. For small models or on windows of larger models, performing the Inquire by right clicking the mouse is more convenient than using the Inquire button. However if the display in the graphical display area contains joints, members, and elements which are spaced closely together, you may find that by selecting the Inquire button and then selecting Joints, Members, or Elements is more useful since only those entities (Joints, Members, or Elements) will be detected when you click in the graphics area.

The default angular increment for the Set Arrow Key option under the Zoom button has been changed from 10 degrees to 5 degrees.

The Label Dimensions, Coordinates, Names, Comments option under the Label button has been modified. You may now have the comment pointing to a joint, member, or element by clicking on the display comment with the Pointer option in the dialog. Then, click on a reference point on the structure and position the comment.

The View button has been changed and is documented above under Display View.

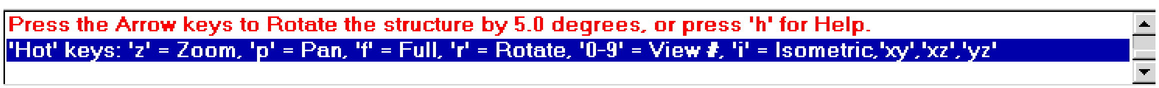
Graphics Display Area

The display of joints has been changed in this release. Previously, a joint would be displayed with an “x” and now a joint is displayed with a solid •.

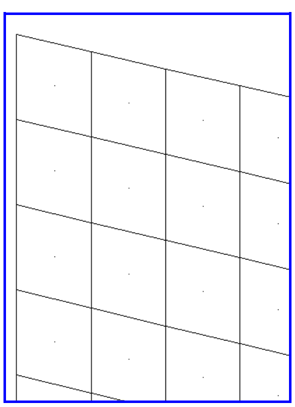
New “hot keys” have been added which initiate actions in the Graphics Area. These additional “hot keys” are:

“F” or “f”	to return to the Full structure (use after zooming)
“1” - “9”	will cause view 1, 2, 3 . . ., 9 to be displayed
“0nn”	will cause view nn to be displayed where nn is an integer from 10 - 99
“i”	will cause an isometric rotation to be displayed
“xy”	will cause an xy plane rotation to be displayed
“xz”	will cause an xz plane rotation to be displayed
“yz”	will cause a yz plane rotation to be displayed

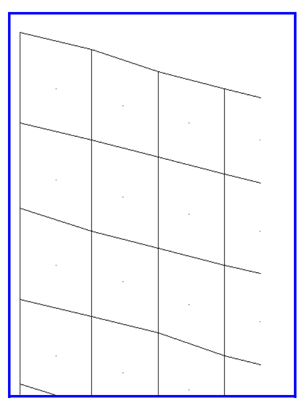
The Message Area will show prompts for these “hot” keys as shown below:



The resolution used for the display in the Graphics Area has been increased by almost a factor of 64. This will improve the resolution of the display and result in sharper displays for large models and especially for large finite element models. The effect of this increase in resolution is best demonstrated by the images below:



Version 26



Version 25

Other GTMenu Enhancements

The time required to bring a model from GTSTRUDL into GTMenu has been greatly reduced. Examples of the time required for two models are shown below:

Example 1

4350 joints
3000 members
3581 finite elements
12 independent loadings
4 dependent loadings

Time to Prepare Model (going from GTSTRUDL to GTMenu)
Version 25 25 seconds
Version 26 1 second

Example 2

14793 joints
1156 members
18665 finite elements
4 independent loadings

Time to Prepare Model (going from GTSTRUDL to GTMenu)
Version 25 790 seconds
Version 26 5 seconds

The time required to translate the loads from the Menu database to the STRUDL database has been decreased by over 50% in some cases. An example is shown below:

Example 1

14793 joints
1156 members
18665 finite elements
4 independent loadings

Time to translate loads (going from GTMenu to GTSTRUDL)

Version 25	56 seconds
Version 26	23 seconds

2.8 Nonlinear

1. The following nonlinear features remain prerelease features in Version 26 and are described in Chapter 5 of this Release Guide:

    - Plastic hinges for frame members
    - LIST PLASTIC HINGE DISPLACEMENT
    - LIST PLASTIC HINGE STATUS
    - PERFORM PUSHOVER ANALYSIS
    - LIST PUSHOVER DUCTILITY RATIO
    - LIST PLASTICITY DUCTILITY RATIO
    - Nonlinear member end connection springs
    - PRINT NONLINEAR EFFECTS

2. The NONLINEAR EFFECTS command has been extended to include a FRICTION DAMPER effect. The FRICTION DAMPER effect may be applied to plane and space truss members and permits the truss member axial force to increase to a maximum constant user-specified slip force. Loading-unloading cycles are hysteretic. The FRICTION DAMPER effect may be used in both static and dynamic nonlinear analysis. The FRICTION DAMPER is described in Section 5.6 of this Release Guide.

3. In previous versions, geometry nonlinearity for space frame members was limited to doubly-symmetric cross-sections. As of this version, this restriction is no longer present. Nonlinear geometric behavior for space frame members now supports any cross-section shape.

2.9 Offshore

1. Offshore fatigue analysis has been extended to support joint constraints (joint ties and rigid bodies).

2. The new LIST FATIGUE SECTION command has been added to the results output functions for offshore fatigue analysis. This command enables the computation and output of fatigue damage and life results at intermediate sections along a fatigue member according to the PSD method of fatigue analysis.

2.10 Reinforced Concrete

1. The 1999 ACI Code has been added as a prerelease feature. Beams and columns may now be designed using the 1999 ACI Code. More details regarding the 1999 ACI may be found in Section 5.2 of this Release Guide.

2.11 Steel

1. When the magnitude of the section forces are too small, the steel design SELECT or CHECK command may print them in an exponential (E) format. A new enhancement has been added to the steel design SELECT or CHECK command output that when the section force values are too small, a value of 0.0 is printed instead of a very small value in an exponential (E) format. To take advantage of this new enhancement, the user must specify the OUTPUT LONG NAME command prior to the SELECT or CHECK command. Note that the OUTPUT LONG NAME command can be added anywhere before the SELECT or CHECK command and is the default in Version 26.

   Following is the steel design output from the SELECT command that shows the small values in an E format:

```
SELECT  MEMBERS  434  TO  436

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

 434       WSHAPES9    29           1400.000 H1-1b C      0.928       -4.043       0.113     -1.534 MM   METN
 LRFD2     W6x20                             KL/r, B7     0.184    1.074E-12  -2148.277   -171.718     1
 435       WSHAPES9    29              0.000 H1-1b C      0.984       -4.957      -0.028      0.370 MM   METN
 LRFD2     W6x20                             KL/r, B7     0.749   -2.620E-14  -2143.148   -100.411     1
 436       WSHAPES9    17           1400.000 H1-1b C      0.943       -7.557       0.135      1.505 MM   METN
 LRFD2     W6x20                             KL/r, B7     0.184    1.457E-12   2110.414   -214.619     1
```

   Following is the above SELECT command example that uses the new output procedure. Note that values of 0.0 are printed instead of small E format values:

```
OUTPUT  LONG  NAME
SELECT  MEMBERS  434  TO  436

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

 434       WSHAPES9    29           1400.000 H1-1b C      0.928       -4.043       0.113     -1.534 MM   METN
 LRFD2     W6x20                             KL/r, B7     0.184        0.000  -2148.277   -171.718     1
 435       WSHAPES9    29              0.000 H1-1b C      0.984       -4.957      -0.028      0.370 MM   METN
 LRFD2     W6x20                             KL/r, B7     0.749        0.000  -2143.148   -100.411     1
 436       WSHAPES9    17           1400.000 H1-1b C      0.943       -7.557       0.135      1.505 MM   METN
 LRFD2     W6x20                             KL/r, B7     0.184        0.000   2110.414   -214.619     1
```

2. A new steel design option has been added to the SUMMARIZE command that allows the deletion of the summary data.

3. The default parameter values for individual steel design codes are now printed when PRINT PARAMETER DICTIONARY, PRINT DESIGN DATA, or PRINT DATA is specified. When the above print commands are used, default values for parameters such as STEELGRD, TBLNAM, etc. are printed for each steel design code. A partial output from the PRINT PARAMETER DICTIONARY is shown below:

```
 {   30} >
 {   31} > PRINT  DESIGN  DATA
1
 ****************************************
 *  PROBLEM DATA FROM INTERNAL STORAGE  *
 ****************************************


 JOB ID - MS393        JOB TITLE - Check new PRINT PARAMETER DICTIONARY OUTPUT

 ACTIVE UNITS -  LENGTH        WEIGHT        ANGLE       TEMPERATURE      TIME
                   FEET          KIP           RAD           DEGF          SEC


          ********** DESIGN DATA **********


 STRUDL DATA SET

 PARAMETER DICTIONARY-----------------------------------------/
 NAME        TREATMENT    STANDARD    L    W    A    TEMP TIME

 CODE        REQUIRED     ALPHA
 TBLNAM      CMPALPHA     QQSTTTBL
 STEELGRD    STANDARD     A36
 FYLD        COMPALWY     QQSTFYLD     -2    1
 FTS         COMPALWY     QQSTFYLD     -2    1
 K           STANDARD         1.00
 KY          STANDARD         1.00
 KZ          STANDARD         1.00
 COMPK       STANDARD     NO
 GAY         STANDARD     COMPUTE
 GAZ         STANDARD     COMPUTE
 GBY         STANDARD     COMPUTE
 GBZ         STANDARD     COMPUTE
 ........    ........     ........
 ........    ........     ........
 ........    ........     ........
 ........    ........     ........
 REDFY       STANDARD         1.00
 REDFU       STANDARD         1.00
 L-LTB       COMPUTE      QQMSECPR      1
 REDSTRES    STANDARD     YES

 Default Parameter Values for Individual Design Codes
     Codes 69AISC and W69AISC
          Parameter      Default Value          Parameter      Default Value
          ----------     -------------          ----------     -------------
           TBLNAM         STEELW                 STEELGRD       A36

     Code 78AISC and W78AISC
          Parameter      Default Value          Parameter      Default Value
          ----------     -------------          ----------     -------------
           TBLNAM         STEELW78               STEELGRD       A36

     Code DBLANG
          Parameter      Default Value          Parameter      Default Value
          ----------     -------------          ----------     -------------
           TBLNAM         EQDBLANG               STEELGRD       A36
```

```
Code TOWER
     Parameter      Default Value         Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------         ----------     -------------
      TBLNAM         ANGLES                STEELGRD       A36                   SLENTEN        500

Code NF17
     Parameter      Default Value
     ----------     -------------
      TBLNAM         STEELW78

Code NF83
     Parameter      Default Value
     ----------     -------------
      TBLNAM         WSHAPES9

Code BS449
     Parameter      Default Value         Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------         ----------     -------------
      TBLNAM         UNIBEAMS              STEELGRD       43                    KL/R           200

Code TOWER2
     Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------
      TBLNAM         ANGLES                STEELGRD       A36
      SLENTEN        500                   W/TMAX         25

Code ASD9 and LRFD2
     Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------
      TBLNAM         WSHAPES9              STEELGRD       A36
      SLENCOMP       200                   SLENTEN        300

Code APIWSD20 and APILRFD1
     Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------
      TBLNAM         PIPES9                STEELGRD       A36
      SLENCOMP       200                   SLENTEN        300

Code BS5950
     Parameter      Default Value         Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------         ----------     -------------
      TBLNAM         UNIBEAMS              STEELGRD       43                    SLENCOMP       180

Code AISI89
     Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------
      TBLNAM         UNISTRUT              STEELGRD       A570GR33
      SLENCOMP       200                   W/TMAX         60

Code CAN97
     Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------
      TBLNAM         WSHAPES9              STEELGRD       A36
      SLENCOMP       200                   SLENTEN        300

Code EC3
     Parameter      Default Value         Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------         ----------     -------------
      TBLNAM         EUROPEAN              STEELGRD       S235                  SLENCOMP       200

Code IS800
     Parameter      Default Value         Parameter      Default Value
     ----------     -------------         ----------     -------------
      TBLNAM         ISBEAMS               STEELGRD       A36
      SLENCOMP       180                   SLENTEN        400
```

4. A new steel design code names IS800 has been added as a prerelease feature. 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 are as follows:

I-shapes	Single Angles
Channels	Tees

Double Angles	Solid Square Bars
Solid Round Bars	Solid Rectangular Bars
Pipes	Structural Tubes

The PARAMETERS for the IS800 code are documented in Section 5.4 of this Release Guide.

5. AISC LRFD specifications from the "Specification for the Design of Steel Hollow Structural Sections", April 15, 1997 has been implemented for Pipe cross-sections.

6. Three new cross-sections has been added to the EC3 code. The new cross-sections are Solid Round Bar, Single Angle, and Double Angles. You may code check or design solid round bars based on axial and bending loads. Single and double angles may be code checked or designed for axial loads only. No new parameters have been added to the EC3 code for the above cross-sections. EC3 code parameters are documented in the Table EC3, Section 2.1 of Volume 2A are applicable to the solid round bars, single angles, and double angles cross-sections.

7. Twenty-five new steel grades has been added to the EC3 code.

8. A new parameter for the 1/3 allowable stress increase has been added to the ASD9 code. An example of this parameter is as follows:

```
PARAMETER
ALSTRINC     33.3333     LOADS  'Wind-L'  'Wind-R'  'Seismic'
```

This parameter is used to specify the allowable stress increase value for the ASD9 code only. This parameter can be used to specify the 1/3 allowable stress increase for wind or seismic loads. The user specified value for this parameter must be followed by the load list. An example for this parameter is to specify a value of 33.3333 followed by a load list. The default value for this parameter is 0.0.

9. When the CHECK or SELECT command member list contains inactive members, a warning message is issued and the member is ignored. You can use OUTPUT WARNING MESSAGE LIMIT to suppress the warning messages. If you do not want to see the inactive member warning messages, use the ALL BUT option or specify the active members in the member list of the CHECK or SELECT command.

10. Two new parameters have been added to steel design to allow the CHECK and SELECT commands to print actual and allowable section stresses for allowable stress codes or section limiting values for limit state and load and resistance factor codes. The new parameters are "PrintStr" and 'PrintLim'. You may use either one of these parameters and they will have the same effect. This new option is applicable to the following codes:

AISI89,	APILRFD1,	APIWSD20,	ASD9,
BS5950,	CAN97,	EC3,	IS800,
LRFD2,	NF17,	NF83,	TOWER2,
78AISC			

Examples for these parameters are shown below:

Example # 1

```
PARAMETERS
            CODE        ASD9        ALL  MEMBERS
            PrintStr    YES         ALL  MEMBERS
```

Example # 2

```
PARAMETERS
            CODE        LRFD2       ALL  MEMBERS
            PrintLim    YES         ALL  MEMBERS
```

These parameters are used to print the actual and allowable section stresses for allowable stress codes or section limiting values for limit state, load, and resistance factor codes. These parameters are applicable to the steel design CHECK and SELECT commands. The default CHECK and SELECT output print the section forces. A value of 'YES' for these parameters indicates that the actual and allowable section stresses or section limiting values should be printed instead of default section forces. The default value for these parameters is 'NO'.

11. Deflection check and design is now available as a prerelease feature. The deflections check and design may be performed with or without a code check or select and is available for all steel design codes. New parameters have been added for the deflection check and design. More information regarding this new feature may be found in Section 5.3 of this Release Guide.

12. A new command has been implemented to output the results of a CHECK or SELECT. The results output by this new LIST CODE CHECK command are the same as those available in the graphical display of code check results in GTMenu. The output includes the member name, pass/fail status, critical section location, load name, code provision, and actual/allowable ratio, and profile.

    This new command is documented in Section 2.1.14.4.3 of Volume 1 of the GTSTRUDL Reference Manual.

2.12 Steel Tables

1. Tables containing single and double angles from AISC have been reorganized. The AISC ordering of the angles is now used. The new profile ordering is based on the largest leg size to the smallest. This new profile ordering makes it easier to find cross-sections in the angle tables. The reordered tables are:

   Single angles tables: ANGLES, EQANGLE, ULANGLE, USANGLE
   Double angles tables: EQDBLANG, LLDBLANG, SLDBLANG

2. The following Indian Standard Tables are now included in the GTSTRUDL database:

ISBEAMS	ISCHAN	ISEQANGL
ISCOLUMN	ISCHAMCP	ISUNANGL

   The above tables are from Indian Standard, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision, IS 808 : 1989.

3. Steel profile tables for LRFD Plate girders and the EC3 codes are now documented in Appendix C of Volume 2A of the GTSTRUDL Reference Manual.

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. Eigenvalue analysis using tridiagonalization will no longer abort when structural instabilities exist in the model. (GPRF 2000.17)

2. The PRINT DYNAMIC STIFFNESS/MASS/DAMPING MATRIX command will no longer abort if the specified matrices were input as FULL or SYMMETRIC matrices using the MATRIX command and rows having only 0.0 matrix values were not included in the input. (GPRF 2000.18)

3. The CREATE RESPONSE SPECTRUM ACCELERATION command now produces correct results. (GPRF 2001.05)

4. Interpolation of spectral accelerations, velocities, and displacements on a response spectra curve is now correct when the input frequency exactly matches the frequency corresponding to the last point on the curve. (GPRF 2001.12)

5. Incorrect syntax in the DYNAMIC PARAMETERS STORE command will no longer cause an infinite loop. (GPRF 2002.1)

3.2 Elastic Buckling

1. Elastic buckling no longer aborts if joint constraints (JOINT TIES and RIGID BODIES) are present in the model. (GPRF 2001.04)

3.3 Finite Elements

1. The CALCULATE RESULTANT command will no longer abort or output incorrect names for joints and elements in error or warning messages under the following conditions:

    a. A joint is specified on the cut that doesn't exist or is not on the cut.

b. An element is specified in the element list which does not exist or does not contain a joint on the cut.

c. Results do not exist for the first element specified on the cut.

(GPRF 2001.10)

2. The CALCULATE AVERAGE STRESS (AND) ENVELOPE command will now produce complete output when TRIDIMENSIONAL elements are present in the model and results do not exist for the last active loading condition. (GPRF 2001.15)

3. The CALCULATE RESULTANT command will no longer abort for cuts which are not detected as straight line cuts. The ability to specify cuts which are not straight line cuts was a pre-release feature in Version 25 and is now a released feature in Version 26. In some instances, cuts which the user thought were straight line cuts were detected as not being straight line cuts due to the accuracy of the coordinates specified by the user and an abort resulted. (No GPRF issued)

3.4 General

1. Analysis (stiffness, dynamic, etc.) will no longer abort if a member has a calculated zero length as a result of member eccentricities. Now the following error message is output and analysis terminated.

```
ERROR_STCHCK - - Length of member 212 = 0.0.  Zero length is not permitted.
                 Scan mode entered.
```

(GPRF 2000.15)

2. The MOVING LOAD generator will no longer abort if the load path contains an incompatible list of members. An example of an incompatible list of members is ‘A1’ to ‘B3’ since you may only increment the integer portion of alphanumeric names.
(GPRF 2001.11)

3. The PRINT MEMBER PROPERTIES command will no longer abort if rigid body elements are present in the model and one or more of the rigid body elements were not the last of all members and elements to be created. (GPRF 2001.13)

3.5 GTMenu

1. These errors in GTMenu were corrected in Version 25 after the Version 25 Release Guide was printed and are listed here for your information:

   a. When deleting joints, the message that a joint was a Master or Slave joint will no longer occur for joints which are not Master or Slave joints.

   b. An abort will no longer occur when adding joint releases with spring constants.

   (No GPRF issued)

2. When splitting members that belong to groups, an abort will no longer occur. Previously, an abort would occur due to not reserving enough space for the newly created members to be added to the groups. (No GPRF issued)

3. Projected element loads created in GTMenu will now be translated correctly back to GTSTRUDL. Previously, the loads would appear to be correct when the PRINT LOAD DATA command was issued. However, the following warning message would be output during STIFFNESS ANALYSIS:

   **** STRUDL ERROR 4.39 - NO COMPONENT GIVEN FOR PROJECTED LOADING 1 - LOADING IGNORED

   (No GPRF issued)

4. Steel design parameters will now be correct in the input file created in GTMenu. The following errors have been corrected:

   a. Previously, the parameter values were incorrect if parameters had only been given for individual members.

   b. Previously, the parameter values were incorrect when the user had overridden parameters on individual members with the same parameter specified on all members.

Additionally, steel design parameters and constraints are now correct in the input file when the model contains a mixture of finite elements and members and some or all of the members had been created after the finite elements.
(No GPRF issued)

5. In the input file created in GTMenu, element load types and directions will not be duplicated. In some instances, the input file would have duplicates such as shown below which would generate errors when the input file was opened in GTSTRUDL:

    (list of elements) -
    SURFACE FORCE GLOBAL PZ -2.50000000E+00
    SURFACE FORCE GLOBAL PZ -2.50000000E+00 (duplicate without element list - error)

(No GPRF issued)

6. Check Model will no longer show a blank list of joints indicating that they are free with no members or elements attached when eccentricities are applied to members in the model. (No GPRF issued)

7. Joint temperature gradients are now converted to the current length unit when generating the input file in GTMenu. (No GPRF issued)

8. When creating joints and members or joints and elements and then specifying the spacing for the mesh along the U, V, and W directions, you may no longer select multiple primary numbering directions. Now, you may select only U, V, or W. Previously, you could select all of them and it was unclear as to what was actually being used for the numbering.
(No GPRF issued)

9. An abort no longer occurs during Create Elements Only for large models with high connectivity. (No GPRF issued)

10. An abort will no longer occur if a model is being animated (deformed shape, mode shape, ...,etc.) and the dialog which initiated the animation is closed by the user by clicking on the "X" in the upper right corner of the dialog. Now, the animation is stopped and the dialog is then closed normally. (No GPRF issued)

11. The GTMenu Button Bar will no longer disappear under Windows XP. Previously, the Button Bar would disappear under Windows XP if the desktop Display Properties "Windows and buttons" Appearance was set to "Windows XP style."

3.6 Nonlinear Analysis

1. Nonlinear analysis of structures which contain members having elastic end connections will no longer produce incorrect results. Previously, the results did not reflect the presence of member elastic end connections. (GPRF 2000.13)

2. Nonlinear spring elements connected to slave joints no longer cause nonlinear static analysis to abort. (GPRF 2001.02)

3. The incremental P-Delta effect for nonlinear geometric frame members is no longer ignored for axial member temperature or axial member distortion loads. (GPRF 2001.14)

3.7 Offshore

1. Fatigue analysis results are now computed correctly for members having member eccentricities. (GPRF 2001.03)

3.8 Rigid Bodies

1. Member/element loads applied to rigid body elements will no longer produce an abort during stiffness analysis. (GPRF 2000.14)

2. The RIGID BODY INCIDENCES command now detects the specification of duplicate member, element, or rigid body identifiers. (GPRF 2001.06)

3. The presence of joint constraints (JOINT TIES, RIGID BODIES) no longer automatically computes consistent mass matrices for members and elements unless the user has specified the INERTIA OF JOINTS LUMPED or CONSISTENT command. Joint constraints require the consistent mass format and in previous versions the consistent mass of members and elements was being automatically computed even if the user did not specify the INERTIA OF JOINTS LUMPED or CONSISTENT command. (GPRF 2001.09)

4. Analysis (stiffness, nonlinear, dynamic) will no longer abort if members and elements having mixed degrees-of-freedom are incident on a released slave node. An example of when this condition occurred previously was when a model contained space frame members (6 DOF) and SBHQ plate elements (5 DOF) which were both incident on the same released slave joint. (GPRF 2001.16)

3.9 Steel Design

1. The Parameter CB now accounts for a member's P-Delta effect. The moments used in the C_b computation for I-shapes and plate girder cross-sections in the LRFD2 code now accounts for a member's P-Delta effect. (GPRF 2001.01)

3.10 Windows and Office XP

1. GTSTRUDL will no longer abort during Startup for installations with Office XP and "Alternative User Input" installed (i.e. voice input) and a Network Security key. In version 26, an option has been added during Installation to detect this and to change the dialog which is displayed when searching for a security key. The Security Devices tab under Options has an option to change this setting if Office XP with "Alternative User Input" is installed after GTSTRUDL.

CHAPTER 4

KNOWN DEFICIENCIES

This chapter describes known problems or deficiencies in Version 26. 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 will 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 members 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:

```
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
```

as shown by the underlined portion of the REPEAT command. 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)

7. If a loading contains JOINT TEMPERATUREs and that loading is later deleted and a new loading is created with the same name, the JOINT TEMPERATUREs will be present in the new loading.
A PRINT LOADING DATA will show that the JOINT TEMPERATURES from the original loading 1 still exist in the newly defined loading with the same name. Subsequent STIFFNESS or NONLINEAR ANALYSIS commands will consider JOINT TEMPERATUREs in the newly define loading with the same name
(GPRF 2001.08)

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)

7. GTMenu is limited to 32,000 members, 32,000 joints, 32,000 finite elements, and 1,000 views. If these limits are exceeded, an abort or incorrect results may occur. Other areas of GTSTRUDL do not have these limitations.

   (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 body 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 response to user feedback.

The following features are included in Version 26 as prerelease features:

1. ACI Code 318-99
2. Steel Deflection Check and Design
3. Steel Design by Indian Standard Code IS800
4. GTSTRUDL Profile Tables for the Design based on the IS800 code
5. Nonlinear Effects Command (revised)
6. Nonlinear Analysis Output Commands
7. Pushover Analysis
8. Calculate Error Estimate Command
9. Dynamic Analysis External File Solver to Improve Efficiency of Dynamic Analysis Results Computation
10. Nonlinear Dynamic Analysis
11. The Locate Interference and Duplicate Joint Command
12. Automatic Generation of Static Equivalent Earthquake Loads
13. Reference Coordinate System Command
14. Rectangular and Circular Concrete Cross-Section Tables
15. Hashing Algorithm to Accelerate Input Processing

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

5.2 ACI Code 318-99

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

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

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

Example:

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

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

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

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

CONSTANT	Explanation	ACI 318-99	Assumed Value
FCP	Compressive strength of concrete, f'_c		4000 psi
FY	Yield strength of reinforcement, f_y		60000 psi
WC	Unit weight of plain concrete		145 pcf
DENSITY	Unit weight of reinforced concrete [1]		150 pcf
FC	Allow compr. stress in concrete, F_c	A.3.1	0.45(FCP)
VU	Ult. shear stress in beam with web reinf. [2]	11.5.6.9	$(8\sqrt{FCP} + v_c)$ [5]
V	Allow. shear stress in beam with web reinf.	A.3.1(b)	$(5.5\sqrt{FCP})$
RFSP	Splitting ratio, $f_{ct}/(\sqrt{f'_c})$ [3]	9.5.2.3	6.7
FYST	Yield strength of stirrups		60000 psi
FYSP	Yield strength of spiral		60000 psi
FS	Allowable tension stress in primary reinf.		20000 psi for
FSC	Allowable compressive stress in column reinf. [4]	A.3.2	Grades 40, 50
FV	Allowable tension stress in stirrups [5]		24000 psi for
			Grade 60
PHIFL	Flexure capacity reduction factor	9.3.2	0.90
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	0.70
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

CONSTANT	Explanation	ACI 318-99	Assumed Value
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 STRUDL 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.

5.3 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 explanation 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 number of section 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
```

5.4 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.

5.5 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.

5.6 Nonlinear Effects Command (revised)

The Nonlinear Effect command has been modified to include nonlinear spring connections, plastic hinges, and friction dampers. 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 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

nonlinear member specs =
TENSION (ONLY)
COMPRESSION (ONLY)
GEOMETRY (AXIAL)
NLS Connection Specs
Plastic Hinge Specs
Friction Damper Specs
$list_1$

and where NLS Connection Specs =

NLS (CONNECTION) MEMBERS $list_2$ * START Curve Specs
END Curve Specs

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

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

$$\text{Plastic Hinge Specs} = {}^{*}\begin{array}{l} \underline{\text{STA}}\text{RT} \\ \underline{\text{END}} \end{array} \text{, Fiber Specs, } {}^{*}\left\{ \begin{array}{l} \text{Steel Specs} \\ \text{R - C Specs} \end{array} \right\}$$

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

$$\text{Steel Specs} = [\underline{\text{FY}}]\ v_{FY} \;\; [\underline{\text{EH}}]\ v_{EH} \;\; [\underline{\text{ESH}}]\ v_{ESH} \;\; [\underline{\text{FSU}}]\ v_{FSU} \;\; - \\ [\underline{\text{ESU}}]\ v_{ESU} \;\; [\underline{\text{ALPHA}}]\ v_{\alpha}$$

$$\text{R - C Specs} = \underline{\text{R - C}} \;\; \begin{array}{l} \underline{\text{RECT}}\text{ANGLE } [\underline{\text{B}}]\ v_B\ [\underline{\text{H}}]\ v_H \\ \underline{\text{CIRC}}\text{ULAR } [\underline{\text{B}}]\ v_{DIAM} \end{array} \;\; [\underline{\text{FCP}}]\ v_{FCP} \;\; [\underline{\text{EC0}}]\ v_{EC0} \;\; -$$

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

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

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

Friction Damper Specs = $\underline{\text{FRI}}$CTION ($\underline{\text{DAM}}$PER) [$\underline{\text{FSL}}$] f_{slip}

Elements:

$list_{DEL}$ = $list_1$ = $\begin{Bmatrix} \underline{\text{ALL}}\ (\underline{\text{MEM}}\text{BERS}) \\ list_2 \end{Bmatrix}$

$list_2$ = 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,

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.

<u>Note</u>:

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 START/END specification, Fiber Specs, Steel Specs, and R-C Specs for reinforced concrete plastic hinges.

The plastic hinge specification begins with the selection of the START/END options, indicating the member ends at which the plastic hinges are permitted to form. 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. The strain in any fiber is assumed to be constant, derived from the assumption of uniform axial elongation plus constant curvature with respect to the principal axes.

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.

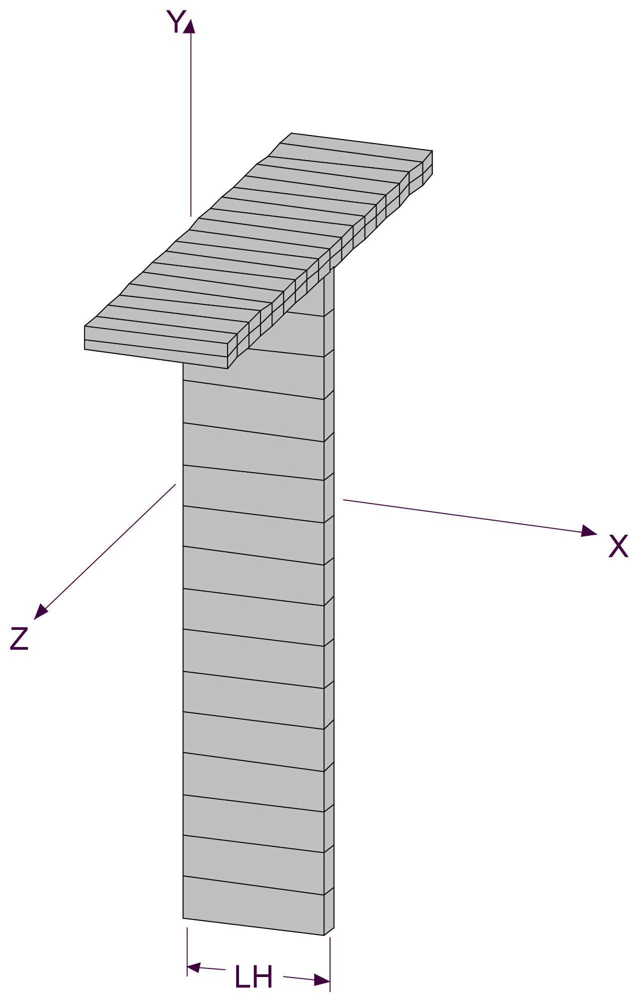


Figure 2.5.2-1 A Tee-Shaped Plastic Hinge, Showing Fiber Divisions

The Steel Specs include the data items FY, ESH, EH, ESU, and FSU, and define the constitutive parameters for 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)/(ESU - 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.

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.

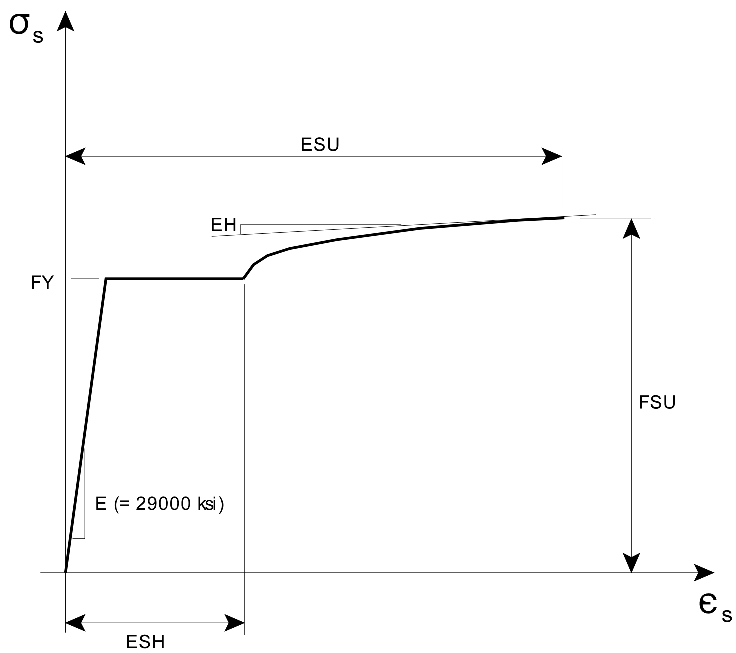


Figure 2.5.2-2 Assumed Steel Stress-Strain Curve for Plastic Hinges

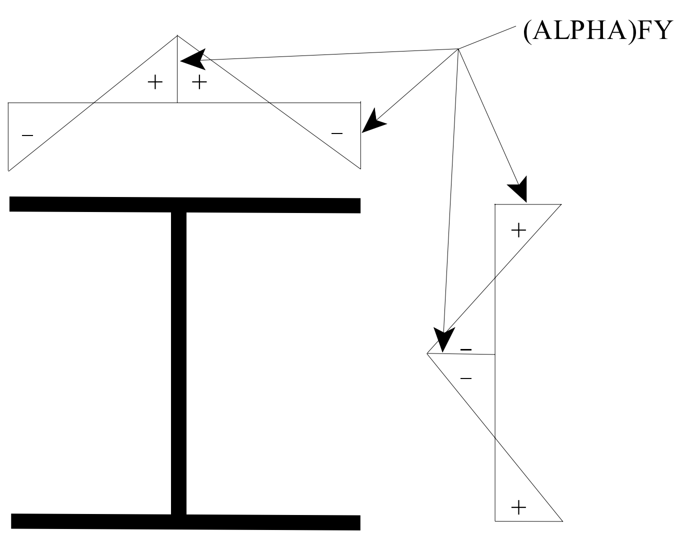


Figure 2.5.2-3 Assumed Residual Stress Distribution for Wide Flange Plastic Hinges

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 concrete material characteristics are defined by the data items FCP, which refers to the ultimate strength of unconfined concrete, EC0, which refers to the strain at which FCP occurs, and FYS, which is the yield stress of the secondary reinforcing: ties, hoops, and spirals.

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 ϵ'_{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 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.

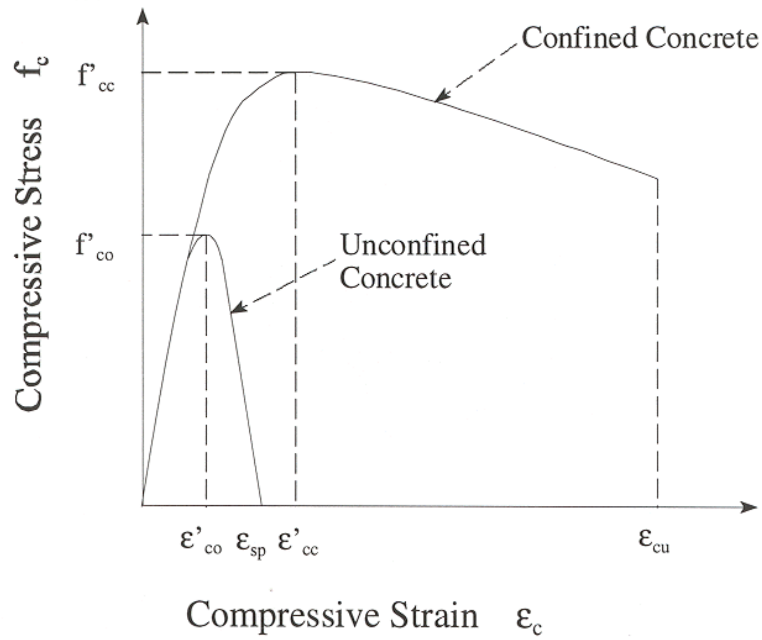


Figure 2.5.2-3 Confined Concrete Compression Stress-Strain Model

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.

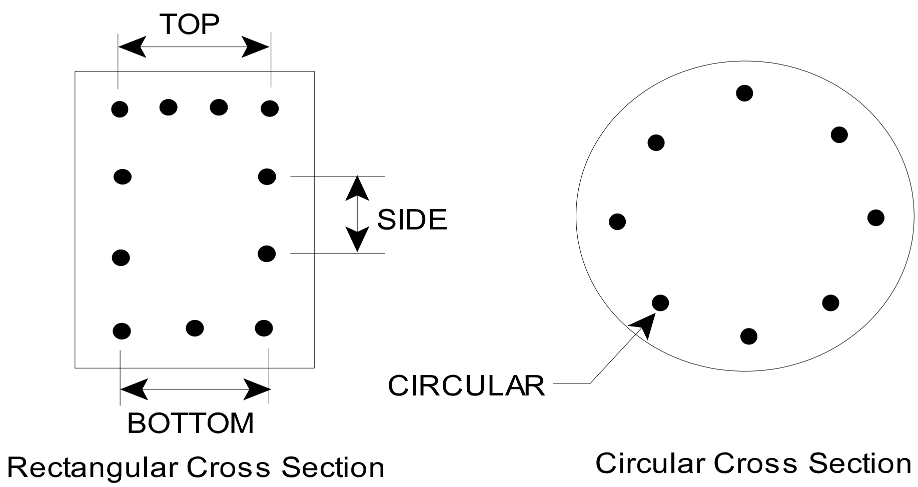


Figure 2.5.2-4 TOP, BOTTOM, SIDE and CIRCULAR BAR Data for Rectangular and Circular Plastic Hinge Cross Sections

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.

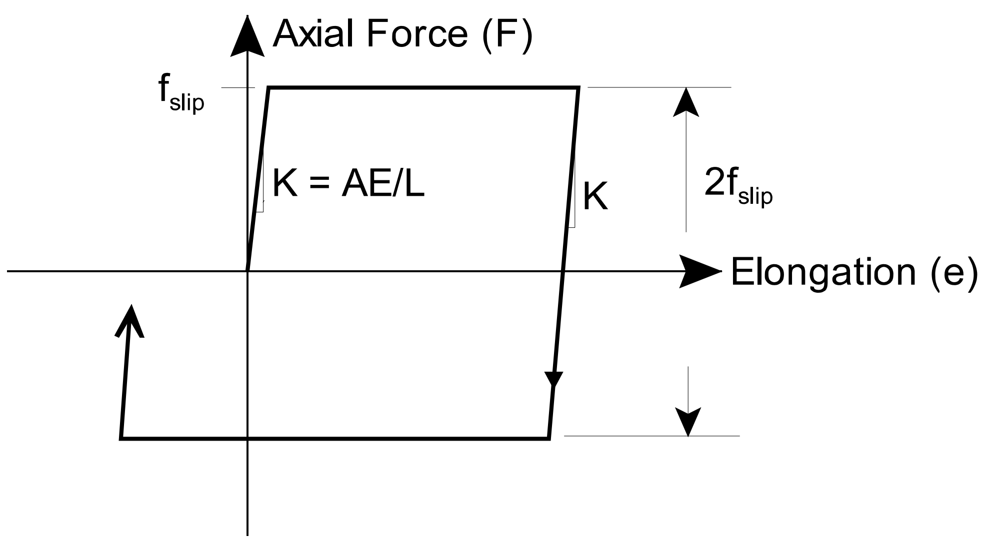
COVER is the final reinforcing bar data item and specifies the decimal clear cover.

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.

Friction 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.

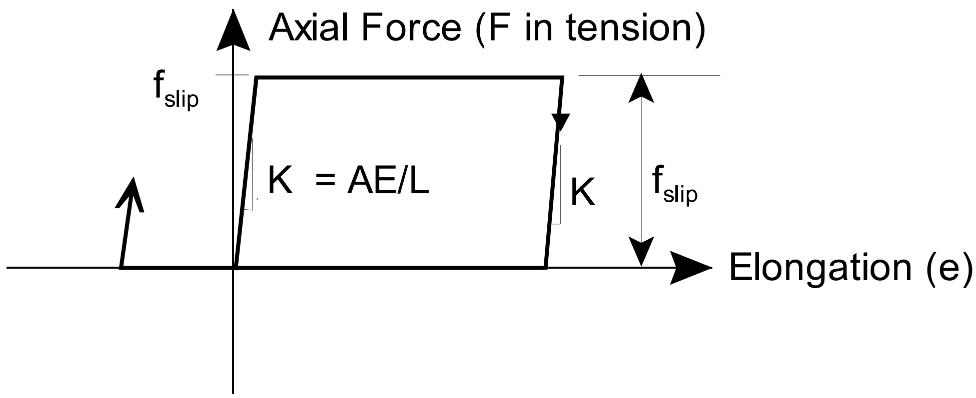


$$-f_{slip} \leq F = (AE/L)e \leq f_{slip}$$

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

Figure 2.5.2-5 Friction Damper Axial Force-Elongation Behavior

Nonlinear GEOMETRY is also in effect automatically for the friction damper effect. The TENSION/COMPRESSION ONLY effect may also be specified in conjunction with the FRICTION DAMPER effect. The FRICTION DAMPER EFFECT in conjunction with the TENSION ONLY effect is illustrated in Figure 2.5.2-6.



$$0 \leq F = (AE/L)e \leq f_{slip}$$

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

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

The FRICTION DAMPER effect may not be specified in conjunction with the PLASTIC HINGE or NLS CONNECTION effects.

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 following 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 Volume 3 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.

5.7 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:

PRINT NONLINEAR EFFECTS (MEMBERS { ALL / list })

Elements:

list = optional list of members.

Explanation:

The PRINT NONLINEAR EFFECTS command is to print 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 is 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:

LIST PLASTIC (HINGE) DISPLACEMENTS (MEMBERS { ALL / list })

Elements:

list = list of member identifiers.

Explanation:

The LIST PLASTIC HINGE DISPLACEMENTS command prints a list of plastic hinge and NLS connection displacements at the start and end of all members for which plastic hinge and NLS connection nonlinearity 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 displacements for plastic hinges include axial displacement, bending about the plastic hinge y axis, and bending about the plastic hinge z axis. The 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. Note however that plastic hinge and NLS connection degrees-of-freedom are not permitted to overlap.

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 and NLS connection members, active and inactive.

Errors:

The following informational massage is printed if no plastic hinge displacement results exist. This will be the case if no nonlinear analysis has been performed or no members with plastic hinge 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 STATUS Command

General form:

<u>LIST</u> <u>PLA</u>STIC (<u>HIN</u>GE) <u>STA</u>TUS (<u>MEM</u>BERS $\begin{Bmatrix} \underline{\text{ALL}} \\ \text{list} \end{Bmatrix}$)

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 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 x (ratio of number of plastic hinge fibers "yielded" to total number 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 results exist. If a member list is not given, then the output is produced for all plastic hinge members, active and inactive.

Errors:

The following informational massage 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 2.5.7-3 on the following page 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 2.5.7-3 Example of Output from the LIST PLASTIC HINGE STATUS Command

5.8 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 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 and the LIST PLASTIC HINGE DUCTILITY RATIO, which are used to list the ductility ratio results from a pushover analysis and are described in Sections 2.6.6.4.

2.6.6.1 The PUSHOVER ANALYSIS DATA and the PERFORM PUSHOVER ANALYSIS Commands

PUSHOVER ANALYSIS DATA -- General form:

PUSHOVER (ANALYSIS DATA)

INCREMENTAL (LOAD) $\left\{ \begin{array}{l} i_{IL} \\ 'a_{IL}' \end{array} \right\}$

CONSTANT (LOAD) $\left\{ \begin{array}{l} i_{CL} \\ 'a_{CL}' \end{array} \right\}$

LOADING (RATE) v_{LRate}

CONVERGENCE RATE v_{CRate}

CONVERGENCE (TOLERANCE) COLLAPSE v_{CTol}

MAXIMUM (NUMBER OF LOAD) TRIALS i_{LT}

MAXIMUM (NUMBER OF LOAD) INCREMENTS i_{LI}

MAXIMUM (NUMBER OF) $\left\{ \begin{array}{l} \text{CYCLES} \\ \text{ITERATIONS} \end{array} \right\}$ i_{CYC}

CONVERGENCE (TOLERANCE) $\left\{ \begin{array}{l} \text{EQUILIBRIUM} \\ \text{DISPLACEMENT} \end{array} \right\}$ v_{ETol}

END (PUSHOVER DATA)

PERFORM PUSHOVER ANALYSIS – General form:

PERFORM PUSHOVER (ANALYSIS) $\left(\begin{array}{l} \text{NJP } i_{NJP} \\ \text{WITHOUT REDUCE (BAND)} \end{array} \right)$

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. Note further that the loading name PAI__000 contains two underscore characters.

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 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 the product of v_{LRate} and 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.

6. Following each successful 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 having convergent nonlinear analyses. 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:

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

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

7. 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.

8. 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 flow chart shown in Figure 2.6.6-1 illustrates the pushover analysis procedure described in Items 1 through 7 above:

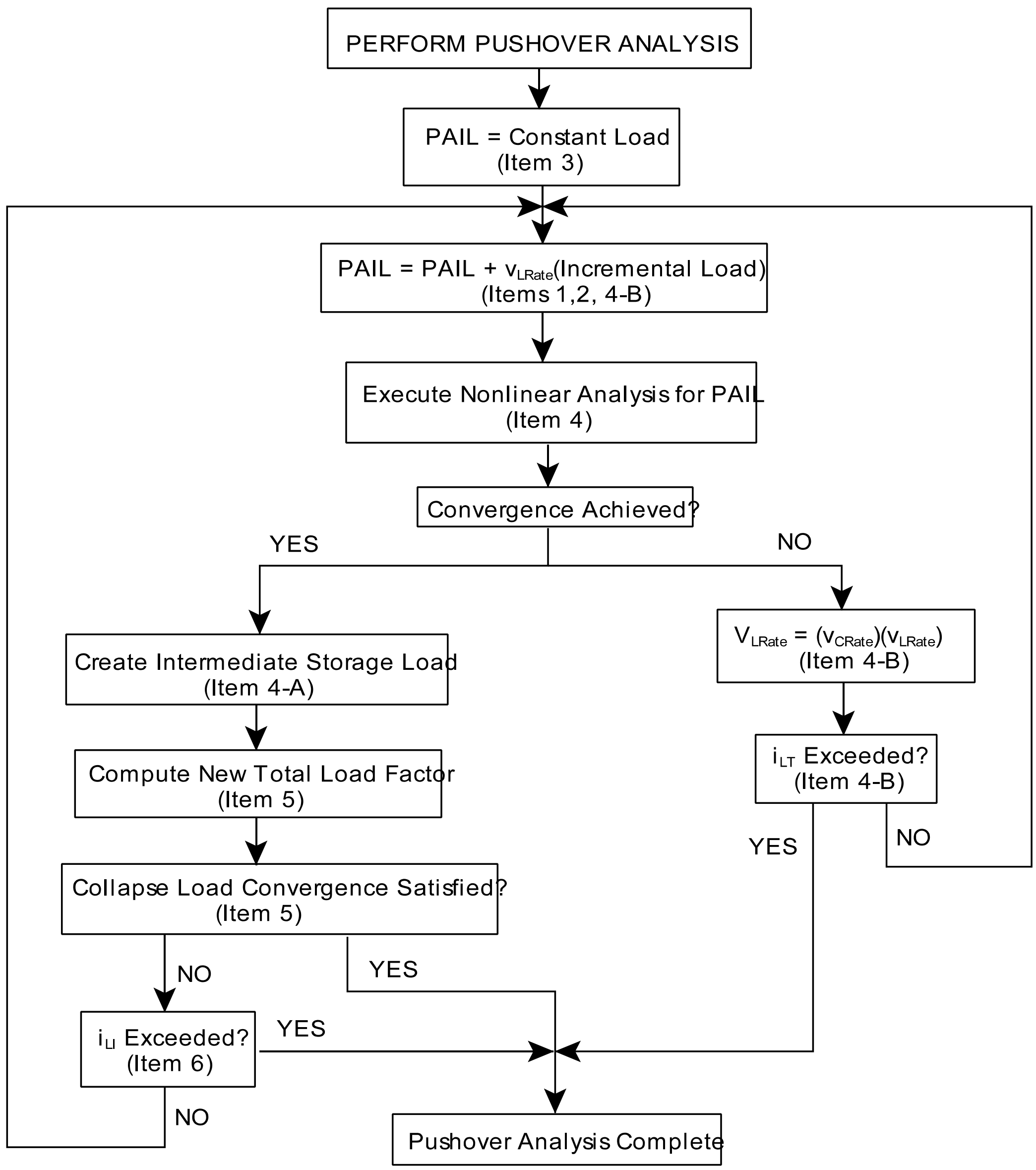


Figure 2.6.6-1 Flow Chart of Pushover Analysis

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. Regardless of the mode, 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 found equilibrium convergence for all 40 load increments. Collapse condition may not be indicated.
Current loading rate = 0.0127

This message means that the nonlinear anlysis converged successfully for each of the maximum number of load increments without meeting the collapse load tolerance. The current loading rate is printed. The pushover analysis is terminated and the current load factor is printed. Also all intermediate incremental storage loads are stored in the load group “IncrLds.”

**** WARNING_STPACP -- The sequence of 10 load adjustment trials in load increment 19 failed to produce equilibrium conver- gence before collapse tolerance = 0.002 was satisfied. Collapse condition may be indicated.
Current loading rate = 0.10297E-5.

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 a stated number of these trials failed to produce a convergent load increment before the collapse tolerance was satisfied. This is a reasonably strong indication that a collapse condition has been reached. The current loading rate is printed.

**** WARNING_STPACP -- Pushover load increment 24 has not converged after 10 load rate adjustment trials. Collapse condition may be indicated or insufficient number of trials specified.
Current loading rate = 0.10976E-3.

The message indicates that the specified maximum number of load adjustment trials, i_{LT}, is not sufficient to find a new load rate that produces a new applied load for which nonlinear analysis converges. The current loading rate, which is printed, may be small enough to also suggest that this situation indicates a collapse condition.

**** INFO_STPACP -- Collapse tolerance = 0.002 has been satisfied in load increment 33.
Collapse condition may be indicated.
Current loading rate = 0.43794E-5

This message is printed if the collapse tolerance, in this case 0.002, is achieved prior to the total number of load increments being executed. This may indicate a collapse condition, particularly if the collapse tolerance is sufficiently small.

**** 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.

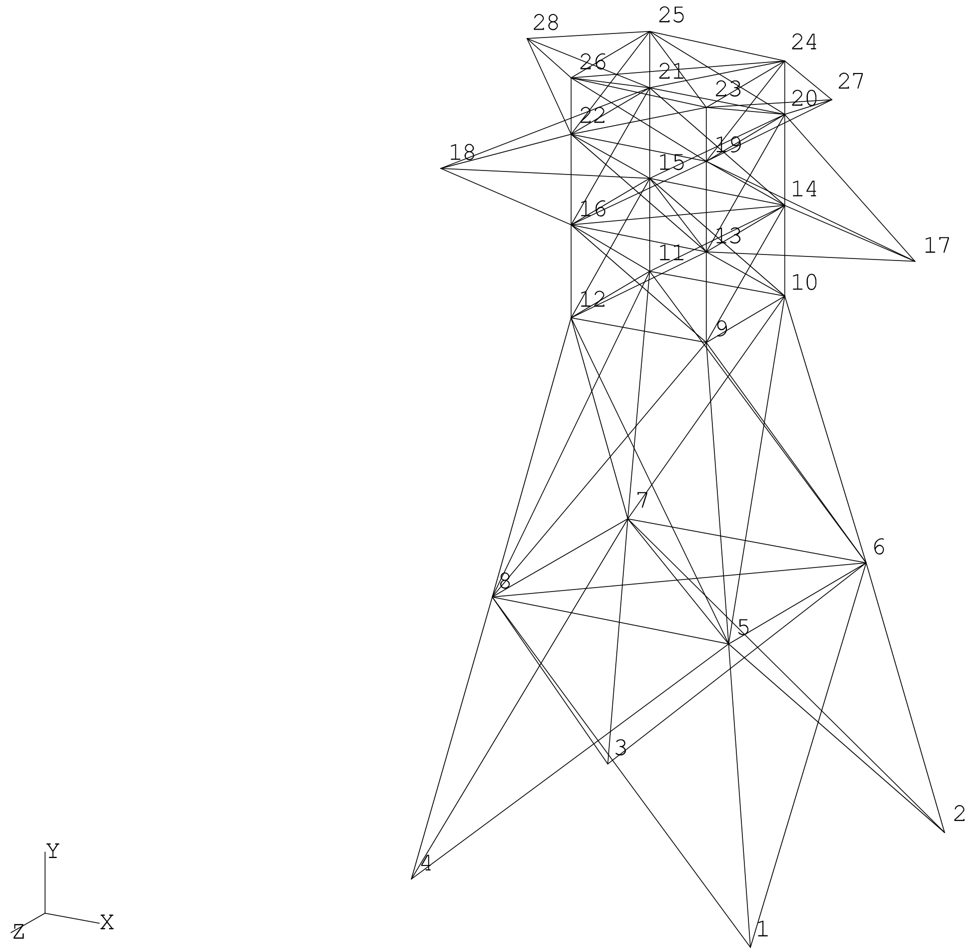


Figure 2.6.6-1 Simple Transmission Tower for Pushover Analysis Example PA-1

```
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 an 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:

$$\underline{\text{LIST}}\ \underline{\text{PUSH}}\text{OVER}\ (\underline{\text{ANA}}\text{LYSIS}\ \underline{\text{DUC}}\text{TILITY}\ \underline{\text{RAT}}\text{IO})\ \left\{\begin{array}{l}\underline{\text{TX}}\\ \underline{\text{TY}}\\ \underline{\text{TZ}}\\ \underline{\text{RX}}\\ \underline{\text{RY}}\\ \underline{\text{RZ}}\end{array}\right\}\ \underline{\text{TAR}}\text{GET}\ (\underline{\text{JOI}}\text{NT})\ \left\{\begin{array}{l} i_T \\ 'a_T' \end{array}\right\}$$

Elements:

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 member nonlinearity as described in the NONLINEAR EFFECTS command (Section 2.5.2, Volume 3, GTSTRUDL 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 any amount of plastic hinge formation was 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 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 push-over 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 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:

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.

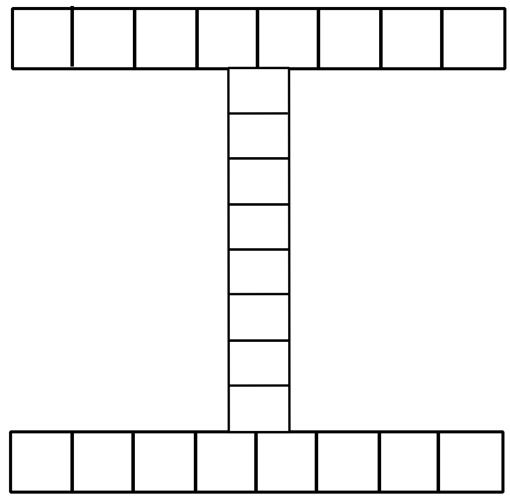


Figure 2.6.6-6 Plastic Hinge Cross Section Fiber Grid, Example PA-2

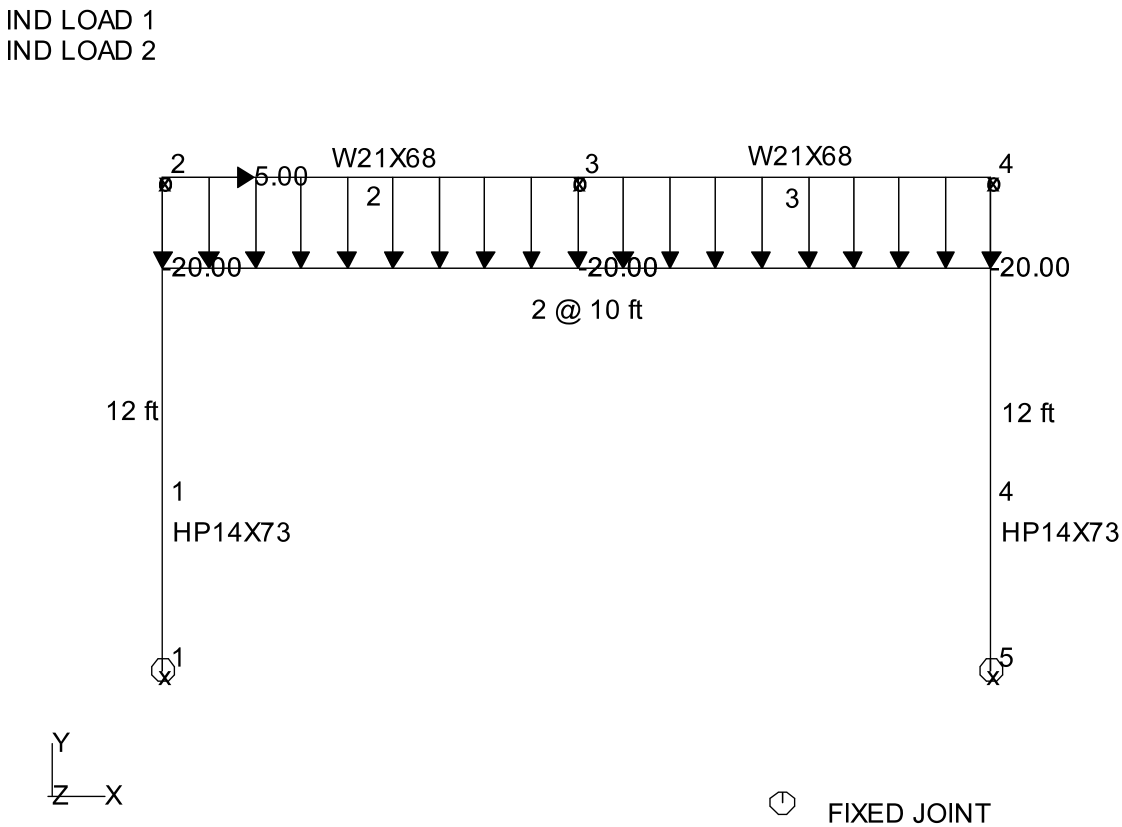


Figure 2.6.6-7 Portal Frame Model for Example PA-2

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
$   Number 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 1.0
  CONVERGENCE RATE 0.5
  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

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.

```
{  100} >
{  101} > LIST PUSHOVER DUCTILITY RATIO TX TARGET JOINT 2

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


Pushover Analysis Ductility Ratio
=================================
  Target joint = 2        DOF = TX
    Ductility Ratio =  7.624940
```

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:

$$\underline{\text{LIST}}\ \underline{\text{PLA}}\text{STIC}\ (\underline{\text{HIN}}\text{GE})\ \underline{\text{DUC}}\text{TILITY}\ (\underline{\text{RAT}}\text{IO})\ \left\{\begin{matrix}\underline{\text{TX}}\\ \underline{\text{RY}}\\ \underline{\text{RZ}}\end{matrix}\right\}\ (\underline{\text{MEM}}\text{BER list})$$

Elements:

list = list of members. If not given, all members having plastic hinge 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 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 Eq. 2.6.6-1 above, where the displacements are the member end plastic hinge displacements rather than global joint displacements. Because the plastic hinges have only three local degrees-of-freedom, displacement x, rotation y, and rotation z, the degree-of-freedom options provided by the command are limited to TX (displacement x), RY (rotation about the local y axis), and RZ (rotation about the local z axis).

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 push-over 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 corrected 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, "—," indicate 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.

```
{  102} > LIST PLASTIC HINGE DUCTILITY RATIO RZ MEMBERS 1 TO 4

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

```
Plastic Hinge Ductility Ratios
===============================

                      Ductility Ratios -- Displacement = RZ
 Member                    Start             End
 ------                   -------          -------


1                         ---              ---
2                         ---               1.346
3                          1.346            1.143
4                          1.665            5.766
```

Figure 2.6.6-10 Plastic Hinge Ductility Ratio Results, Example PA-2

This page intentionally left blank.

5.9 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

CALCULATE ERROR (ESTIMATE) (BASED ON) –

* ENERGY (NORM)
  MAX DIFFERENCE
  DIFFERENCE FROM AVERAGE
  PERCENT MAX DIFFERENCE
  PERCENT DIFFERENCE FROM AVERAGE
  NORMALIZED PERCENT MAX DIFFERENCE
  NORMALIZED PERCENT DIFFERENCE FROM AVERAGE –

(AT) * TOP / MIDDLE / BOTTOM (SURFACES) (FOR) { → ALL / ELEMENT list }

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:

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

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

$$\eta = \frac{\| e_{\sigma} \|_{L2}}{\| \sigma \|_{L2} + \| e_{\sigma} \|_{L2}} \times 100$$

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

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

Difference from Average Method

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

Percent Maximum Difference Method

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

Percent Difference from Average Method

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

Normalized Percent Maximum Difference

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

Normalized Percent Difference from Average Method

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

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.

5.10 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 as shown in Section 3.3 of this Release Guide. 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.

5.11 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) $\begin{Bmatrix} \to \text{OFF} \\ i_{ISL} \\ 'a_{ISL}' \end{Bmatrix}$
    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 loaidng 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:

DYNAMIC ANALYSIS NONLINEAR $\left\{ \begin{array}{l} \text{BETA } v_b \\ \text{NJP } i_{NJP} \end{array} \right\}^*$

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 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.

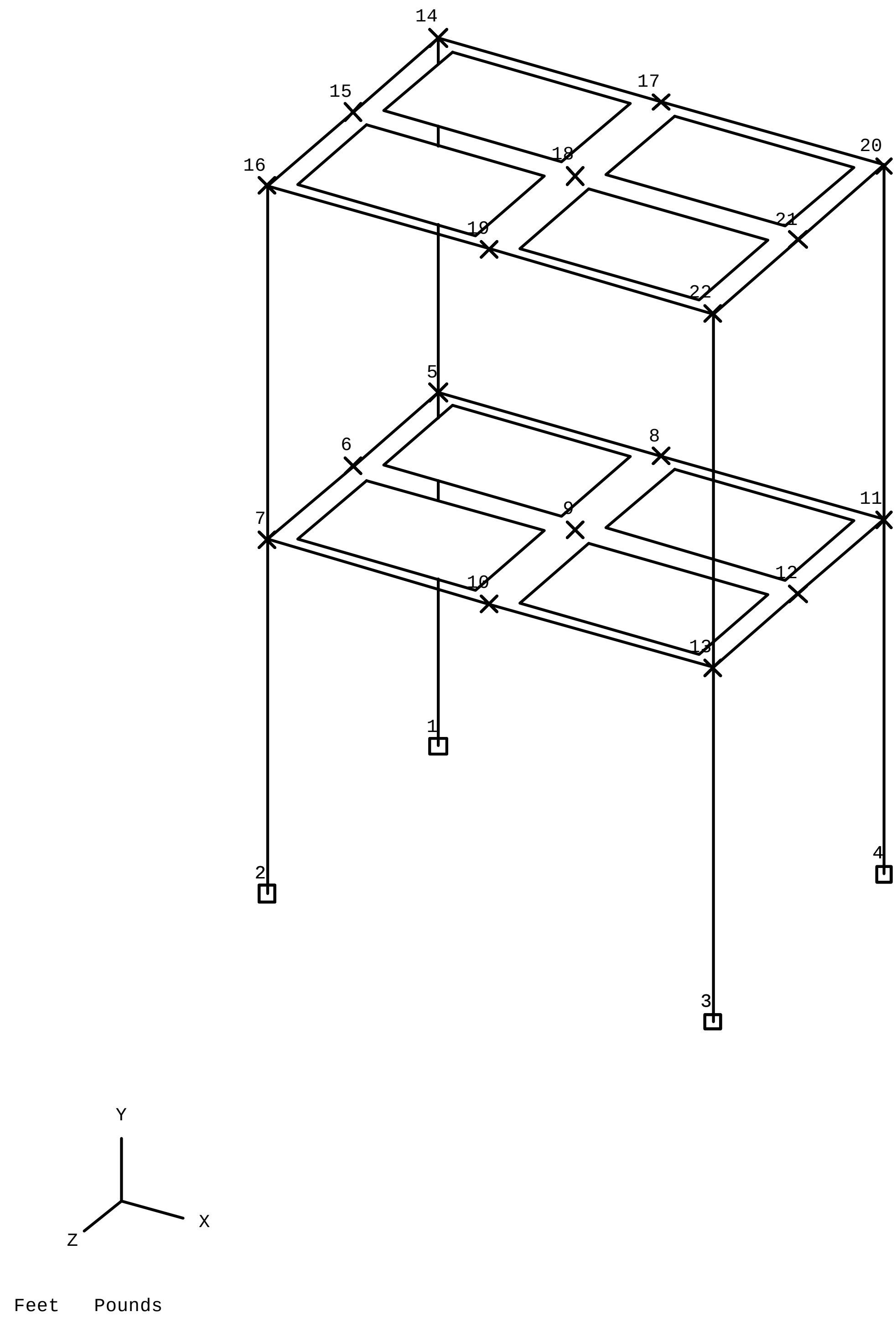


Figure 2.5.6.3-1 Nonlinear Dynamic Analysis Example Structure – Joints Labeled

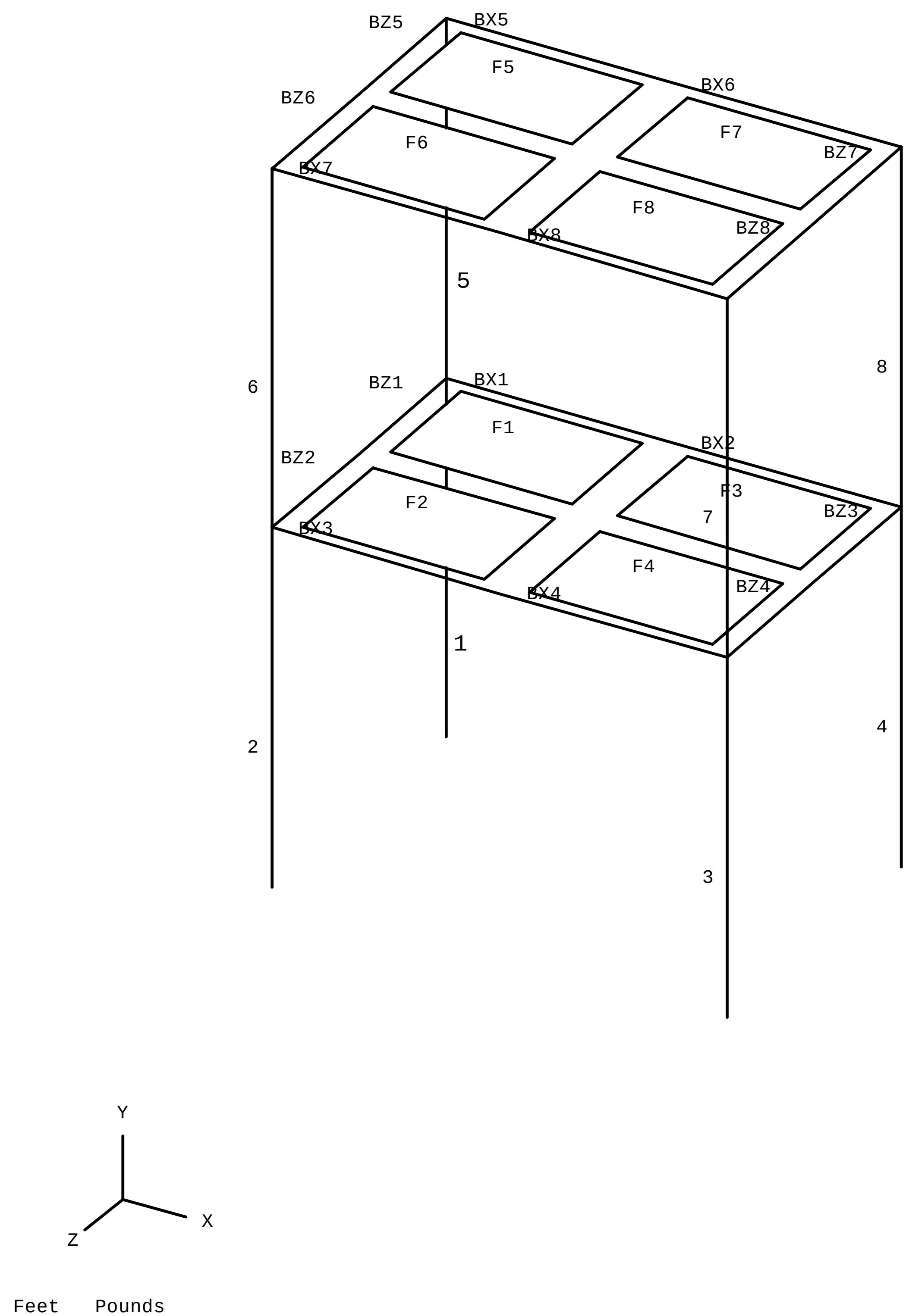


Figure 2.5.6.3-2 Nonlinear Dynamic Analysis Example Structure – Members and Finite Elements Labeled

```
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 MATRIX 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.

5.12 The Locate Interference and Duplicate Joints Command

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 are two common problems with this conversion, which 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:

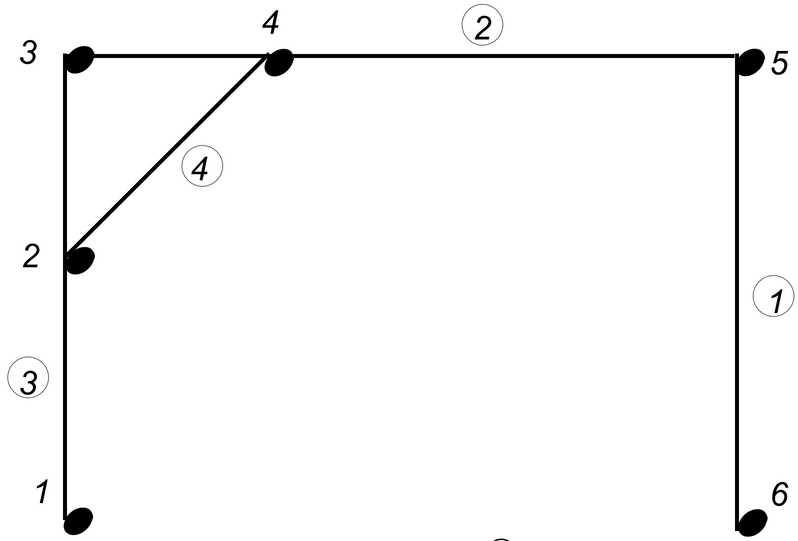
MB_Int This group is a list of all members that had one or more interfering joints. It can be useful to use the command LABEL MEMBER GRP 'MB_Int' while in the Scope environment to see which members have interfering joints. Note that the group name is case-sensitive: 'mb_int' or 'MB_INT' will not work.

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:

LOCATE DUPLICATE JOINTS (TOLERANCE v_2) (AND) (REMOVE)

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.

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

General form:

$$\underline{\text{FOR}}\text{M}\ \ \underline{\text{STAT}}\text{IC}\ \ (\underline{\text{EAR}}\text{THQUAKE})\ \ \underline{\text{LOA}}\text{D}\ \ \begin{matrix} 'a_{sl}' \\ i_{sl} \end{matrix}\ \ ('\text{title}_{sl}')\ \ -$$

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

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.
$'\text{title}_{sl}'$	=	optional static load title of up to 64 characters in length.
v_x	=	scaling factor for the mass load in the global X direction. v_x is taken as 0.0 by default.
v_y	=	scaling factor for the mass load in the global Y direction. v_y is taken as 0.0 by default.
v_z	=	scaling factor for the mass load in the global Z direction. v_z is taken as 0.0 by default.
$'a_{RS}'/i_{RS}$	=	name of the response spectrum load to be used for the calculation of static earthquake load $'a_{sl}'/i_{sl}$.
v_{RS}	=	scaling factor to be applied to the response spectrum static earthquake load $'a_{sl}'/i_{sl}$.

Explanation:

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

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

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

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

$$\left\{ F_{EM} \right\} = \left[M \right] \left\{ v_{XYZ} \right\} g \qquad \text{Eq. 5.13-1}$$

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 5.13-1, 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).

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

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:

$$\left\{ f_{RS} \right\}_i = -v_{RS}\, \Gamma_i\, S_{ai} \left[M \right] \left\{ \Phi_i \right\} \qquad \text{Eq. 5.13-2}$$

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,
$\left\{ \Phi_i \right\}$	=	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.

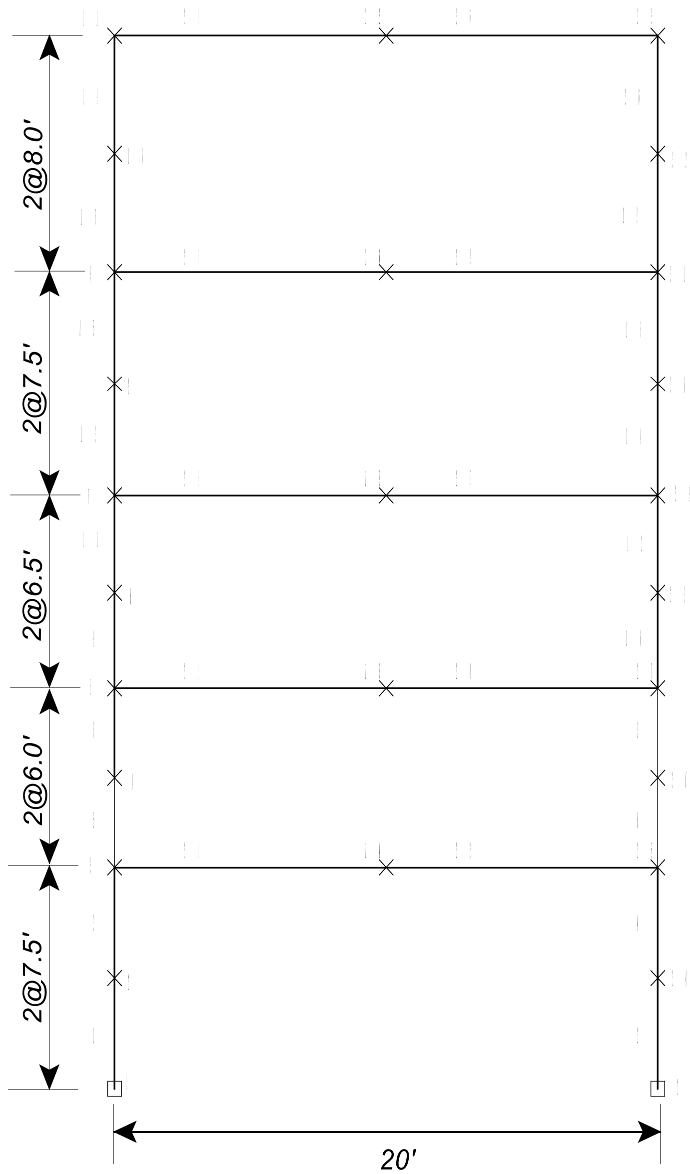
```
**** ERROR_STGELL -- Results do not exist for response
                     spectrum loading 1-G.2.  Response
                     spectrum analysis has not yet been run
                     or the specified loading is not a
                     response spectrum load.  SCAN MODE
                     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 5.13-1 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 5.13-2.



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 5.13-1 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 5.13-2 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 5.13-2 Command Listing for Example SEL-1 (Continued)

Figure 5.13-3 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 5.13-3 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 5.13-3 PRINT APPLIED JOINT LOAD Results for Example SEL-1 (Continued)

5.14 Reference Coordinate System Command

General form:

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

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

Explanation:

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

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

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

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

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

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

5.14-1 Printing Reference Coordinate System Command

General form:

PRINT REFERENCE (COORDINATE) (SYSTEM) $\left\{ \begin{array}{l} \rightarrow \text{ALL} \\ \text{list} \end{array} \right.$

Explanation:

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

This page intentionally left blank.

5.15 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

5.16 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:

```
                          ⎧ HASHED     ⎫
SET ELEMENTS              ⎨            ⎬
                          ⎩ SEQUENTIAL ⎭
```

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.