GT STRUDL ®

Version 30

Release Guide

Volume 1 of 2

April 2009

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 30, with a release date in the GTSTRUDL title block of April 2009.

The GTSTRUDL® computer program is proprietary to, and a trade secret of the Georgia Tech Research Corporation, Atlanta, Georgia, U.S.A.

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

DISCLAIMER

Commercial Software Rights Legend

Table of Contents

Chapter 1

Introduction

Version 30 covers GTSTRUDL operating on PC's under the Windows Vista, Windows XP, Windows 2000 and Windows NT operating systems. Chapter 2 presents the new features and enhancements which have been added since the release of Version 29.1. Chapter 3 provides you with details regarding error corrections that have been made since the Version 29.1 release. Chapter 4 describes known problems with Version 30. Chapter 5 describes prerelease features -- new features which have been developed and subjected to limited testing, or features for which the user documentation has not been added to the GTSTRUDL User Reference Manual. The command formats and functionality of the prerelease features may change before they become supported features based on additional testing and feedback from users.

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

5.2 Design Prerelease Features

- 5.2.2 AISC13 Steel Design Code and Parameters
- 5.2.2 LRFD3 Steel Design Code and Parameters
- 5.2.3 ACI Code 318-99
- 5.2.4 Rectangular and Circular Concrete Cross Section Tables
- 5.2.5 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

5.3 Analysis Prerelease Features

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

5.4 General Prerelease Features

5.4.1 Rotate Load Command

5.4.2 Reference Coordinate System Command

5.4.3 GTMenu Point Coordinates and Line Incidences Commands

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

Chapter 2

New Features in Version 30

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 30. This release guide is also available online upon execution of GTSTRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 Dynamics

1. Space frame members may now have member releases including elastic connections when performing a dynamic analysis with an INITIAL STRESS loading.

2. The CREATE TIME HISTORY command has been extended as shown below:

$$\underline{\text{CRE}}\text{ATE }\ \underline{\text{TIM}}\text{E }\ (\underline{\text{H}}\text{ISTORY})\ \ \underline{\text{FIL}}\text{E }\ \text{'filename}_{\text{new}}\text{'}\ \ (\underline{\text{FR}}\text{OM}) \begin{Bmatrix} \text{joint specs} \\ \text{time segment specs} \end{Bmatrix}$$

where,

$$\text{joint specs} = \underline{\text{JOI}}\text{NT} \begin{Bmatrix} \text{'}a_1\text{'} \\ i_1 \end{Bmatrix} \ \underline{\text{TRAN}}\text{SLATION} \begin{Bmatrix} \underline{\text{X}} \\ \underline{\text{Y}} \\ \underline{\text{Z}} \end{Bmatrix} \ \underline{\text{LOA}}\text{DING} \begin{Bmatrix} \text{'}a_2\text{'} \\ i_2 \end{Bmatrix}$$

$$\text{time segment specs} =$$

$$\underline{\text{TIM}}\text{E }\ \underline{\text{SEG}}\text{MENT }\ (\underline{\text{OF}})\ \ \underline{\text{FIL}}\text{E }\ \text{'filename}_{\text{src}}\text{'}\ \ \underline{\text{STA}}\text{RT }\ (\underline{\text{TIM}}\text{E})\ v_S\ -$$

$$\underline{\text{END}}\ (\underline{\text{TIM}}\text{E})\ v_E\ \ \underline{\text{RAM}}\text{P }\ (\underline{\text{FUN}}\text{CTION}) \begin{Bmatrix} \underline{\text{LIN}}\text{EAR} \\ \underline{\text{COS}}\text{INE} \end{Bmatrix} \ \underline{\text{BUI}}\text{LD }\ (\underline{\text{TIM}}\text{E})\ v_B$$

The new feature is provided by the *time segment specs*, which allow you to create a new time history file by extracting a segment of an existing time history file and factoring an initial portion of that segment with a ramp function. This feature allows you to isolate a much shorter, perhaps critical, portion of a longer kinematic or force/moment time history function in order to perform, for example, a preliminary transient analysis. This feature is described in further detail in Section 2.4.8.1 of Volume 3 of the GTSTRUDL Reference Manual.

3. A number of modifications have been made to improve the efficiency of the computation of response spectrum CQC and DSM mode combinations when the results are stored into files on the hard drive (when the EXTERNAL FILE SOLVER is turned on). The following tabulation of statistics illustrates the impact of the efficiency improvements:

# of joints	2378
# of plate finite elements	2460
# of frame members	729
# of DOFs	13662

The time to compute CQC mode combination results (displacements, accelerations, member and finite element nodal forces, finite element stresses, resultant joint loads, and support reactions) for three response spectrum loads is shown below:

Previous version(s)	7930 seconds
Version 30	300 seconds

4. Dynamic analysis external file solver support, as specified by the DYNAMIC PARAMETERS command, has been added to harmonic analysis. Harmonic analysis with the external file solver enabled is now able to solve significantly larger jobs at measurably improved speeds over the existing harmonic analysis.

5. Response spectrum Analysis has been extended to provide for the computation of modal response and modal combinations in accordance with NRC Regulatory Guide 1.92, Revision 2. The RESPONSE SPECTRUM LOAD/MODE FACTORS command has been extended to allow the computation of the Gupta Method rigid and periodic modal response factors. The COMPUTE RESPONSE SPECTRUM, LIST RESPONSE SPECTRUM, and CREATE PSEUDO STATIC LOAD commands have been extended to provide for the computation and reporting of ALGebraic modal response combinations.

2.2 General

1. A new feature, CALCULATE PRESSURE, has been added to report a calculated pressure for each joint incident to the specified finite elements, based on spring forces at the joint and the tributary area of the joint. This command approximates the pressure felt by a continuous supporting material, such as soil under a foundation or concrete bearing surface supporting a base plate. The sign of the reported

pressure is the same sign as the reaction in the specified direction. This corresponds to a positive pressure for a base plate at the bottom of a column under compression.

Command Syntax:

$$\underline{\text{CAL}}\text{CULATE}\ \underline{\text{PRE}}\text{SSURE}\ \underline{\text{PLA}}\text{NE} \begin{Bmatrix} \underline{\text{X}} \\ \underline{\text{Y}} \\ \underline{\text{Z}} \end{Bmatrix} (\underline{\text{EQ}}\text{UAL})\ v_1 -$$

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

$$(\underline{\text{ELE}}\text{MENTS}\ \mathit{list})\ \big((\underline{\text{SHO}}\text{W})\ \underline{\text{CAL}}\text{CS})\ (\underline{\text{SUM}}\text{MARY}\ (\underline{\text{ONL}}\text{Y})\big)$$

The CALCULATE Pressure command is documented in Section 2.1.12.19 of Volume 1 of the GTSTRUDL Reference Manuals.

2. Error processing and output messages have changed for the SAVE command. Messages now conform to the GTSTRUDL standard message format. If a SAVE file is unable to be created or opened in write mode due to permission status, Read Only status or illegal file name, GTSTRUDL will issue a warning, ignore the SAVE command and continue processing commands.

Examples of new informational messages:

```
 {   46} > SAVE 'FR386_ArchiveSave_1.gts'

****INFO_XXSAVE - Data base saved in file
C:\30\Testing\FR386_ArchiveSave_1.gts

{   47} >
{   48} > $ ---- SAVE with overwrite message
{   49} > SAVE 'FR386_ArchiveSave_1.gts'

****INFO_XXSAVE - File exists and will be overwritten
                  Data base saved in file
C:\30\Testing\FR386_ArchiveSave_1.gts
```

Examples of new failure messages:

```
{   53} > SAVE 'FR386_ArchiveSave_1.gts'

 ****ERROR_XXSAVE - Unable to overwrite specified save file.
                    Check file and/or folder permissions.
  File: FR386_ArchiveSave_1.gts
```

```
{   14} > SAVE 'Bad?File.gts'

****ERROR_XXSAVE - Unable to open the specified save file.
                   Check folder permissions and/or file name.
     File: Bad?File.gts
```

3. Command Archiving has been added to GTSTRUDL. All commands interactively typed, created by dialogs or menu picks, or read from a CINPUT file are now recorded in the GTSTRUDL database. This will allow you to re-create a command sequence from a SAVE (.gts) file. Note that data created or edited in GTMenu will **not** be recorded, although a warning entry will be added to the Command Archive in the case when GTMenu is exited with a translation. Command Archiving is turned on by default when GTSTRUDL is initialized, but you can turn it off with the ARCHIVE OFF command. The Command Archive can be deleted with the DELETE COMMAND ARCHIVE command. The archived commands can be written to a file with the PRINT COMMANDS command. If no file name is specified, the file is named "GTS_commands.gti" by default.

   Syntax:

   PRINT (ARCHIVED) COMMANDS ('*filename*')

   ARCHIVE (COMMANDS) OFF
   ARCHIVE (COMMANDS) ON

   DELETE COMMAND (ARCHIVE)

   Examples:

```
{   40} > PRINT COMMANDS 'FR386_Archive_4'

****INFO_CMDUTL -   Archived commands were written to file
                    FR386_Archive_4.gti

{   41} >
{   42} > $ ---- Notice of overwrite
{   43} > PRINT COMMANDS 'FR386_Archive_4'

****INFO_CMDUTL -   Specified file already exists and will be overwritten.
                    Archived commands were written to file
                    FR386_Archive_4.gti

{   62} > $ --- Archive options
{   63} > ARCHIVE COMMANDS OFF

****INFO_CMDARC -  Command Archiving is turned OFF.
```

```
{    64} > $ This comment should not be in the archive because of OFF
{    65} > $ Or this one
{    66} > ARCHIVE ON

****INFO_CMDARC -  Command Archiving is turned ON.

{    71} > DELETE COMMAND ARCHIVE

****INFO_CMDARC -  All previous commands are deleted from the Command
                   Archive.
```

The PRINT COMMAND ARCHIVE command is documented in Section 2.1.14.2.2, Volume 1 of the GTSTRUDL User Reference Manual. The Command Archive Options are documented in Section 2.1.12.18, Volume 1 of the Reference Manuals.

4. A new special-purpose independent loading command, FORM NOTIONAL LOAD, has been implemented. The FORM NOTIONAL LOAD provides for the automatic creation of an independent static notional load condition in accordance with the Direct Analysis Method of the 2005 AISC 13th Edition Specifications for Structural Steel Buildings. The command format is shown below:

$$\underline{\text{FOR}}\text{M } \underline{\text{NOT}}\text{IONAL } (\underline{\text{LOA}}\text{D}) \begin{Bmatrix} i_{NL} \\ 'a_{NL}' \end{Bmatrix} \text{ from specs } \underline{\text{GRA}}\text{VITY } (\underline{\text{AXI}}\text{S}) \begin{Bmatrix} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{Bmatrix} \text{ -}$$

$$\underline{\text{NLDIR}} \begin{Bmatrix} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{Bmatrix} \underline{\text{NLF}}\ v_{NL}\ (\underline{\text{JOI}}\text{NTS list}_J)$$

$$\text{from specs } = \underline{\text{FRO}}\text{M} \begin{Bmatrix} i_1 \\ 'a_1' \end{Bmatrix} v_1 \left(\begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} v_2 \begin{Bmatrix} i_3 \\ 'a_3' \end{Bmatrix} v_3 \ldots \begin{Bmatrix} i_n \\ 'a_n' \end{Bmatrix} v_n \right)$$

The FORM NOTIONAL LOAD command is documented is Section 2.1.11.3.8 of Volume 1 of the Reference Manuals.

5. The DEAD LOAD and SELF WEIGHT LOAD commands now issue a warning message identifying members for which DENSITY = 0.0. Previous versions issued no such message.

6. A warning message is now issued when Member Releases have been specified for a member but no member release data has been given as shown in the following example:

   MEMBER RELEASES
   1 2 $ Note that no release data has been given

   This feature was released in Version 29.2 and is reported here also.

7. A new command has been implemented to locate duplicate members and if desired, remove the duplicate members from the model. The new LOCATE DUPLICATE MEMBER command is shown below:

$$\underline{\text{LOC}}\text{ATE }\underline{\text{DUP}}\text{LICATE }\underline{\text{MEM}}\text{BERS }\textit{order}\text{ -}$$

$$\text{((}\underline{\text{AND}}\text{) }\underline{\text{REM}}\text{OVE (}\left\{\begin{array}{l}\underline{\text{ADD}}\ \underline{\text{LOA}}\text{DS}\\ \underline{\text{FOR}}\text{CE (}\underline{\text{REM}}\text{OVAL)}\end{array}\right\}\text{) -}$$

$$\text{(}\underline{\text{MEM}}\text{BER }\textit{list}\text{)}$$

$$\text{order} = \left\{\begin{array}{l}\rightarrow \underline{\text{DIS}}\text{REGARD (}\underline{\text{INC}}\text{IDENCE) (}\underline{\text{ORD}}\text{ER)}\\ \quad\underline{\text{RES}}\text{PECT (}\underline{\text{INC}}\text{IDENCE) (}\underline{\text{ORD}}\text{ER)}\end{array}\right\}$$

   The command is documented in Section 2.1.12.11.3 of Volume 1 of the Reference Manuals.

8. The SET ELEMENTS HASHED command has been moved to release status. The SET ELEMENTS HASHED command will significantly improve the processing time required to read large input files (.gti files). The command is documented in Section 2.l.1.9 of Volume 1 of the Reference Manuals.

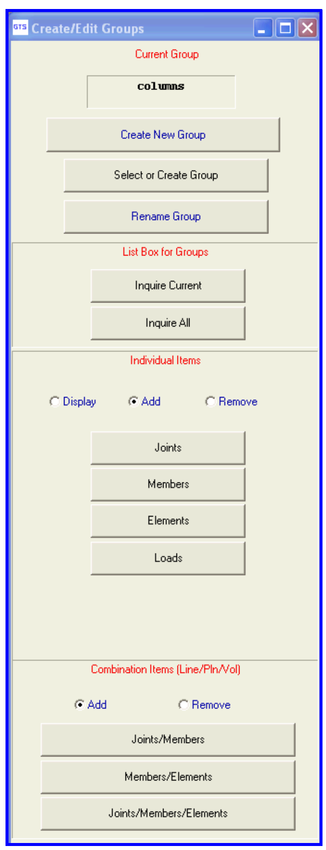
9. A new command, ACTIVE SOLVER, as been implemented with the following syntax:

$$\underline{\text{ACTI}}\text{VE } \underline{\text{SOLV}}\text{ER} \left\{ \begin{array}{l} \underline{\text{GTSES}} \\ \underline{\text{GT64M}} \\ \underline{\text{STAND}}\text{ARD} \end{array} \right\}$$

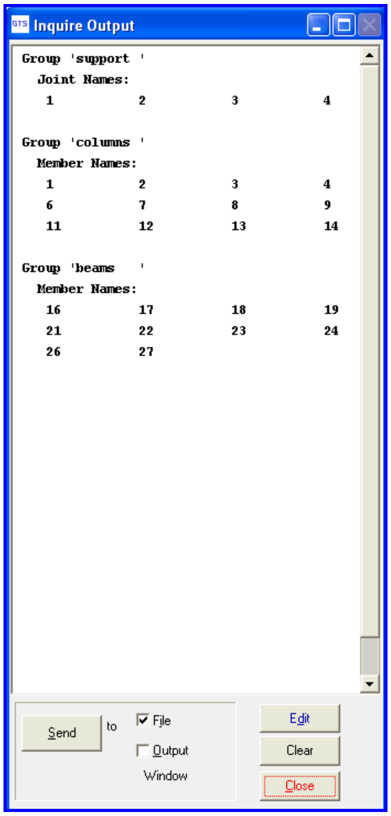
The purpose of this command is to facilitate the integration of the newest high performance equation solvers and analysis results data storage strategies when static and dynamic analyses are executed in the same job. This new command is described in detail in Section 2.1.13.4 of Volume 1 of the Reference Manuals. The new GT64M solver is described later in this Release Guide in Section 2.11.

2.3 GTMenu

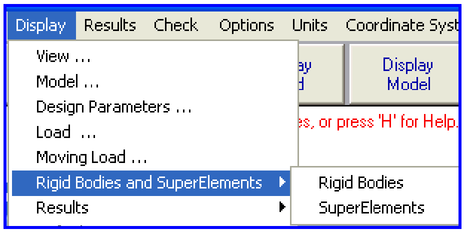
1. The Create Groups dialog now allows you to Inquire about the contents of the current group or for all groups. The revised Create Groups dialog is shown below:



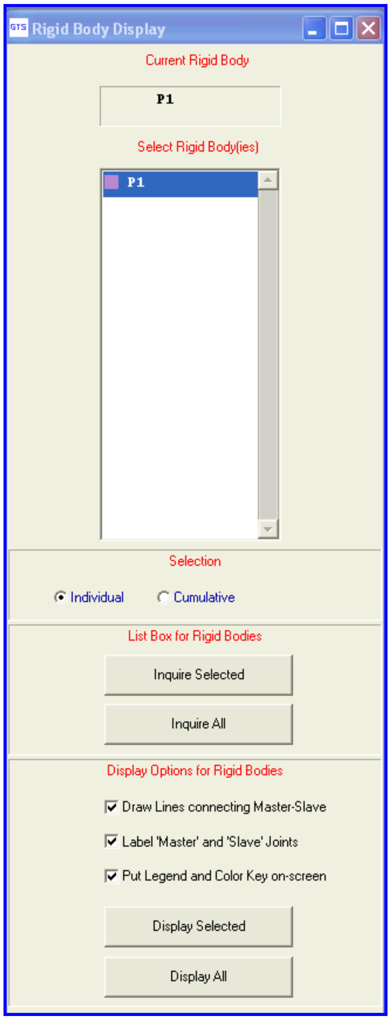
An example of the Inquire All Groups output is shown below:



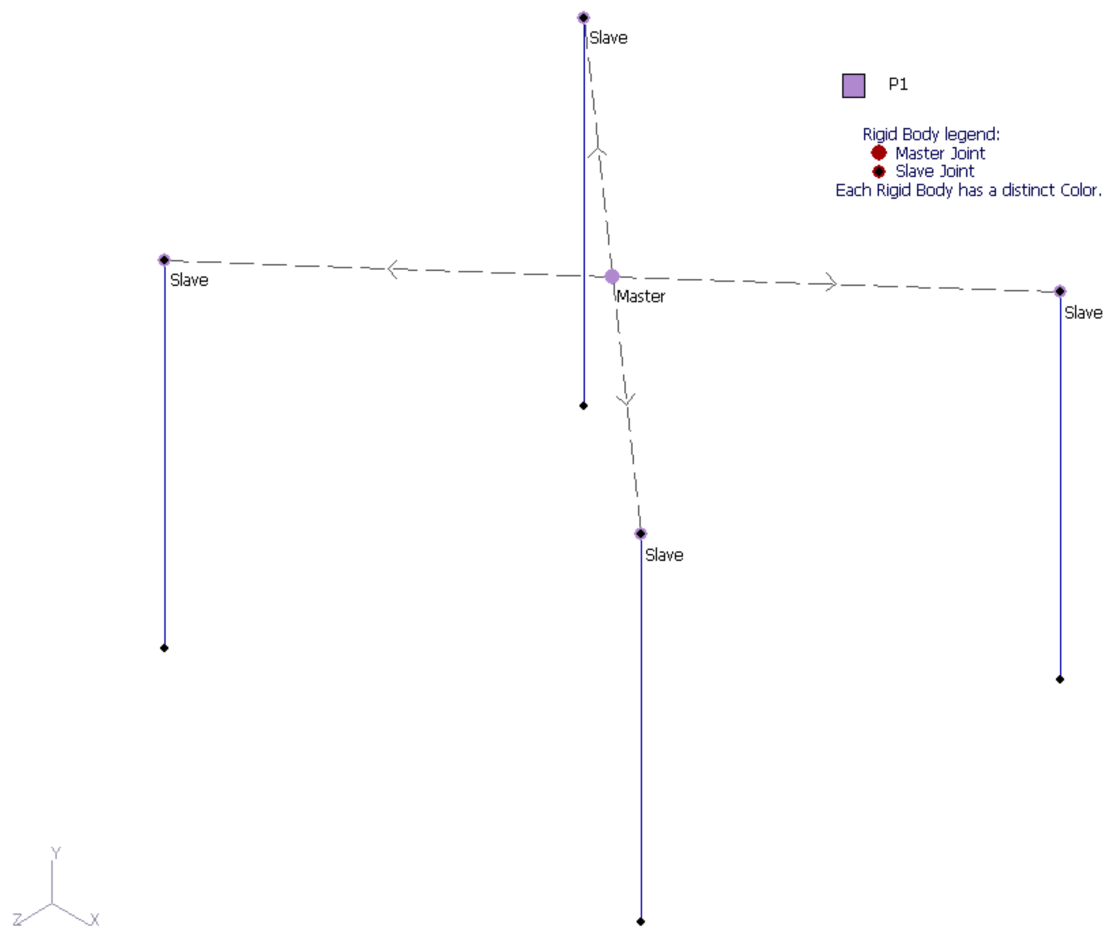
2. Rigid Body and Superelement information may now be graphically displayed in GTMenu. The Display pulldown now has options for Rigid Bodies and Superelements as shown below:



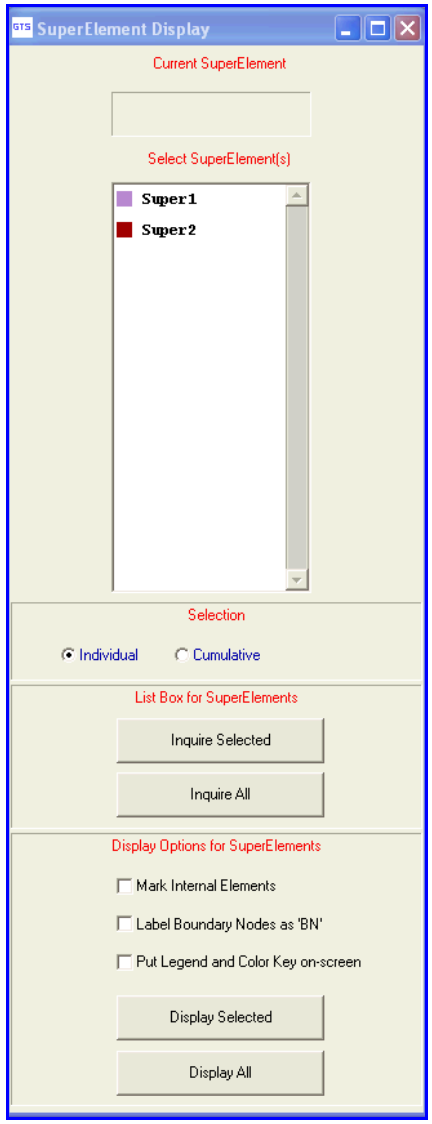
The new Rigid Body Display dialog is shown below:



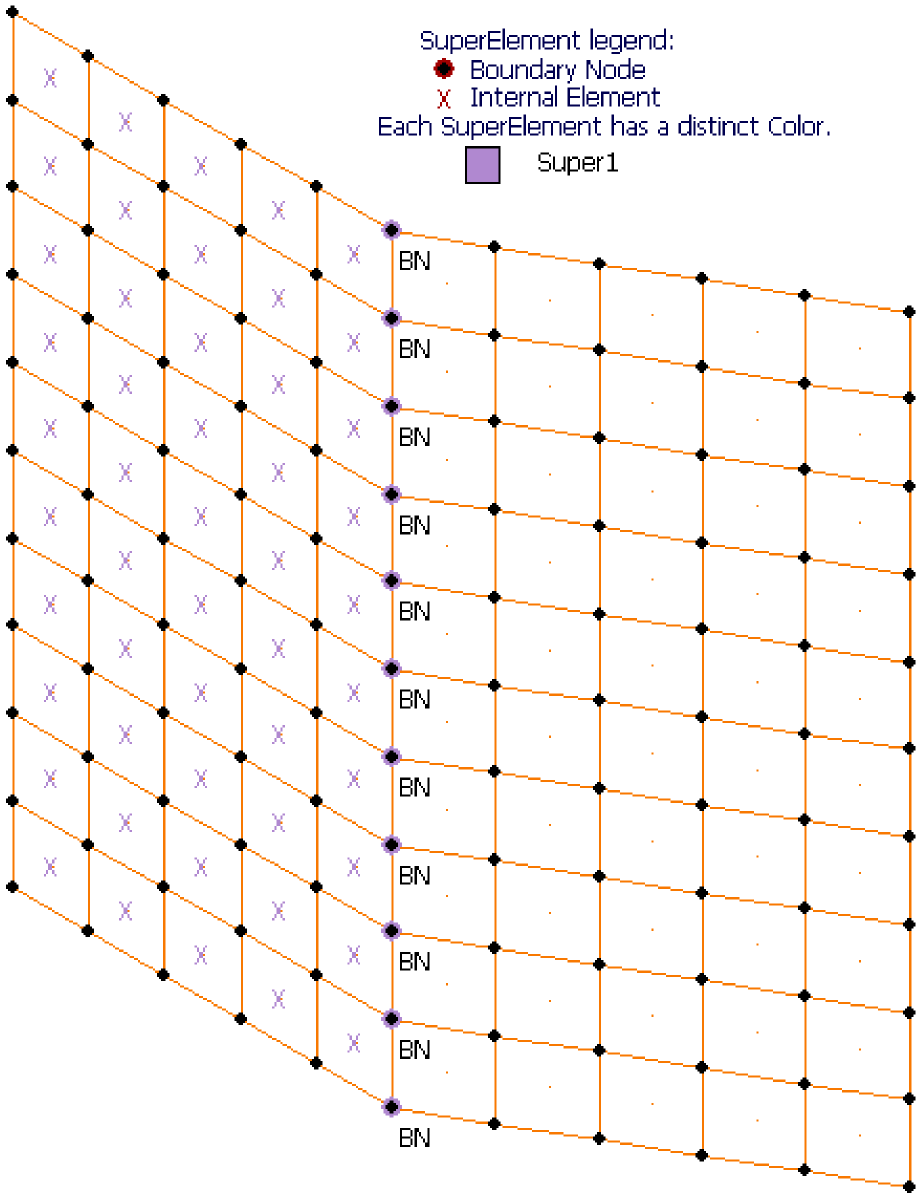
Options on this dialog allow you to draw lines connecting the Master and Slave joints, Label the Master and Slave joints, and place a legend and color key on the screen. An example of the graphical display produced by this dialog is shown below:



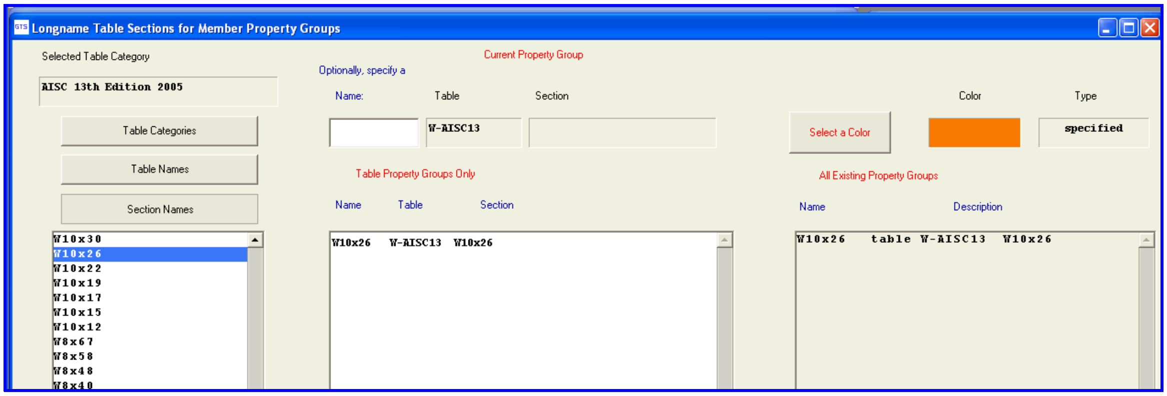
The new Superelement Display dialog is shown below:



Options on this dialog allow you to mark internal elements in the superelement, label the boundary nodes with 'BN', and place a legend and color key on the screen. An example of the graphical display produced by this dialog is shown below:

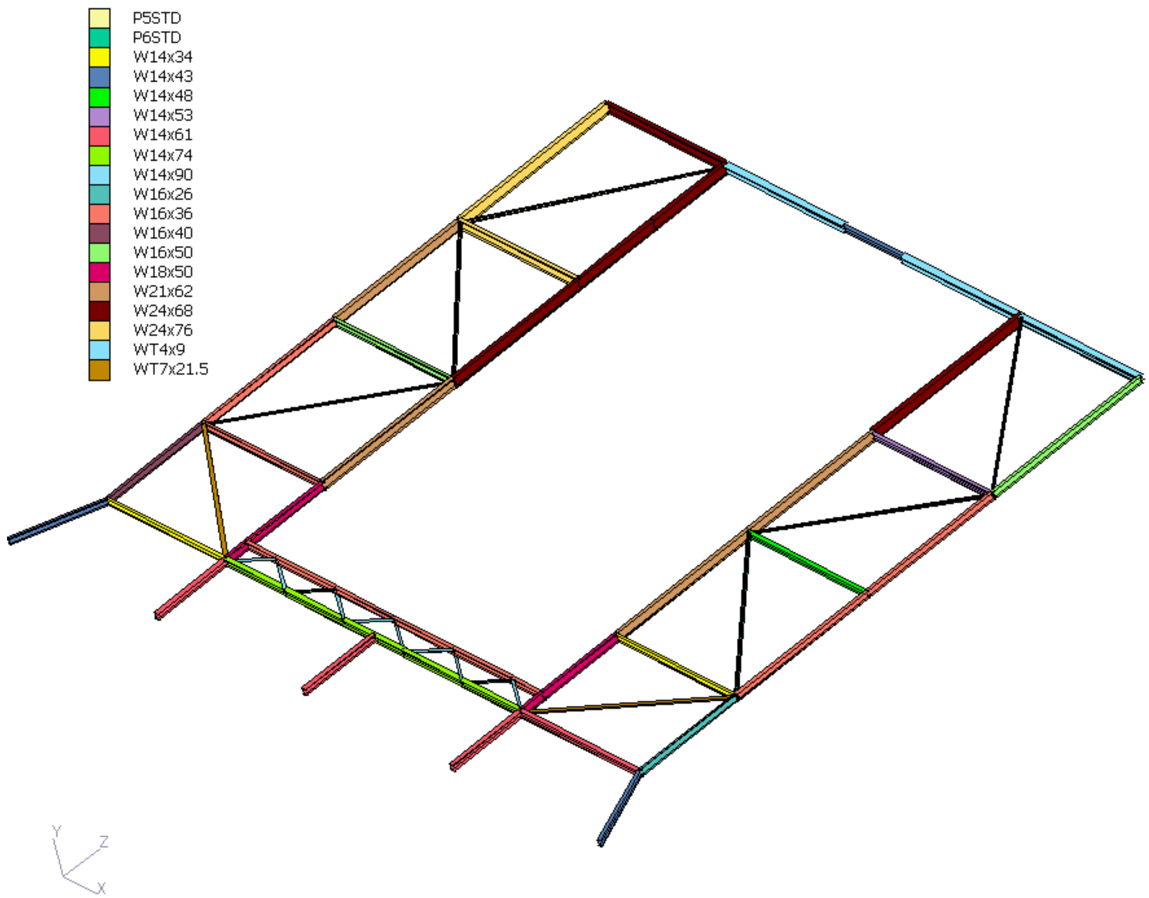


3. The Table Property Group dialog has been modified so that you may now specify a color to be assigned to a table Property Group. This color may then be used when Color by Sections is used in the Display Model dialog which is discussed below. An example of the modified Table Property Group dialog is shown below:

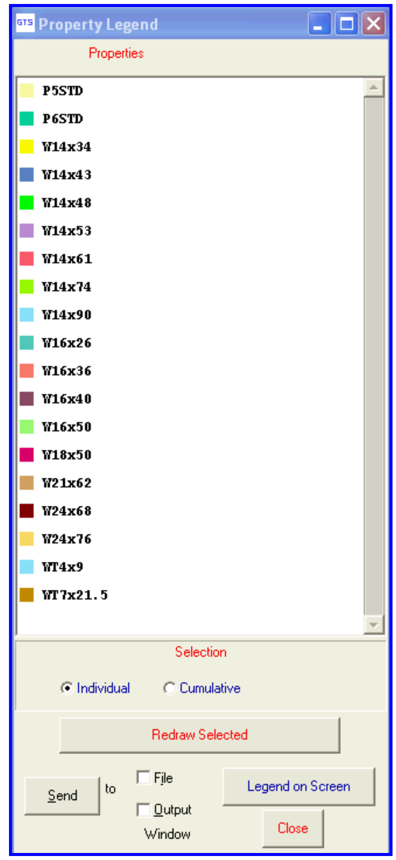


4. The Display Model dialog has been modified and a new option has been added which will allow you to "Color by Section" members which have table property groups. The revised Display Model dialog is shown below:

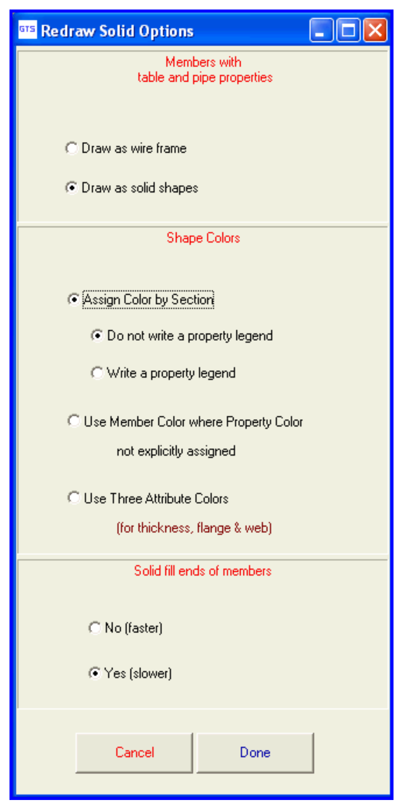
An example of the graphical display resulting from the new Color by Section option is shown below:



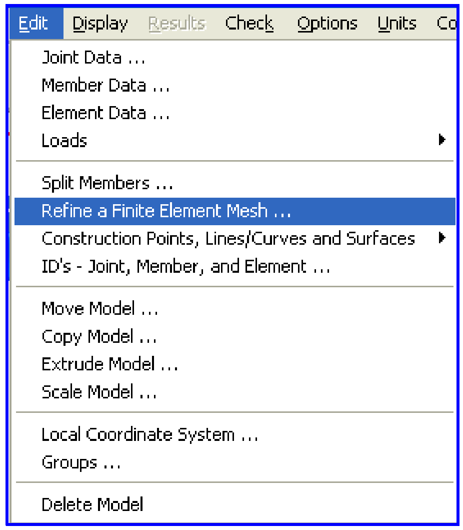
A Property Legend dialog pops up when Color by Section is selected. This will allow you to selectively or cumulatively select property groups to graphically color. An example of the new Property Legend dialog is shown here:



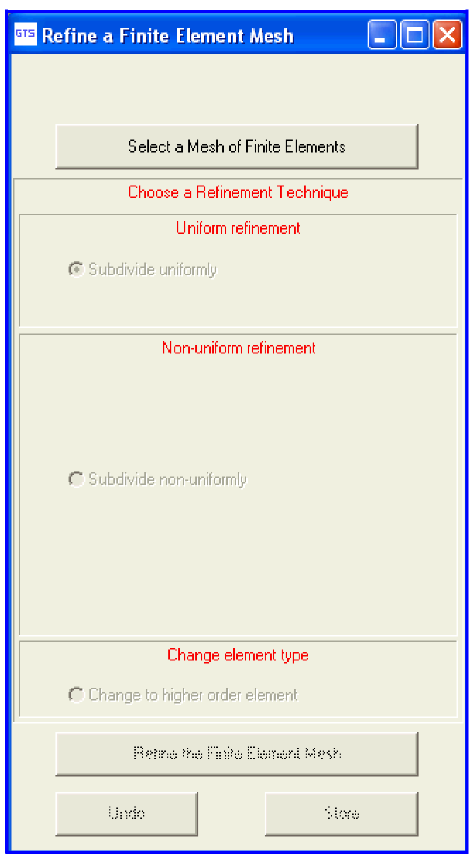
5. You may also select from the Options pulldown for Redraw Solid an option which will result in Color by Section being always performed when you select Redraw Solid. The revised Redraw Solid dialog is shown here:



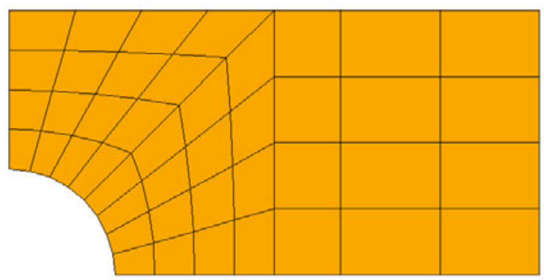
6. A new option has been added to the Edit pulldown which will allow you to "Refine a Finite Element Mesh". The revised Edit pulldown is shown below:

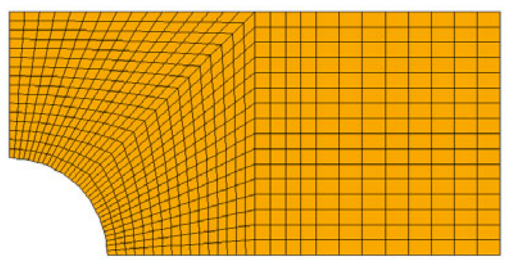


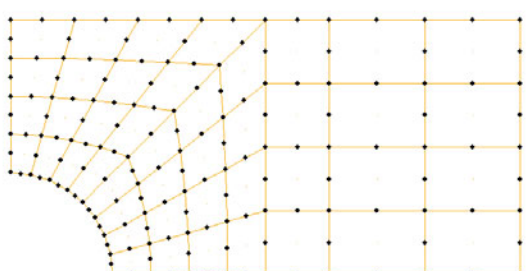
The Refine a Finite Element Mesh dialog is shown below:



Using this dialog, you may refine a finite element mesh with either 2D or 3D elements by either uniformly subdividing each element, non-uniformly subdividing each element, or by changing the elements to a higher order element (a 4 node quadrilateral element is changed to an eight node quadrilateral element). Several examples of the use of this dialog are shown below with the original mesh shown first, followed by uniformly subdividing each element into four elements, and then an example which demonstrates changing each four node element to an eight node element:

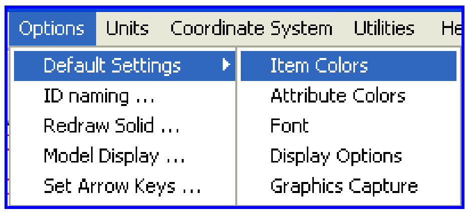




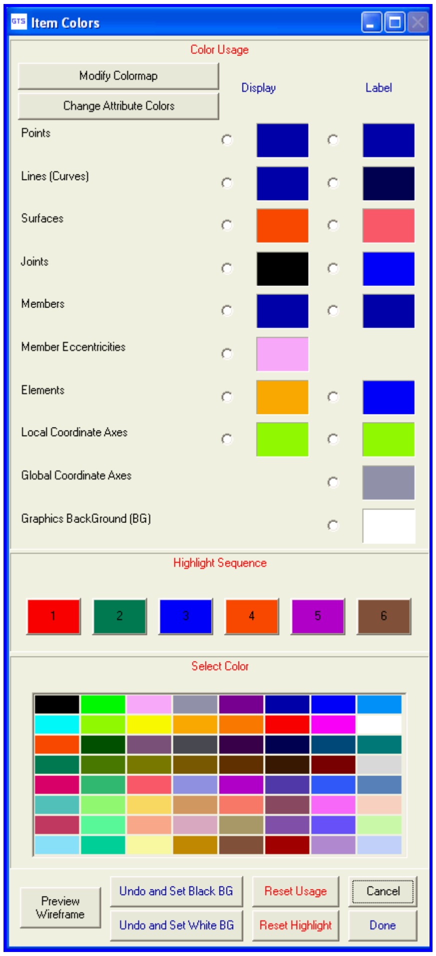


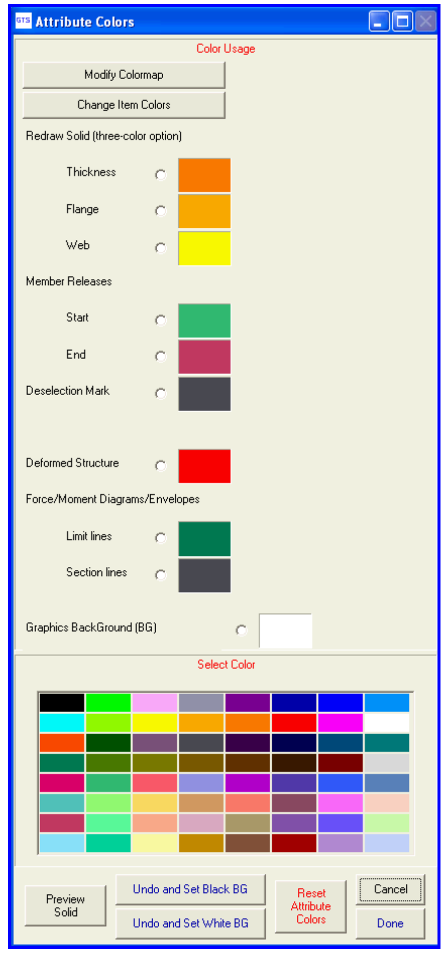
7. The color pallette used by GTMenu has been increased from 16 colors to 64 colors. In addition, the Default Setting option under the Options pulldown has been modified so you may now specify Item and Attribute Colors where the Item Colors dialog is used to specify the colors used for Points, Lines, Joints, Members, Elements, ...,etc. and the Attribute Colors dialog is used to specify the colors used for the Three Color Option of Redraw Solid, Member Releases, Deformed Structure, and Force and Moment Diagrams.

   The revised Options pulldown showing the new Default Setting options is shown below:

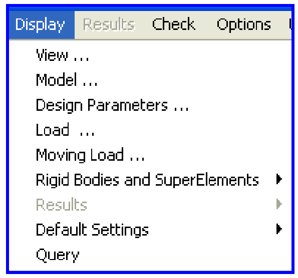


The new Item and Attribute Colors dialogs with the new 64 color pallette are shown below:

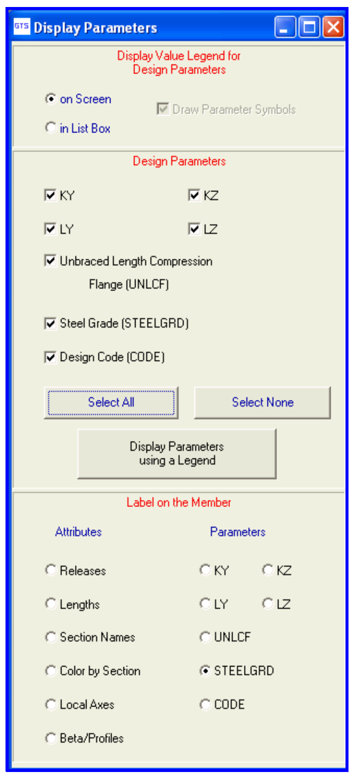




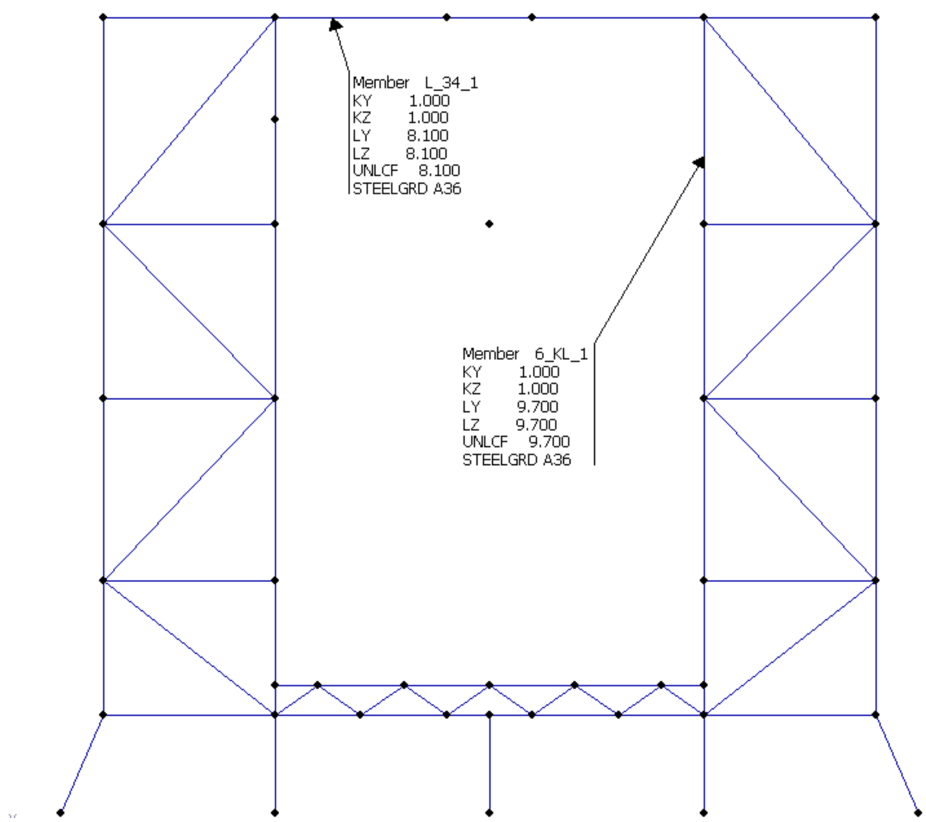
8. The Display pulldown has been modified so you may now display steel design parameters on the graphical display. The modified Display pulldown is shown below:



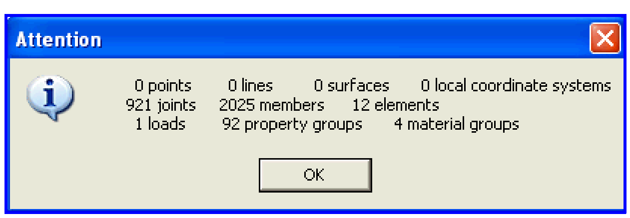
The new Design Parameters dialog is shown below:



This dialog allows you to display and label the following design parameters: KY, KZ, LY, LZ, UNLCY, STEELGRD, and CODE. You may display the parameters on the screen with a legend in a List Box or you may have the legend written for each member. An example showing the parameters with legend for specific members is shown below:



9. The Display pulldown has a new Query option which will pop-up a box indicating information about the number of points, lines, joints, members, elements, and loads in the model. An example is shown below:



The Query option may also be found under the Inquire button.

10. The Default Settings options are also now available under the Display pulldown.

11. Character sizes have been adjusted to accommodate higher resolution and wide screen displays.

12. When File->Save Model is selected and the user elects to overwrite an existing save (.gts) file, the existing save file is renamed to "GTMenuBackup.gts" instead of being deleted as it was previously. The current data base is then saved into the specified file name. When the save process is completed, the backup file (GTMenuBackup.gts) is then deleted. This prevents the loss of the data in the original save file if the save process does not complete properly. Note that the new save model process requires enough disk space for two copies of the save file (the newly written save file and GTMenuBackup.gts) until the save process is completed

13. A member label can now be placed on very short members.

14. Steel design results are now available in GTMenu when parameter TRACE value of 1, 2, or 3 has been specified for the steel design code check or select command.

15. A horizontal scroll bar has been added to the Loads and Section Properties dialogs which will allow you to scroll horizontal to see all of the load description or section property data.

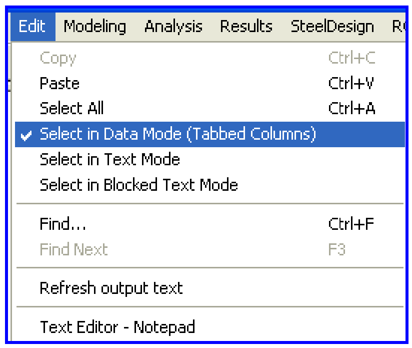
16. The graphical labeling of plastic hinge status has been improved.

17. The Rotate Menu now reflects the current structure rotation.

18. When a member with design parameters is split, the split members now have the design parameters of the original member. The input file created by GTMenu will now contain the design parameters for the members newly created by the split. In addition, the Parameters in the input file are now compressed with a range of members (1 TO 5) being specified on a single line instead of each member being listed (1 2 3 4 5).

2.4 GTSTRUDL Output Window

1. Two new modes for selecting text from the text output window have been added. The first new mode is "Select in Data Mode (Tabbed columns)" and the second is "Select in Blocked Text Mode". "Select in Text Mode" is the original line based selection method and is still the default selection mode. You may switch between modes after highlighting, that is you can select in Text mode and then switch to Data mode before copying to the clipboard. If you hold down the 'Alt' key before starting the selection, the selection mode is automatically changed to Data.

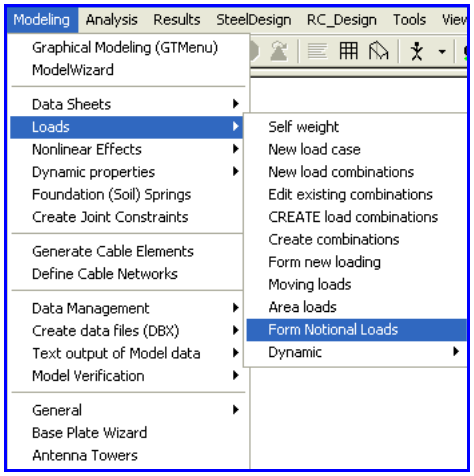


Blocked Text mode allows you to select a block of text from the window, as above, but when you copy to the clipboard, no substitutions of tabs for blanks are done. When you paste into another program, the inserted text is exactly like the block you highlighted. For example, this is useful for selecting echoed commands without the line numbers such as ("{ 345}").

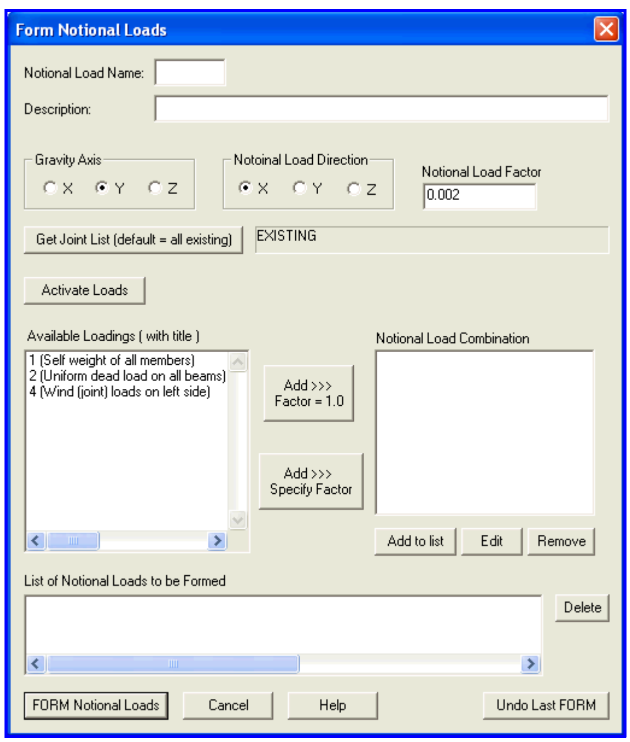
RESULTANT JOINT DISPLACEMENTS FREE JOINTS

JOINT		/----------------DISPLACEMENT	----------------	--------------//	------------------ROTATION-----	
		X DISP.	Y DISP.	Z DISP.	X ROT.	Y ROT.
13	GLOBAL	0.1086346	0.0020808	0.0000000	0.0000000	0.0000000
14	GLOBAL	0.1086954	0.0000000	0.0000000	0.0000000	0.0000000
15	GLOBAL	0.1086346	-0.0020808	0.0000000	0.0000000	0.0000000
16	GLOBAL	0.1086346	0.0020808	0.0000000	0.0000000	0.0000000
17	GLOBAL	0.1086954	0.0000000	0.0000000	0.0000000	0.0000000
18	GLOBAL	0.1086346	-0.0020808	0.0000000	0.0000000	0.0000000
19	GLOBAL	0.1086346	0.0020808	0.0000000	0.0000000	0.0000000
20	GLOBAL	0.1086954	0.0000000	0.0000000	0.0000000	0.0000000
21	GLOBAL	0.1086346	-0.0020808	0.0000000	0.0000000	0.0000000
22	GLOBAL	0.1086346	0.0020808	0.0000000	0.0000000	0.0000000
23	GLOBAL	0.1086954	0.0000000	0.0000000	0.0000000	0.0000000
24	GLOBAL	0.1086346	-0.0020808	0.0000000	0.0000000	0.0000000
25	GLOBAL	0.1986606	0.0027558	0.0000000	0.0000000	0.0000000
26	GLOBAL	0.1984879	0.0000000	0.0000000	0.0000000	0.0000000
27	GLOBAL	0.1986606	-0.0027558	0.0000000	0.0000000	0.0000000
28	GLOBAL	0.1986606	0.0027558	0.0000000	0.0000000	0.0000000
29	GLOBAL	0.1984879	0.0000000	0.0000000	0.0000000	0.0000000
30	GLOBAL	0.1986606	-0.0027558	0.0000000	0.0000000	0.0000000
31	GLOBAL	0.1986606	0.0027558	0.0000000	0.0000000	0.0000000
32	GLOBAL	0.1984879	0.0000000	0.0000000	0.0000000	0.0000000
33	GLOBAL	0.1986606	-0.0027558	0.0000000	0.0000000	0.0000000
34	GLOBAL	0.1986606	0.0027558	0.0000000	0.0000000	0.0000000
35	GLOBAL	0.1984879	0.0000000	0.0000000	0.0000000	0.0000000
36	GLOBAL	0.1986606	-0.0027558	0.0000000	0.0000000	0.0000000

2. A new Form Notional Loads option has been added under the Modeling pulldown Loads option as shown below:

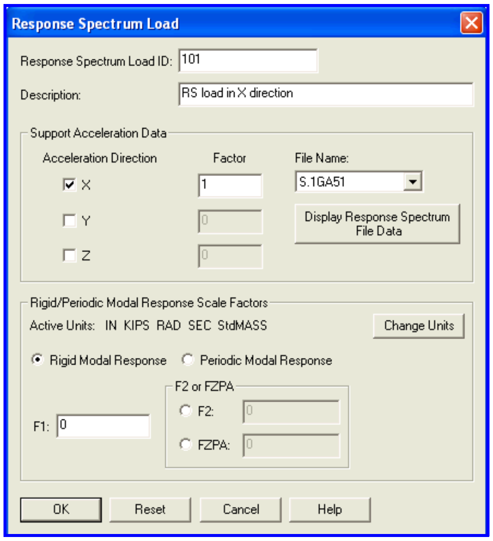


The new Form Notional Loads dialog is shown below:

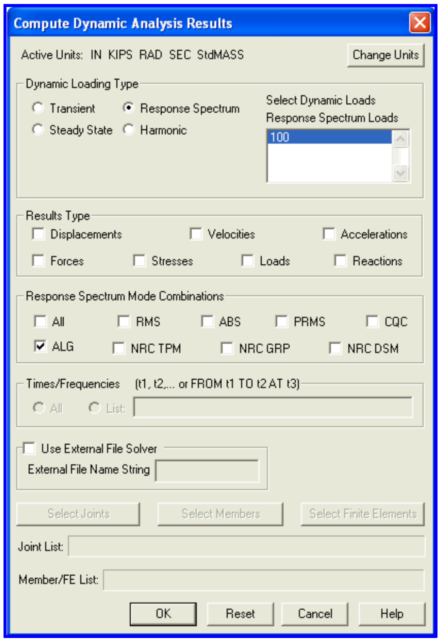


This dialog is used to construct new notional loading cases from previously entered loading data. Notional loads are perturbation lateral joint loads that are used to account for lateral geometric imperfections and structural inelasticity according to the *Direct Analysis Method* of the 2005, 13th Edition AISC Specification. The new loading condition is an independent static load, consisting of joint loads only, applied in a specified lateral direction, to a specified set of joints.

3. The Response Spectrum Load dialog under the Dynamic option of the Loads option under the Modeling pulldown has been modified to allow the user the specify either Rigid or Periodic Response Scale Factors according to the Gupta Method. The revised dialog is shown below:



4. The new Algebraic (ALG) Modal Combination method has been added to the Compute Dynamic and List Dynamic dialogs as shown below:



5. The following changes have been added to the Steel Design dialogs:

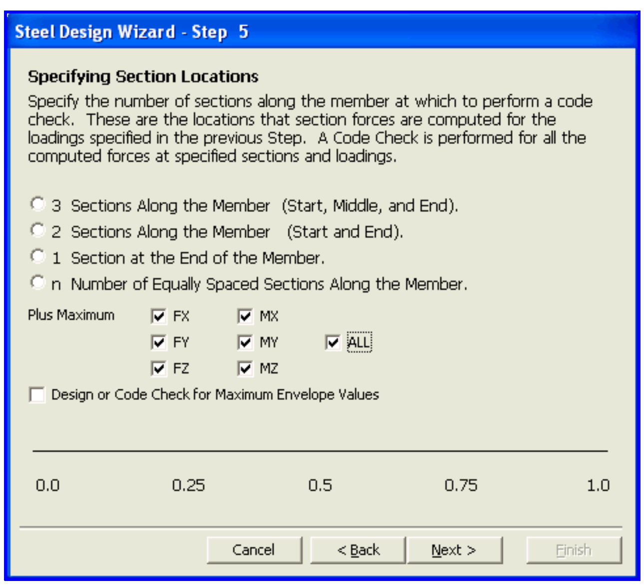
   a. Steel grades for structural tubes and pipes have been added to the Parameters dialog for the ASD9 code

   b. Steel grades for pipes have been added to the Parameters dialog for the APIWSD20 and APILRFD1 codes

   c. Parameters for the AISC13 code have been added to the Parameters dialog

   d. The Steel Design Wizard has been updated for the AISC13 code

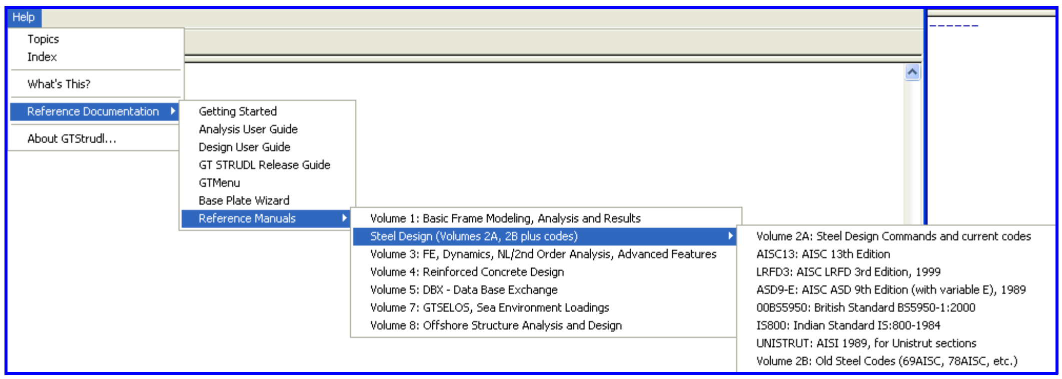
   e. Step 5 of the Steel Design Wizard has been modified to account for the new Section specification option which has been added to the SELECT and Check commands. This option allows the user to specify the maximum forces or moments section locations to be added to the user specified section locations. This new Section option also allows the user to create a single section point which contains the maximum axial, shear in Y and Z directions, and moments in X, Y, and Z directions. The revised Step 5 of the Steel Design Wizard dialog is shown below:



   f. A button has been added to the Steel Design Wizard to display the deflection loads specified by the parameter 'DefLoads'.

6. The Steel code check datasheet is now available for TRACE values of 1, 2, and 3.

7. The Help Menu has been modified with the Reference Manual Documentation more fully explained so users have a better understanding of the contents of the various volumes of the Reference Manuals. The modified Help pulldown is shown below:



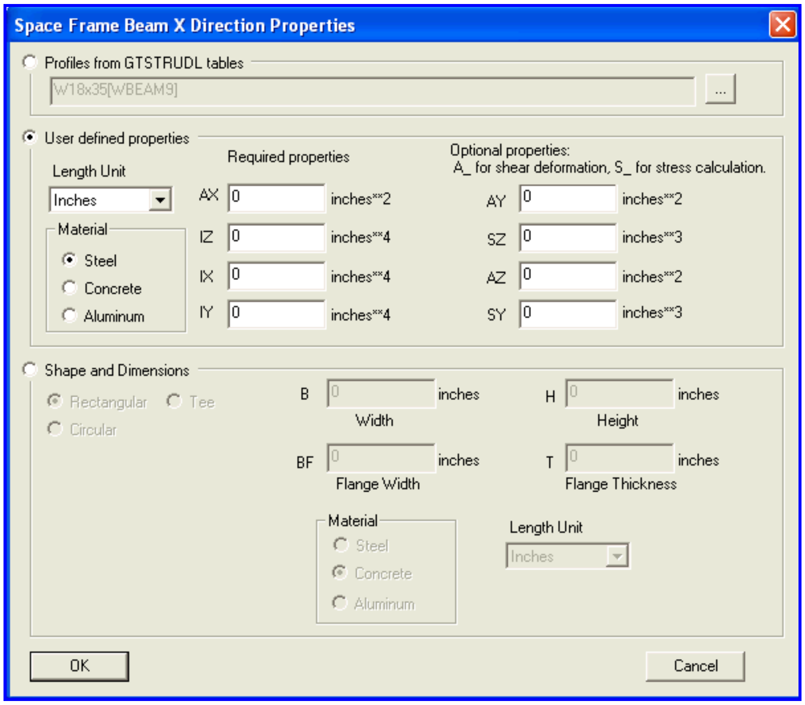
2.5 Model Wizard

1. The Model Wizard now writes a GTSTRUDL input file in the current Working Directory with a name determined by the model type, i.e. "CircularTankWizard.gti". This results in faster processing times for large models and automatically keeps a copy of the created input file in your Working Directory. The model name is fixed, and will be overwritten with no warning each time that model type is used, so rename the created input file if you want to keep it.

2. The Model Wizard is now version dependent so that installation of new versions of GTSTRUDL will not affect previous installations.

3. The Circular Plate Wizard now includes a changeable length unit for the thickness.

4. The Plane Frame and Space Frame Wizards now allow you to specify all prismatic properties such as AX, AY, AZ, IX, IY, IZ ,etc. Previously, the Plane Frame Wizard permitted only AX, IZ, and SZ while the Space Frame Wizard did not allow you to specify any prismatic properties. The new prismatic property options in the Space Frame Wizard are shown below:



2.6 Nonlinear

1. Nonlinear static analysis has been extended to write analysis results to external results save files, in order to be consistent with the storage of linear static analysis results and dynamic analysis results on such files. Now, if a nonlinear static analysis is executed in the same job with linear static analysis and/or dynamic analysis for which results are stored on external files, the post-processing of all analysis results, including the nonlinear static analysis results, will be consistent, allowing the successful post-processing of all analysis results. In previous versions that support the external file storage of linear static analysis and dynamic analysis results, the inconsistency of nonlinear static analysis results stored in virtual memory and file-stored linear static analysis and dynamic analysis results precluded the processing of both types of results in the same job.

2.7 Offshore

1. The computation of chord SCF values for fatigue analysis has been improved. In earlier versions, excessively conservative chord SCF values were computed by requiring that any chord SCF had to be no less than the maximum corresponding brace SCF from all braces connected to the chord. This requirement is not necessary and has been removed. This feature was released in Version 29.2 and is reported here also.

2. The fatigue analysis type options of the PERFORM FATIGUE ANALYSIS command have been expanded to include a DETERMINISTIC method in addition to the PSD and DISCRETE wave spectrum methods. Rather than being stochastic in nature like the PSD and DISCRETE spectrum methods, the DETERMINISTIC method assumes that the wave load data from GTSelos consists of multiple sets of regular waves (not statistical waves), each wave having an occurrence characteristic that is treated directly as the number of stress amplitude cycles for the wave. The stress amplitude derived from the wave load analyses for each wave and the number of stress amplitude cycles for each wave are then used to compute the partial fatigue damage due the wave. The partial damages from all waves are then accumulated to compute the total fatigue damage from which the total fatigue life is estimated.

3. The offshore fatigue analysis feature has been enhanced with the addition of COMPUTE FATIGUE LJF EFFECTS command, which performs an analysis whereby the stiffness characteristics of fatigue brace members at selected joints are modified to reflect the flexibility of typical brace-to-chord welded connections.

2.8 Steel Design

1. When a value of -99.0 is displayed for provision 'H1-1 COM' of the ASD9 code or provision 'EQ1.6-1A' of the 78AISC or 69AISC codes, the actual/allowable ratio indicates an incomplete combined stress computation. This generally means that the value of f_a/F'_e is greater than or equal to 1.0 which causes the denominator of Equation H1-1 of the 1989 AISC ASD Ninth Edition or Equation 1.6-1a of the 1978 and 1969 AISC codes to be less than or equal to zero [$(1-- f_a/F'_e) \leq 0.0$]. When f_a/F'_e is greater than 1.0, the stress due to axial force is larger than the Euler buckling stress. Instead of printing -99.0 for the default code check results in Version 29.2, the combined axial and bending stress values based on Equation H1-3 of the 1989 AISC ASD Ninth Edition or Equation 1.6-2 of the 1978 and 1969 editions of AISC are printed. The error messages that the denominator of Equations H1-1 or 1.6-1a is a less than or equal to zero are still printed, but in the code check default output, the value for the combined axial and bending stress (Equation H1-3 or 1.6-2) also will be displayed. If you request summarized results, you will see a value of -99.0 for provision 'H1-1 COM' or 'EQ1.6-1A', which indicates the problem described above when computing the Euler buckling stress and the interaction equation.

   An example of the new default output for the ASD9 code is shown below:

```
 {    73} >
 {    74} >
 {    75} > OUTPUT  WARNING  MESSAGE  LIMIT  1
 {    76} >
 {    77} > CHECK  ALL  MEMBERS
1

     **********************
     * DESIGN TRACE OUTPUT *
     **********************

     JOBID - MS615      TITLE - Check message limits for ASD9 code

     ** MEMBERS WHICH FAIL ARE MARKED BY TWO ASTERISKS (**) **


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

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

 **** INFO_STMESL -- The warning message limit of   1 has been reached.
                    Any further warning messages of the above type will
                    be suppressed.


  ** 1        WCOLUMN9    4              0.000 H1-3 COM    26.915     -100.000      0.000    -37.500 FEET KIP
     ASD9      W14x53                            B7 COMP      2.344        0.609    562.500      0.000 FAILED

 **** ERROR_MSA9S7 -- DENOMINATOR (1 - fa/Fez) FOR MAJOR AXIS BENDING STRESS
           RATIO IN COMBINED STRESSES EQUATION (EQ. H1-1) IS LESS THAN OR
           EQUAL TO ZERO FOR MEMBER 2        LOADING 1        SECTION     0.00
           ACTUAL STRESS fa IS GREATER THAN EULER STRESS Fez. Z AXIS ELASTIC
           BUCKLING FAILURE.
```

```
**** INFO_STMESL -- The warning message limit of   1 has been reached.
                    Any further warning messages of the above type will
                    be suppressed.

 ** 2        WCOLUMN9    4              0.000 H1-3 COM    26.915     -100.000      0.000    -37.500 FEET KIP
    ASD9     W14x53                           B7 COMP      2.344       -0.609    562.500      0.000 FAILED

 ** 3        WBEAM9      4              7.500 H1-3 COM     4.477        0.000      0.000  5.737E-06 FEET KIP
    ASD9     W12x50                           B7 COMP      0.459   -4.672E-07   -140.016      0.000 FAILED

    ***********************
    * END OF TRACE OUTPUT *
    ***********************
```

2. A new AISC 13th Edition design code has been implemented as a prerelease feature. This new code, AISC13, may be used to select or check any of the following shapes:

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

I shapes	Channels
Single Angles	Round HSS (Pipes)
Rectangular and Square HSS (Structural Tubes)	

Design for bi-axial bending and axial forces:

Tees	Double Angles
Round Bars	Square Bars
Rectangular Bars	

The documentation for the AISC13 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and AISC13 in the GTSTRUDL Output Window.

3. The ANSI/AISC N690-06, specification for safety-related steel structures for nuclear facilities has been added as a prerelease feature. This code is based on the AISC steel construction manual, Thirteenth Edition specification with a few modifications. Both LRFD (load and resistance factor design) and the ASD (allowable stress design) method of the AISC Thirteenth Edition are applicable. Applicable cross-sections are I shapes, Channel, Single Angles, Tees, Double Angles, Round HSS (Pipes), Rectangular and Square HSS (Structural Tubes), Solid Round Bars, Solid Rectangular and Square Bars. Additional documentation is available in the Section 1.3 of the AISC13 code. The documentation for the AISC13 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and AISC13 in the GTSTRUDL Output Window.

4. New error checking has been added into the steel design parameter command. User specified parameter values are now checked for correctness. Alphanumeric parameter values are checked against accepted values and if the specified value is incorrect, an error message is given and the scan mode is set. Real parameter values are checked for negative and zero values.

5. A new Section specification option has been added to the SELECT and CHECK commands. The PLUS option allows the user to specify that maximum forces or moments section locations be automatically added to the user specified section locations. The ONLY option allows the user to specify that a single section point which contains the maximum axial, shear in Y and Z directions, and moments in X, Y, and Z directions be used for selection or checking. Examples of the new Section specification options are shown below:

   a. SELECT MEMBERS 1 TO 5 PLUS MAXIMUM ALL

      The PLUS MAXIMUM ALL option will add sections to the user specified sections where the maximum FX, FY, FZ, MX, MY, and MZ forces and moments are located.

   b. SELECT MEMBERS 1 TO 5 ONLY MAXIMUM ALL

      The ONLY MAXIMUM ALL option will use only sections which contain the maximum forces and moments in the member design.

   The new Section specification options on the SELECT and CHECK commands are documented in Sections 2.6 and 2.8 respectively in Volume 2A of the Reference Manuals.

6. The STEELGRD Parameter now accepts ASTM steel grades shown in Table 1, Page 1-7 of the *Manual of Steel Construction, Allowable Stress Design* Ninth Edition for pipe and structural tube cross-sections. Note that the ASTM steel grades shown in the Table 3, Page 1-92 of the *Manual of Steel Construction, Allowable Stress Design* Ninth Edition for pipes and structural tubes are now available in GTSTRUDL.

7. Efficiency improvements have been implemented into the steel design SELECT and CHECK commands. The time required to perform a code check or design for models with large numbers of members and loadings has been reduced dramatically. An example of the reduction in time is shown below:

   Model Description: 11,681 joints, 19,086 members, 31 design loading conditions

   Time to Select 19,086 members with 3 sections per member:

   Version 29.2 and earlier - 247 minutes

   Version 30 - 30 minutes

8. The EC3 code check for pipe cross-sections will now check Equations 5.5.3, 5.5.4(1), or 5.5.4(3). Previously, the EC3 code check for pipes did not check these equations.

9. Weld design will now allow the user to suppress the following CN.58 warning message:

```
**** STRUDL WARNING CN.58 -  STEELGRADE HAS NOT BEEN SPECIFIED FOR MEMBER 1 AT JOINT 1
```

   To suppress the above warning message, specify a value of 'NO' for the parameter 'WarnMess' as shown below:

```
PARAMETER
WarnMess    NO    ALL
CHECK WELD JOINT 1
```

2.9 Steel Tables

1. The following new Tables from the AISC Thirteenth Edition have been added to GTSTRUDL:

W-AISC13	W shapes from Table 1-1 of the AISC 13th Edition
M/S/HP13	M, S, and HP shapes from Tables 1-2, 1-3, and 1-4 of the AISC 13th Edition
C-AISC13	Channel C and MC shapes from Tables 1-5 and 1-6 of the AISC 13th Edition

L-ALL-13	Single angles from Table 1-7 of the AISC 13th Edition
L-EQ-13	Equal leg single angles from Table 1-7 of the AISC 13th Edition
L-UN-13	Unequal leg single angles from Table 1-7 of the AISC 13th Edition
WTAISC13	Tee WT, MT, and ST shapes from Tables 1-8, 1-9, and 1-10 of the AISC 13th Edition
RecHSS13	Rectangular and square HSS from Tables 1-11 and 1-12 of the AISC 13th Edition
RdHSS13	Round HSS from Table 1-13 of the AISC 13th Edition
2L-ALL13	Double angles from Table 1-15 of the AISC 13th Edition
2L-EQ-13	Equal legs double angles from Table 1-15 of the AISC 13th Edition
2L-LL-13	Long legs back-to-back double angles from Table 1-15 of the AISC 13th Edition
2L-SL-13	Short legs back-to-back double angles from Table 1-15 of the AISC 13th Edition
WBEAM-13	W shapes commonly used as beams from Table 3-6 of the AISC 13th Edition
WCOL-13	W shapes commonly used as columns from Table 4-1 of the AISC 13th Edition

2.10 Utility Programs

1. A new option (ProfileMap.txt) has been added for mapping profile names in a CIS2 file into GTSTRUDL, where users may create a mapping file for profiles not in the AISC standard names file.

   See your installation directory\CIS2\Readme.txt (usually C:\Program Files\GTStrudl\CIS2\Readme.txt) for more information.

2.11 Advanced Multi-processor 32 bit and 64 bit Solvers

New standalone solvers have been added to Version 30 which allow you to take advantage of the multi-processors which are often referred to as multi-cores such as Intel Core 2 Duo and Intel Core 2 Quad CPU's. These solvers are separately licensed products and require that an additional password be added to your password file. These solvers will take advantage of the multiple processors available in today's computers. In addition, the 64 bit solver will take advantage of the large address space (virtual page file) available under the 64 bit version of the Windows Vista operating system.

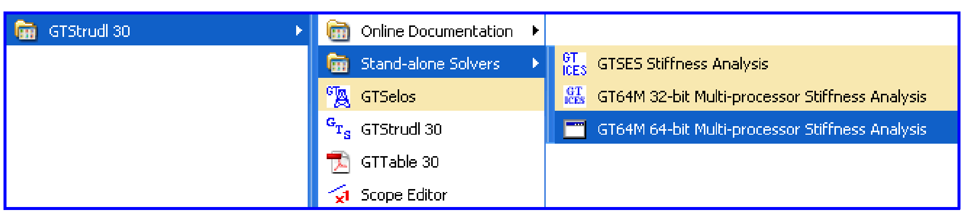
To take advantage of the multi-processors, you must use the split solver following the steps described below:

a. First, assemble the equations

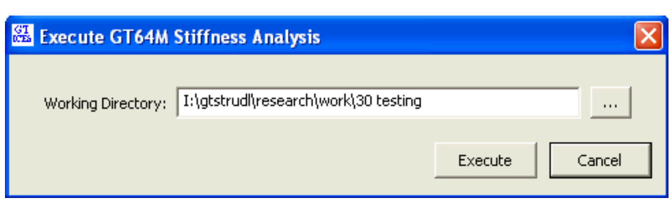
GT64M ASSEMBLE FOR STATICS

You can perform a SAVE after this step and then close GTSTRUDL or leave the GTSTRUDL Output Window open while continuing to the next step.

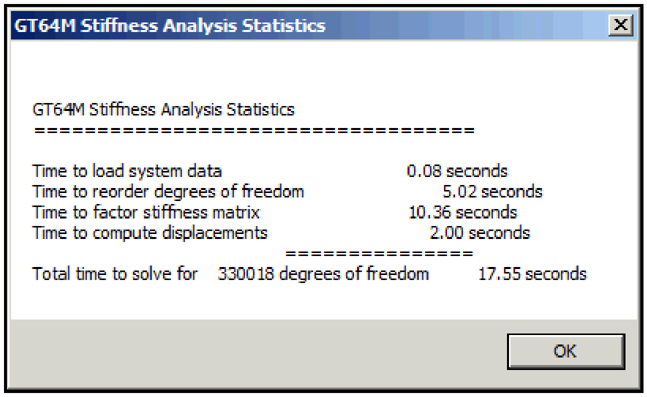
b. Now, execute the standalone GT64M solver by clicking on the Windows START button, then clicking on All Programs, and then selecting GTStrudl30 followed by selecting Stand-alone Solvers and then selecting either GT64M 32 or 64 bit Multi-processor Stiffness Analysis as shown below:



The following pop-up will appear after selecting the solver



When the analysis completes, a pop-up such as the one below will appear with analysis statistics:



c. Now, compute gross results. If you have performed a SAVE and closed the GTSTRUDL Output Window, you should first RESTORE the model and then give the command:

GT64M COMPUTE GROSS RESULTS

The new split solver commands described above may be found in Section 2.1.13.3.2 of Volume 1 of the Reference Manuals.

An example of the performance gains realized by taking advantage of the new solver and multi-processors is shown below:

12,000 joints with 6 degrees-of-freedom per joint = 69,300 total DOFs

Band width = 387 joints

1 loading

Times to solve:

Standard Stiffness Analysis Solver	690 Seconds
GTSES	418 Seconds
GT64M with 1 processor	29 Seconds
GT64M with 4 processors	16 Seconds

The STIFFNESS ANALYSIS command has also been modified to use the GT64M solver. The modified STIFFNESS ANALYSIS command is documented in Section 2.1.13.2 of Volume 1 of the Reference Manuals and is shown below:

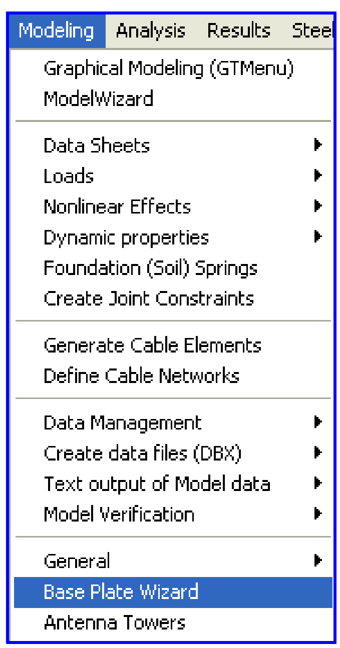
$$\underline{\text{STIFFN}}\text{ESS}\ \ (\underline{\text{ANAL}}\text{YSIS})\ \ \left(\begin{Bmatrix}\underline{\text{NJP}}\ \ \text{i} \\ \underline{\text{WITHO}}\text{UT}\ \ \underline{\text{RED}}\text{UCE}\ \ (\underline{\text{BAND}})\end{Bmatrix}\right)\ \text{-}$$

$$\left(\begin{Bmatrix}\underline{\text{GTSES}} \\ \underline{\text{GT64M}}\ \ (\underline{\text{OOC}})\end{Bmatrix}\right)\ \ (\text{reform})$$

When the GT64M option is used on the STIFFNESS ANALYSIS command, only one processor is used but the solver takes full advantage of the 64 bit virtual address space if executed on a 64 bit operating system. The OOC (out-of-core) option which is also available using the split solver is particularly useful when running on a 32 bit operating system where the virtual address space is limited.

2.12 Base Plate Wizard

A new Base Plate Wizard has been added under the Modeling pulldown as shown below:



The Base Plate Wizard is a separately licensed product and requires that an additional password be added to your password file.

The Base Plate Wizard (BPW) is a graphically orientated module of GTSTRUDL used to create a GTSTRUDL model of a base plate with one or more attachments, anchored to a bearing surface. Dialogs are provided to allow the user to set plate dimensions; notches, holes and skewed edges; finite element mesh size; attachment location and rotation; anchor location and properties; bearing surface properties; stiffeners; modeling constraints not set by the bearing surface or anchors; and loadings. A graphical display allows a visual confirmation of the geometry. Upon request, a GTSTRUDL input file is created. While in the BPW, an analysis can be performed and results viewed in GTMenu or in summary tables in the BPW.

A base plate is modeled as a finite element mesh in the XY plane, with the positive Z axis extending out towards the viewer. The finite element mesh is composed of SBHQ6 quadrilateral and SBHT6 triangular elements.

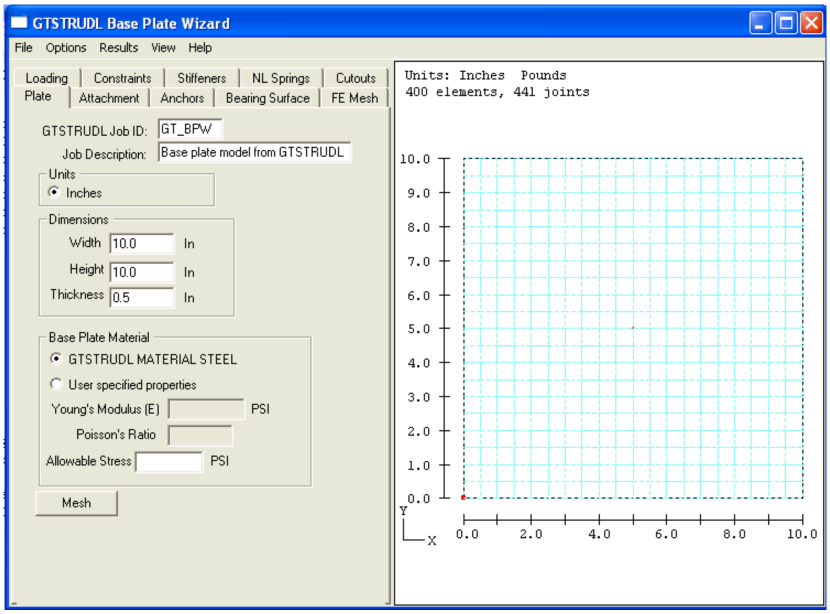
Attachments and stiffeners are modeled as RIGID SOLIDS in the plane of the base plate or as finite element extensions, based on the user's decision. Each attachment or stiffener may use either option and a base plate model may contain attachments and stiffeners modeled with differing options. Attachments may be selected from the AISC hot rolled shapes, or may be user defined.

Anchors are modeled as point constraints in the base plate, with a node at the anchor center. Anchors have axial (Z direction) properties and shear (X-Y plane) properties. Anchors may have differing properties, for example you may have both ½" and 3/4" bolts in the same base plate. Anchors may have allowable tension and shear values specified and variable interaction equation exponents for Pass/Fail checking.

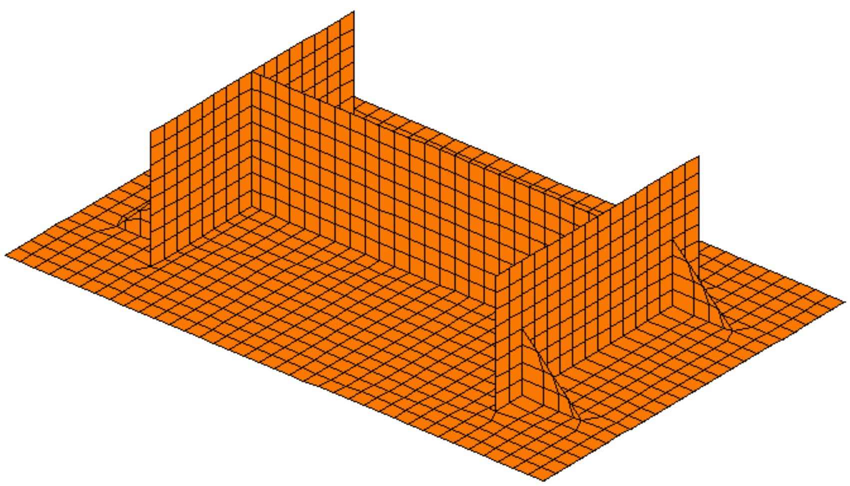
The bearing surface is modeled as a compression-only "semi-infinite elastic foundation". You may specify a f'_c for concrete or an E value and have the spring stiffness (K) calculated, or directly input a K value for the bearing surface. You may specify a gap between the bearing surface and the plate.

Additional modeling constraints may be imposed using either a single node or a line. These constraints will force the base plate mesh nodes to be consistent with the specification and may also have boundary conditions assigned. Constraints can be used to model base plate features not included in the Base Plate Wizard, such as shear lugs or welds to embedded plates.

The Base Plate Wizard is a Windows-style interface with a menu bar, dialogs and a plate display. The dialogs are organized as tabbed pages. The following pop-up dialog appears upon selecting the Base Plate Wizard:



An example of a model created using the Base Plate Wizard is shown below:



The Base Plate Wizard documentation may be found under the Help Menu.

This page intentionally left blank.

CHAPTER 3

ERROR CORRECTIONS

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

3.1 DBX

1. The WRITE AVERAGE STRESS will no longer fail to produce a file when it is not preceded by the CALCULATE AVERAGE STRESS command. (No GPRF issued)

2. The WRITE AVERAGE ELEMENT RESULTS with the UNREGISTERED option will now work as documented. (No GPRF issued)

3.2 Dynamics

1. In previous versions, the FORM UBC97 LOAD command distributed the total design base shear force V over the joints of each floor of the structure without first reducing V by the top floor force F_t (as defined by Equation 30-14, Volume 2 of the 1997 Uniform Building Code). The total computed lateral forces thus exceeded the specified total design base shear by an amount equal to F_t, after F_t was additionally distributed to the top floor joints of the structure. This has been corrected to where the total design base shear force V is first reduced by F_t prior to being distributed to the joints of the structure. (GPRF 2009.02)

2. Response Spectrum Analysis will no longer abort if rigid bodies and or joint ties are present in the model. (GPRF 99.18 - this GPRF was closed in a previous release but had still been listed as open)

3.3 General

1. Finite element types SBHT and SBHQCSH are now accepted by the CALCULATE SOIL SPRINGS command as documented. (GPRF 2009.01)

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

3.4 GTMenu

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

1. Edit Joint Coordinates in GTMenu now works in Line, Plane and List modes.

2. Defining a mixed group which includes members (joints and members, joints, elements and members, or element and members) might lead to an abort in previous versions of GTMenu. This has been corrected.

3. After a right-click to ignore the display of an axis legend in Display Model, the cursor would be left in a mode to fix the corner of a box. The cursor is now reset.

4. Element loads can no longer be assigned to element types UTLQ1 and PSRR in GTMenu. In previous versions, element loads could be assigned to these element types but the elements did not support the loads and the loads were ignored in analysis.

5. A Stress Contour that has all values positive or negative now plots correctly. This problem was corrected in Version 29.2 and is reported here also. (No GPRF issued)

6. Material constants specified with a value of zero are now retained when generating an input file or when exiting GTMenu.

7. GTMenu will no longer abort during a Check Model, Generate Input File, or when ending GTMenu and saving the changes for models which contain over 131,000 members.

8 Joint Releases are now labeled correctly in a Window View.

9. An abort will no longer occur during Redraw Solid for models which contain a mixture of 3D solid elements and 2D elements. This problem was corrected in Version 29.2 and is reported here also. (No GPRF issued)

10. You are no longer allowed to place loads on finite elements of type PSRR in GTMenu as this element does not support element loads.

11. An abort has been corrected when performing multiple copies.

12. An abort has been corrected when performing Check All in the Check Model dialog.

13. Inactive joints, members, or elements are no longer labeled.

3.5 GT STRUDL Output Window

(*GPRF's are not issued for the GTSTRUDL Output Window unless specifically noted below*)

1. Previously, if a non-zero Beta angle was specified for a member created in the datasheet, GTSTRUDL would abort upon exit from the datasheet.

2. An abort no longer occurs when attempting to execute the Results datasheet for member end forces and section forces when rigid bodies are present in the model.

3. The datasheets for displacements, member end forces, and section forces will now work when such results have been computed using the GTSES solver. This problem was corrected in Version 29.2 and is reported here also.

4. The paths to the DXF and CIS/2 Import and Export functions available under the File pulldown have been corrected. This problem was corrected in Version 29.2 and is reported here also.

3.6 Model Wizard

1. The Bridge Truss Wizard will no longer generate an unstable model if all members are requested to be released at joints. (No GPRF issued)

3.7 Nonlinear Analysis

1. Pushover analysis with displacement control activated no longer aborts when inactive and/or deleted joints are present in the model. This problem was corrected in Version 29.2 and is reported here also. (GPRF 2007.04)

2. An abort will no longer occur if the strain exceeds the limits of the piecewise linear stress-strain curve in a custom plastic hinge. This problem was corrected in Version 29.2 and is reported here also. (GPRF 2007.05)

3. The abort of the LIST PLASTIC HINGE STRESSES for members having a plastic hinge defined only at the end has been corrected. (GPRF 2008.01)

4. Nonlinear static analysis no longer has the possibility of exhibiting premature equilibrium convergence when the following conditions arise:

   a. The CONVERGENCE TOLERANCE EQUILIBRIUM command is used to specify the convergence tolerance.

   b. Nonlinear spring elements are the only source of nonlinearity in the model.

   c. A linear elastic support condition is specified in the same direction as the nonlinear spring force/moment component at one or both incident joints of one or more nonlinear spring elements.

   (GPRF 2008.04)

5. The error regarding incorrect nonlinear dynamic analysis reactions being reported at support joints to which are connected nonlinear spring elements having few than 6 defined degrees of freedom has been corrected. (GPRF 2008.07)

6. The END OF NONLINEAR SPRING command now recognizes the words OF, NONLINEAR, and SPRING. (No GPRF issued)

7. Nonlinear analysis will no longer abort in the second of multiple load analyses when rigid bodies are incorrectly specified as having nonlinear geometric effects. This problem was corrected in Version 29.2 and is reported here also.

3.8 Offshore

1. Certain combinations of structural model geometry and and wave heights, wave periods, and wave load generation time increments may cause GTSELOS to produce member wave load data that can cause a GTSTRUDL arithmetic fault and program abort when such wave load data are processed by the GTSTRUDL READ WAVE LOADS FOR FATIGUE... NEW and ASSEMBLE FOR FATIGUE commands. This problem was corrected in Version 29.2 and is reported here also. (No GPRF issued)

2. The ASSEMBLE FOR FATIGUE command no longer aborts when wave loads have been read from multiple wave load files created by GTSELOS and each of the wave load files contains member loads that reflect multiple duplicate wave heights and wave periods. This problem was corrected in Version 29.2 and is reported here also. (No GPRF issued)

3. The error that may cause the incorrect computation of Kuang and Smedley chord-side stress concentration factors and thus possibly incorrect fatigue damages by the COMPUTE FATIGUE LIFE and PERFORM FATIGUE ANALYSIS commands has been corrected. This problem was corrected in Version 29.2 and is reported here also. (No GPRF issued)

4. In the pile structure interaction feature, the pile numbering is no longer required to correspond to the boundary node ordering in the superelement definition. More specifically, pile number 1 is no longer required to be attached to the first boundary node listed in the superelement definition, pile number 2 is no longer required to be attached to the second boundary node listed in the superelement definition,...etc. (GPRF 2008.06)

5. The problem with fatigue brace members connected to slave joints causing the computation of incorrect fatigue stresses and fatigue damage and life has been corrected. (No GPRF issued)

6. The incorrect results produced by the PLOT FATIGUE TRANSFER FUNCTION command when the active length and force units are other than pounds and inches has been corrected. (No GPRF issued)

7. The offshore AUTOMATIC CLASSIFICATION command now functions properly without issuing an error message when a Dynamic Analysis has been performed after a Stiffness Analysis. Previously, when a Dynamic Analysis was performed after a

Stiffness Analysis, the Automatic Classification command would give an error message and not perform the classification. (No GPRF issued)

8. The Punching Shear Check now uses the correct member diameter and thickness for members which have variable cross-section properties. Previously when variable cross-section properties had been specified, the cross-section diameter and thickness at the start of the member were incorrectly used for the end of the member for the punching shear check. (No GPRF issued)

3.9 Reinforced Concrete

1. The DESIGN SLAB command will no longer abort when the ending joint of a member lies on the slab design cut line. (No GPRF issued - the Design Slab command is a prerelease feature)

3.10 Static Analysis

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

2. The STIFFNESS ANALYSIS GTSES command no longer aborts when inactive members or finite elements with undefined properties exist in the model. (GPRF 2007.07)

3. The analysis abort that occurs when a rigid plane, plate, or pin joint constraint has all slave joints coincident with the master joint has been corrected. (GPRF 2008.03)

3.11 Steel Design

1. The deflection parameters DefLimLo, DefLimYL, and DefLimZL are now printed correctly. This problem was corrected in Version 29.2 and is reported here also. (No GPRF issued)

2. The Check and Select commands for a deflection check or design will no longer abort when the new GTSES solver has been used. This problem was corrected in Version 29.2 and is reported here also. (No GPRF issued)

3. The effective section modulus, SZ_e, for non-compact structural tubes is now computed correctly in the ASD9 code. (GPRF 2008.02)

4. The LIST CODE CHECK RESULTS command and the Code Check Results Datasheet will now list the deflection check results. Previously, when there was no axial force, the deflection check results were not listed by the List Code Check Results command and the Code Check Results Datasheet. (No GPRF issued)

3.12 Superelements

1. The DEVELOP STATIC PROPERTIES command no longer aborts for superelements that consist of other superelements that have more than 24 boundary nodes. (GPRF 2008.05)

3.13 Utility Programs CIS/2

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

1. Pipe profiles are now imported correctly, previously they were not recognized.

2. A correction to the importing and exporting of channels has been made. Previously, channels would translate as "reversed" (flanges pointing in the wrong direction).

3. GTSTRUDL models with deleted members are now exported correctly.

4. Physical members with many analytical members are now exported properly, previously an abort occurred.

5. When the proper userdat file is not connected during export, a WARNING about the missing table and profile names is generated and sent to the command screen and into the .log file.

3.14 Utility Program (Scope Editor)

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

1. Rotated labels are now placed properly in the Scope Editor.

CHAPTER 4

KNOWN DEFICIENCIES

This chapter describes known problems or deficiencies in Version 30. These deficiencies have been evaluated and based on our experience, they are seldom encountered or there are workarounds. The following sections describe the known problems or deficiencies by functional area.

4.1 Finite Elements

1. The ELEMENT LOAD command documentation indicates that header information such as type and load specs are allowed. If information is given in the header and an attempt is made to override the header information, a message is output indicating an invalid command or incorrect information is stored. (GPRF 90.06)

2. Incorrect results (displacements, stresses, reactions, frequencies, ... etc.) will result if a RIGIDITY MATRIX is used to specify the material properties for the IPSL, IPSQ, and TRANS3D elements. (GPRF 93.09)

3. The CALCULATE RESULTANT command may either abort or print out an erroneous error message for cuts that appear to be parallel to the Planar Y axis. (GPRF 94.21)

4. If a superelement is given the same name as a member or finite element, an abort will occur in the DEVELOP STATIC PROPERTIES command. (GPRF 95.08)

5. The curved elements, TYPE ‘SCURV’ and ‘PCURV’ will produce incorrect results for tangential member loads (FORCE X). An example of the loading command which will produce this problem is shown below:

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

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

   (GPRF 99.13)

4.2 General Input/Output

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

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

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

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

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

4. Object groups, created by the DEFINE OBJECT command, may not be used in a GROUP LIST as part of a list. If the OBJECT group is the last group in the list, processing will be correct. However, if individual components follow the OBJECT group, they will fail. Also, you can not copy members or joints from the OBJECT group into a new group. (GPRF 99.26)

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

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

   1. Dynamic analysis is performed (response spectra or time history)

   2. Pseudo Static Loadings are created

   3. Buckling Analysis is Performed

   4. Internal member results are output or used in a subsequent steel design after the Buckling Analysis.

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

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

Workaround:

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

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

(GPRF 2004.14)

4.3 GTMenu

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

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

   (GPRF 95.18)

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

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

   (GPRF 95.21)

3. 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)

4. 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)

5. When using the new Refine Finite Element Mesh option with elements which have joint temperature loads, joint temperatures will not be generated correctly except for the first element that was refined (split).

4.4 Scope Environment

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

CHAPTER 5

PRERELEASE FEATURES

5.1 Introduction

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

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

The Prerelease features in Version 30 are subdivided into Design, Analysis, and General categories. The features in these categories are shown below:

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

5.2 Design Prerelease Features

5.2.1 AISC13 Steel Design Code

The code is primarily based on the AISC *Steel Construction Manual, Thirteenth Edition, Specification for Structural Steel Buildings* adopted March 9, 2005. The Specification is contained in the Thirteenth Edition of the AISC Steel Construction Manual (102). The AISC13 code utilizes the Load and Resistance Factor Design (LRFD) techniques of the AISC Specification and also the safety factor Ω has been implemented for the Allowable Strength Design (ASD).

The AISC13 code of GTSTRUDL may be used to select or check any of the following shapes:

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

I shapes
Channels
Round HSS (Pipes)
Rectangular and Square HSS (Structural Tubes)
Single Angles

Design for bi-axial bending and axial forces:

Tees
Double Angles
Square Bars
Round Bars
Rectangular Bars

The documentation for the AISC13 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and AISC13 in the GTSTRUDL Output Window.

5.2.2 LRFD3 Steel Design Code

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

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

Design for bi-axial bending and axial forces:

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

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

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

The documentation for the LRFD3 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and LRFD3 in the GTSTRUDL Output Window.

5.2.3 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.

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

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

Example:

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

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

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

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

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

Notes:

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

2. VU is multiplied by PHISH internally.

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

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

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

This page intentionally left blank.

5.2.4 Rectangular and Circular Concrete Cross-Section Tables

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

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

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

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

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

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

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

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

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

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

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

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

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

The form of the command is as follows:

$$
\begin{array}{l}
\underline{\text{DES}}\text{IGN } \underline{\text{SLAB}} \text{ (}\underline{\text{REIN}}\text{FORCEMENT) (}\underline{\text{US}}\text{ING) } - \\
\left\{ \begin{array}{l} \underline{\text{WOOD}} \text{ (}\underline{\text{AND}}\text{) (}\underline{\text{ARM}}\text{ER)} \left\{ \begin{array}{l} \rightarrow \underline{\text{AVE}}\text{RAGE} \\ \quad \underline{\text{MAX}}\text{IMUM} \end{array} \right\} \\ \underline{\text{CALC}}\text{ULATE (}\underline{\text{RES}}\text{ULTANT) (}\underline{\text{ELE}}\text{MENT) (}\underline{\text{FOR}}\text{CES)} \end{array} \right\} \text{ (}\underline{\text{AL}}\text{ONG) } - \\
\text{(}\underline{\text{CUT}} \left\{ \begin{array}{l} \text{'a'} \\ \text{i}_1 \end{array} \right\}\text{) } \left\{ \begin{array}{l} \underline{\text{JOI}}\text{NTS} \\ \underline{\text{NODE}}\text{S} \end{array} \right\} \text{ list}_1 \ \underline{\text{ELE}}\text{MENT list}_2 \text{ (}\underline{\text{TABLE}} \left\{ \begin{array}{l} \rightarrow \underline{\text{ASTM}} \\ \quad \underline{\text{UNESCO}} \end{array} \right\}\text{)} - \\
{}^{*}\left\{ \begin{array}{l} \underline{\text{TOP}} \text{ (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S i}_2\text{) (}\underline{\text{SPA}}\text{CING v}_1\text{)} \\ \underline{\text{BOT}}\text{TOM (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S i}_3\text{) (}\underline{\text{SPA}}\text{CING v}_2\text{)} \\ \underline{\text{BOTH}} \text{ (}\underline{\text{FACE}}\text{S) (}\underline{\text{BAR}}\text{S i}_4\text{) (}\underline{\text{SPA}}\text{CING v}_3\text{)} \end{array} \right\} - \\
\left\{ \begin{array}{l} \rightarrow \underline{\text{IN}}\text{NER (}\underline{\text{LAY}}\text{ER)} \\ \quad \underline{\text{OUT}}\text{ER (}\underline{\text{LAY}}\text{ER)} \end{array} \right\} \text{ (}\underline{\text{COV}}\text{ER v}_4\text{) (}\underline{\text{LIN}}\text{EAR (}\underline{\text{TOL}}\text{ERANCE) v}_5\text{) } - \\
\text{(}\underline{\text{TOR}}\text{SIONAL (}\underline{\text{MOM}}\text{ENT) (}\underline{\text{WARN}}\text{ING) v}_6\text{)}
\end{array}
$$

where,

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

$list_1$	=	list containing ID’s of the start and end node of the cut
$list_2$	=	list containing the ID of an element in the plane of the cut
i_2	=	bar size to be used for bars on the top surface of the slab
i_3	=	bar size to be used for bars on the bottom surface of the slab
i_4	=	bar size to be used for both the top and bottom surfaces of the slab
v_1	=	reinforcing bar spacing to be used on the top surface of the slab
v_2	=	reinforcing bar spacing to be used on the bottom surface of the slab
v_3	=	reinforcing bar spacing to be used on both surfaces of the slab
v_4	=	optional user-specified cover distance for reinforcing bars
v_5	=	linear tolerance used in element selection rules for moment computation
v_6	=	optional ratio of torsion to bending moment allowed on the cross-section
TOP	=	element surface with +Z PLANAR coordinate
BOTTOM	=	element surface with -Z PLANAR coordinate

Explanation:

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

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

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

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

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

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

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

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

Requirements:

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

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

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

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

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

Usage:

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

Usage Example: Description of Example Structure

The example structure is a rectangular slab system, shown in Figure 5.2.5-1. The clear span of the structure is thirty feet, and the slab strip has a width of ten feet. The two ends of the slab are fully fixed, while the thirty foot sides are free, resembling a fixed-fixed beam. The slab is one foot thick and constructed of normal strength concrete with FCP = 4000 psi. The example structure can be idealized as a subset of a larger slab system, perhaps the design strip running between two column faces in an interior region. The structure is loaded with a distributed surface pressure of 150 psf over the entire surface of the slab.

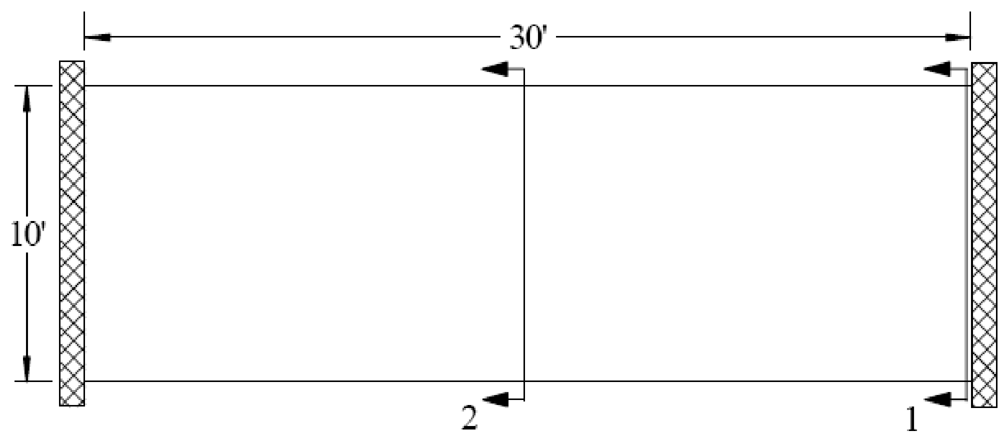
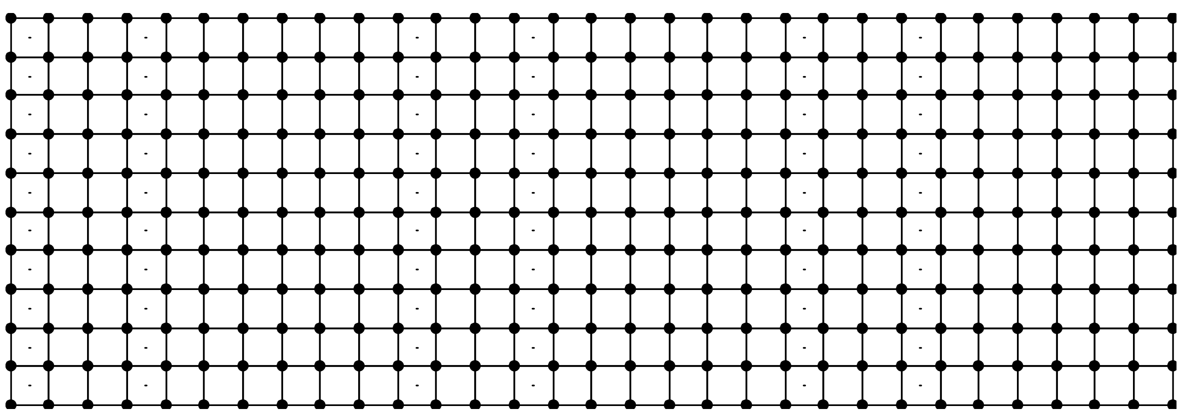


Figure 5.2.5-1 Example Flat Plate Structure (PLAN)

GT STRUDL Finite Element Model

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

Figure 5.2.5-2 Example Finite Element Model



Definition of Cut Cross-Sections

Two "cuts" are considered for the verification example, as shown in Figure 5.2.5-1.

Cut 1-1:

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

"DESIGN SLAB USING CALCULATE RESULTANT JOI 1 11 ELE 1 TOP BAR 5"

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

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

The GTSTRUDL output for this example is as follows:

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

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


     RELEVANT ACTIVE UNITS:  INCH LB

     NUMBER OF ACTIVE LOADINGS:     1

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


** ELEMENT FORCE IMPLEMENTATION **


** DESIGN MOMENT ENVELOPE **

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

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


** SLAB CROSS-SECTION **

     Width        Depth          FCP           FY         Cover        Layer
    ________________________________________________________________________

    120.00       12.00        4000.00       60000.00      0.750        Inner


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

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

     TOP        # 5    13.000      2.862        1561006.4280    1354381.4844     PASSES

     BOTTOM             ( Reinforcement Not Required )
```

Cut 2-2:

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

> “DESIGN SLAB WOOD AND ARMER JOI 166 176 ELE 141 TABLE UNESCO BOTTOM SPACING 10 OUTER LAYER"

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

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

The GTSTRUDL output for this example is as follows:

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

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


       RELEVANT ACTIVE UNITS:  INCH LB

       NUMBER OF ACTIVE LOADINGS:     1

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


  ** WOOD & ARMER IMPLEMENTATION **

       Design using average result acting on section.


  ** DESIGN MOMENT ENVELOPE **

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


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


  ** SLAB CROSS-SECTION **

       Width        Depth          FCP           FY        Cover        Layer
      ________________________________________________________________________

      120.00       12.00        4000.00       60000.00     0.750        Outer


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

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

________________________________________________________________________________________

       TOP               ( Reinforcement Not Required )

       BOTTOM     M14    10.000      2.864        1664920.7190     671358.1875     PASSES
```

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

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

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

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

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

The L_2 norm is given by:

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

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

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

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

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

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

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

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

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

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

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

Maximum Difference Method

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

Difference from Average Method

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

Percent Maximum Difference Method

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

Percent Difference from Average Method

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

Normalized Percent Maximum Difference

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

Normalized Percent Difference from Average Method

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

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

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

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

1. Enter GTMenu.

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

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

4. Select the Loading.

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

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

5.3.2 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

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

2.4.3.7 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

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

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

2.4.3.7.1 The DAMPER ELEMENT DATA Command

Tabular form:

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

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

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

•

•

•

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

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

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

Elements:

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

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

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

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

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

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

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

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

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

Explanation:

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

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

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

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

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

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

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

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

$$\begin{Bmatrix} \rightarrow \underline{\text{GLO}}\text{BAL} \\ \underline{\text{LOC}}\text{AL} \end{Bmatrix}$$

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

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

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

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

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

Command processing is completed when the END option is given.

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

Modifications:

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

Example:

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

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

Errors:

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

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

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

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

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

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

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

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

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

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

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

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

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

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

2.4.3.7.2 The PRINT DAMPER ELEMENT DATA Command

General form:

PRINT DAMPER (ELEMENT DATA)

Explanation:

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

Example:

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

```
{   17} > PRINT DAMPING ELEMENT DATA

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


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


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

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

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

Errors:

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

```
{    9} > PRINT DAMPING ELEMENT DATA

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


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


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

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

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

2.4.3.7.3 The DELETE DAMPER ELEMENT DATA Command

General form:

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

Elements:

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

Explanation:

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

This page intentionally left blank.

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

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

Elements:

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

r_1, r_2, r_3 = values in current angle units of the load component rotation angles θ_1, θ_2, θ_3 as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

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

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

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

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

1. Rotate all joint loads, including applied joint support displacements.

2. Rotate all member force and moment loads defined with respect to the global coordinate system. Member force and moment loads defined with respect to the member local coordinate system are simply copied without rotation.

3. Rotate all element force loads defined with respect to the global coordinate system. Element force loads defined with respect to any applicable local or planar coordinate systems are copied without rotation.

4. All other types of loads such as member temperature loads, member distortions, joint temperatures, etc. are copied without changes.

Examples:

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

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

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

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

Error Messages:

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

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

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

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

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

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

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

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

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

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

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

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

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

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

5.4.2 Reference Coordinate System Command

General form:

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

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

Explanation:

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

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

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

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

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

In either case, i_1 or 'a_1' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x, v_y, and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x, v_y, and v_z, are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive directions of these axes; v_1, v_2, and v_3 are the rotation angles R_1, R_2, and R_3 in active angular units between the orthogonal axes of this system and the axes of the overall global coordinate system. The description of these angles is the same as given in Section 2.1.7.2 of Volume 1 of the GTSTRUDL User Reference Manuals for rotated joint releases (θ_1, θ_2, and θ_3).

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

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

Modifications of Reference Systems:

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

Examples:

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

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

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

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

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

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

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

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

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

The above command deletes Reference System 2.

5.4.2-1 Printing Reference Coordinate System Command

General form:

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

Explanation:

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

5.4.3 GTMenu Point and Line Incidences Commands

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

(1) **GTMenu POINT COORDINATES**

General Form:

GTMenu POINT COORDINATES

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

.
.
.

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

Elements:

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

Where,

$i_1, i_2, ..., i_n$	=	unsigned integer Point identifiers.
'a_1', 'a_2', ..., 'a_n'	=	1 to 8 character alphanumeric Point identifiers.
v_1, v_2, v_3	=	Cartesian Point coordinates (integer or real).

(2) **GTMenu LINE INCIDENCES**

General Form:

GTMenu LINE INCIDENCES

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

$$\vdots$$

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

Elements:

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

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

Where,

$i_1, i_2, ..., i_n$	=	unsigned integer Line/Curve identifiers.
‘a_1’, ‘a_2’, ..., ‘a_n’	=	1 to 8 character alphanumeric Line/Curve identifiers.
$i_1, i_2, ..., i_p$	=	unsigned integer Point identifiers used.
‘a_1’, ‘a_2’, ..., ‘a_p’	=	1 to 8 character alphanumeric Point identifiers.
v_1	=	positive number (integer or real).
k_2	=	integer between 2 and the number of incidences.
1, 2, ...,p	=	Point subscripts for a Line/Curve. The following table gives the number of Points used to specify different types of Line/Curve:

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

End of Document