GT STRUDL®
Release Guide

PROCESS, POWER & MARINE

Volume 1 of 2

Version 33, February 2014

PART OF HEXAGON

INTERGRAPH®

Notice

This GT STRUDL Release Guide Volume 1 of 2 is applicable to GT STRUDL Version 33 and higher numbered versions for use on PCs under the Microsoft Windows 8/7/Vista/XP operating systems.

Intergraph Corporation
300 Intergraph Way
Huntsville, AL 35813

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Manuals, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on eCustomer, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee’s internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee’s internal, non-commercial. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

Table of Contents

Chapter **Page**

CHAPTER 3 ERROR CORRECTIONS

CHAPTER 4 KNOWN DEFICIENCIES

CHAPTER 5 PRERELEASE FEATURES

This page intentionally left blank.

Chapter 1

Introduction

Version 33 is the first release of GTSTRUDL since GTSTRUDL was acquired by Intergraph Corporation from Georgia Tech Research Corporation. Version 33 covers GTSTRUDL operating on PC's under the Windows 8, Windows 7, Windows Vista and Windows XP operating systems. Version 33 will be the last release tested to run on the Windows XP operating system as Microsoft is scheduled to drop support for Windows XP in 2014.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 32. Chapter 3 provides you with details regarding error corrections that have been made since the Version 32 release. Chapter 4 describes known problems with Version 33. 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

5.2.1 LRFD3 Steel Design Code. Rather than use the LRFD3 code which is a prerelease feature, users should use the AISC13 or AISC 14 codes which are released features.

5.2.2 ACI Concrete Code 318-99. Rather than use the 318-99 code which is a prerelease feature, users should use the 318-05 code which is a released feature.

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

5.2.4 ISO 19902:2007(E) code for fixed steel offshore structures.

5.2.5 ASCE4805 Steel Design Code. This code is for the ultimate strength design of steel transmission pole structures.

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 33

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

2.1 Dynamics

1. Nonlinear dynamic analysis has been enhanced to detect when a solution is diverging and to issue a warning message so indicating.

2. Speed and memory management improvements have been made to the GTSELANCZOS eigenvalue analysis solution. An example of the improvements for a large model are shown below:

#Elements	60,744
#Joints	59,736
#DOF	354,408

Time to Solve for 100 Modes

Version 32	445 seconds
Version 33	118 seconds

3. The maximum allowable number of time points used for the acceleration time history integration procedure of the CREATE RESPONSE SPECTRUM command is increased from 1 million to 4 million.

Documentation

Section 2.4.8.2, Volume 3

4. The MODE FACTORS COMPUTE option of the RESPONSE SPECTRUM LOAD command has been updated to include the selection of the LINDLEY-YOW method for the computation of the rigid and periodic mode factors pursuant to the provisions of the U.S. Nuclear Regulatory Commission Regulatory Guide 1.92 Revision 2.
The new command syntax is shown below:

$$\underline{\text{MOD}}\text{E } (\underline{\text{FAC}}\text{TORS}) \begin{Bmatrix} r_1 \ (i_1) \ r_2 \ (i_2) \ldots \ r_m \ (i_m) \\ \underline{\text{COM}}\text{PUTE compute specs} \end{Bmatrix}$$

where

$$\text{compute specs} = \begin{Bmatrix} \underline{\text{RIG}}\text{ID} \\ \underline{\text{PER}}\text{IODIC} \end{Bmatrix} \begin{Bmatrix} \underline{\text{GUP}}\text{TA } (\underline{\text{F1}} \ v_{F1}) \begin{Bmatrix} \underline{\text{F2}} \ v_{F2} \\ \underline{\text{FZPA}} \ v_{FZPA} \end{Bmatrix} \\ \underline{\text{LIN}}\text{DLEY-YOW } \underline{\text{F1}} \ v_{F1} \ \underline{\text{FZPA}} \ v_{FZPA} \end{Bmatrix}$$

Documentation

Section 2.4.4.6.3, Volume 3

5. A new option, BROADEN, has been added to the CREATE RESPONSE SPECTRUM command. When the BROADEN option is given, the created response spectrum points are also peak-broadened in a manner suggested by NRC Regulatory Guide 1.122 (Reference 69, Volume 3) and specified by ASCE 4-98, Section 3.4.2.3 (Reference 118, Volume 3), using a frequency variation factor that can be specified directly or taken as 0.15 by default. There is also an optional cutoff frequency parameter that allows a maximum frequency – less than or equal to the maximum frequency of the unbroadened, intermediate response spectrum file – to be imposed on the broadened response spectrum.

   Documentation

   Section 2.4.8.2, Volume 3.

6. A new command, CREATE RESPONSE SPECTRUM FILE command, has been implemented for Version 33. The CREATE RESPONSE SPECTRUM FILE command is used to envelope and broaden a set of input response spectrum files in a manner suggested by NRC Regulatory Guide 1.122 (Reference 69, Volume 3) and specified by ASCE 4-98, Section 3.4.2.3 (Reference 118, Volume 3).

The syntax of the CREATE RESPONSE SPECTRUM FILE command is shown below:

$$\underline{\text{CRE}}\text{ATE }\underline{\text{RES}}\text{PONSE }(\underline{\text{SPE}}\text{CTRUM})\ \underline{\text{FIL}}\text{E 'rs\_file'}$$

$$(\underline{\text{ACC}}\text{ELERATION}\begin{Bmatrix}\rightarrow \underline{\text{LIN}}\text{EAR}\\ \underline{\text{LOG}}\end{Bmatrix})\ (\begin{Bmatrix}\underline{\text{VER}}\text{SUS}\\ \underline{\text{VS}}\end{Bmatrix})\ (\underline{\text{FRE}}\text{QUENCY}\begin{Bmatrix}\rightarrow \underline{\text{LIN}}\text{EAR}\\ \underline{\text{LOG}}\end{Bmatrix})$$

$$\underline{\text{USE}}\begin{Bmatrix}\underline{\text{MAX}}\text{IMUM}\\ \underline{\text{RMS}}\end{Bmatrix}(\underline{\text{OF}})\ \underline{\text{FIL}}\text{ES list}_{\text{RSF}}$$

$$(\underline{\text{BRO}}\text{ADEN }(\begin{Bmatrix}\rightarrow 0.15\\ [\underline{\text{FVF}}]\ v_{\text{FVF}}\end{Bmatrix})\ [\underline{\text{CF}}]\ v_{\text{CF}})$$

$$(\underline{\text{LIS}}\text{T }\underline{\text{COM}}\text{MANDS }([\text{NPPL}]\ v_{\text{NPPL}}))$$

$$\underline{\text{END}}\ (\underline{\text{OF}}\ \underline{\text{CRE}}\text{ATE }\underline{\text{RES}}\text{PONSE }\underline{\text{SPE}}\text{CTRUM})$$

Documentation

Section 2.4.8.3, Volume 3.

7. Stiffness and mass matrix instabilities and/or ill-conditionality may cause convergence failure of the GTSELANCZOS/GT64MLANCZOS tridiagonalization Lanczos vector iterations, further causing the GTSELANCZOS/GT64MLANCZOS eigenvalue analysis to end silently, without providing any information about the failure of the analysis to produce results. This has been improved in Version 33, with the following message now being issued when this type of analysis failure is detected:

```
**** ERROR_GT64MLZEVA -- Problem detected in Lanczos vector  iterations.
        The model may contain unremovable instabilities. Analysis terminated.
```

2.2 Finite Elements

1. The CALCULATE AVERAGE command now has a SUMMARY ONLY option. If selected, only the summary for each active load is printed while results for individual joints are suppressed. If the ENVELOPE option is included, then the summary over all active loadings is printed.

   Syntax:

   CALCULATE AVERAGE *type* ((AND) ENVELOPE) ((SUMMARY) ONLY) -
   surface list

   Where *type*, *surface* and *list* are defined in the Reference documentation

   Documentation:

   Section 2.3.7.2, Volume 3, GTSTRUDL Reference Manual

2.3 General

1. The external file solvers GTSES, GT64M, GTSELANCZOS and GT64MLANCZOS now use the folder "GTSresults" for all external files. "GTSresults" is created, if needed, in the current GTSTRUDL Working Directory. This results folder name is fixed and no user control is provided. This change will reduce the "clutter" in the Working Directory when the external file solvers are used.

   Note: To be able to restore external file results saved (using the SAVE command) under version 32 or earlier and restored under version 33 (using the RESTORE command), before running GTSTRUDL 33 create a folder named "GTSresults" and copy all the external files to the new folder. A practical way to do this would be to select all files in the Working Directory, excluding only the files you are sure that are not external result files (.gti, .txt, etc.), and copy them to "GTSresults". It is allowable to have unreferenced files in the "GTSresults" folder so if extra files are copied there no harm is done.

   Documentation (Reference Manual, Volumes 1 and 3):

   2.1.13.2 The STIFFNESS ANALYSIS Command
   2.1.13.4 The ACTIVE SOLVER Command
   2.1.13.5 The PERFORM NUMERICAL INSTABILITY ANALYSIS Command
   2.4.5.2 Eigenproblem Solution Specifications

2.4.10.2 DYNAMIC ANALYSIS NONLINEAR Command
2.5.4.3 NONLINEAR ANALYSIS Command

2. Prerelease features (Chapter 5) will now output a warning message indicating they are in prerelease status. This change allows Quality Assurance customers a simple way to be aware if they are using prerelease features.

   Example :

```
{   694} > design slab using wood and armer average joi 1 11 ele 1

 ****WARNING_CRCCMN - This feature is a Prerelease feature.
                      User feedback and suggestions are welcome.
```

3. A new ALL option has been added to the list option of the PHYSICAL MEMBER COMPLIANCE CHECK command. This option allows user to check the compliance of all physical members in a single command.

   Documentation:
   Section 2.1.2.10.2, Volume 1.

4. The CREATE AUTOMATIC command has two new features. First is the option to create FORM LOADS, which are independent loads suitable for nonlinear analysis. Second is the option to use multiple "SETS" of EXCLUSIVE loads, which allow you to create loadings by combining one load (or load group) from each SET. For example, this gives you the ability to create all combinations of Live and Wind loads by putting the different load types in different SETs.

   Note: COMBINATION is no longer assumed and you must follow CREATE AUTOMATIC with either COMBINATION or FORM.

$$\underline{\text{CRE}}\text{ATE}\ \ \underline{\text{AUT}}\text{OMATIC} \left\{ \begin{array}{l} (\underline{\text{LOA}}\text{D})\ \underline{\text{COM}}\text{BINATIONS} \\ \underline{\text{FOR}}\text{M}\ (\underline{\text{LOA}}\text{DS}) \end{array} \right\} (\text{id\_spec})$$

$$\left\{ \begin{array}{l} \underline{\text{ONE}}\ (\underline{\text{ONL}}\text{Y})\ (\underline{\text{OF}}) \\ \underline{\text{EXC}}\text{LUSIVE} \\ \underline{\text{IND}}\text{IVIDUAL} \end{array} \right\} (\underline{\text{SET}}\ s_{id})\ (\underline{\text{LOA}}\text{DS})\ \text{spec}_3$$

Example: Multiple EXCLUSIVE commands in different SETs

```
CREATE AUTOMATIC LOAD COMBINATIONS INITIAL 101
    CONSTANT 'DL' 1.4
    EXCLUSIVE 'LL1' 'LL2' 'LL3' 'LL4' 1.6    $ Assumed SET 1
    EXCLUSIVE SET 2 'WL1' 'WL2' 1.7          $ Specified SET 2
END
```

This command set generates the following 8 loads. Each load from SET 1 is combined with a load from SET 2. Since there are 4 loads in SET 1 and 2 loads in SET 2 the total number of created loads is 4 x 2 = 8. The number of load lists in a SET does not change the total number of combinations created, although more loads would appear in each generated load case.

Documentation:

"Section 2.1.11.3.6 Create Automatic Load Combinations", Volume 1, GTSTRUDL Reference Manual.

5. RESTORE will now internally re-issue a "LARGE PROBLEM SIZE n" command if such a command had been given when the SAVE command was executed. This only applies to version 33 and later - .gts files created under earlier versions can be restored but no automatic LARGE PROBLEM SIZE command will be generated.

   If the LARGE PROBLEM SIZE command is generated an INFO message is printed:

   ```
   ****INFO_XXREST - LARGE PROBLEM SIZE 5 restored.
   ```

6. Three new features have been added to the SELF WEIGHT LOADING RECOMPUTE command (Section 2.1.11.3.1.2, Volume 1). The first two options can be used to recompute active and inactive (default) or active only self weight loads. The third option allows the user to reform active form loads also.

   Documentation:

   Section 2.1.11.3.1.2, Volume 1.

7. The CALCULATE PRESSURE command now only evaluates elements or element faces that lie in the specified plane, e.g. PLANE Y EQUAL 0.0 if no element list is given. Previously, all elements in the model were evaluated and the command would fail if elements existed that were not on the specified plane.

Example:

CALCULATE PRESSURE PLANE Y 0.0 SUMMARY ONLY

Will now report pressures based on only those elements or element faces that lie in the plane Y = 0.0.

Documentation:

Section 2.1.12.19 The CALCULATE PRESSURE Command

8. Active solver status information has been added to the data base summary report produced by the QUERY and RESTORE commands.

Documentation:

QUERY Command, Section 2.1.12.2, Volume 1
The SAVE/RESTORE Commands, Section 2.1.2.4.2, Volume 1

9. The CALCULATE SOIL SPRING VALUES command now has a PRINT OFF option to suppress individual joint output. This prevents large amounts of output when the specified plane contains many joints. When the COMPRESSION ONLY option is not used, a summary of how many joints and tributary elements were involved is the only output.

Syntax:

CALCULATE (SOIL) SPRING (VALUES) (cmp_only) -

$$\text{DIRECTION} \begin{Bmatrix} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{Bmatrix} (v_1) \ ((\text{PLANE})\ \text{TOLERANCE}\ v_2) \ -$$

((PLANE) ANGLE (TOLERANCE) v_3) value_sets -
(APPLY) (PRINT (OFF)) (SHOW (CALCS)) (INPLANE (RATIO) v_6)

Example:

```
{   90} > CALCULATE SOIL SPRING VALUES DIR Y -
{   91} >_ KS 0.1 ELEMENTS 1 TO 8 KS 0.2 ELEMENTS 9 TO 16 APPLY PRINT OFF

****INFO_SSPR06 - Elastic springs type KFY were created for    25 joints
                  based on the tributary areas of    16 elements.
                  You can review the values with the PRINT JOINT RELEASES command.
```

Documentation:

Section 2.1.12.13 The CALCULATE SOIL SPRING VALUES Command, Volume1, GTSTRUDL Reference Manual

10. STIFFNESS ANALYSIS will now print a warning message if any statics checks failures are found. This new message will trigger the Warning Message facility in the GTSTRUDL Output Window (Section 2.6) to alert you to the existence of statics check failures in case they are not visible in the window due to following data.

Examples:

STIFFNESS ANALYSIS

```
 STATICS CHECK FAILED FOR JOINT 5        FOR LOADING CONDITION 6        AT DOF DX
 APPLIED LOAD=  0.00000E+00 CALCULATED LOAD=  0.50000E+04 COMPARISON VALUE=  0.13277E+02

 COMPARISON VALUES ARE  0.132768E+02 0.132768E+02 0.100000E-06 0.100000E-06 0.100000E-06 0.349391E+03

 TIME FOR STATICS CHECK                             0.00 SECONDS

 ****WARNING_STD3BS - One or more statics check failures detected.
                      Review the above messages.
```

STIFFNESS ANALYSIS GTSES

```
 Statics check failed for joint 2989     for loading condition DLX      at DOF DZ
 applied load =  0.00000E+00 calculated load =  0.30621E+03 comparison value =  0.21930E+03
 Comparison values are  0.219301E+03 0.219301E+03 0.219301E+03 0.850579E+03 0.850579E+03 0.850579E+03

 Time for statics check                             0.06 seconds

 ****WARNING_EFD3BS - One or more statics check failures detected.
                      Review the above messages.
```

11. The AREA LOAD command has been enhanced with two new features for ELEVATION. The EXCEPT option will now accept multiple regions, allowing the specification all the “holes” (stairwells, atria, etc.) in a floor in a single ELEVATION command. A new option IGNORE MEMBERS has been added to allow the specification of members to not include in the AREA LOAD calculations. Previously these unloaded members, such as braces, needed to be INACTIVE before issuing the AREA LOAD command.

In addition, a new message has been added:

```
 ****INFO_STARLD - A positive elevation VALUE will create loads in the -Y direction.
                   A negative elevation VALUE will create loads in the +Y direction.
```

This message follows each AREA LOAD command and reminds the user of the AREA LOAD sign convention.

Examples:

ELEVATION 36.75 EXCEPT LIMITS JOINTS 345 789 LIMITS MEMBERS -
45 78 124 158 VALUE TWO WAY 100.0

Two "holes" are not loaded, the first bounded by joints 345 and 789 and the second outlined by members 45, 78, 124 and 158.

ELEVATION 36.75 IGNORE MEMBERS 'B1' TO 'B12' VALUE TWO WAY 100.0

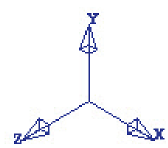
The areas and loadings are determined without considering bracing members 'B1' to 'B12'. These members will not be loaded and their ACTIVE/INACTIVE status is not affected.

Documentation:

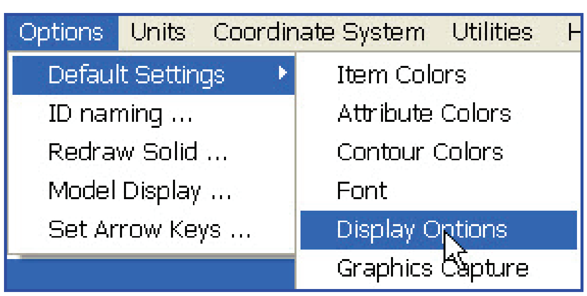
Section 2.1.11.3.7 The AREA LOAD Command, Volume 1, GTSTRUDL Reference Manual.

2.4 GTMenu

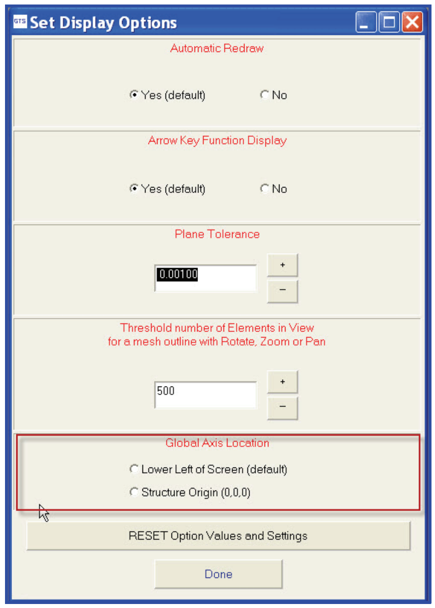
1. The global axes in the display area are now displayed as 3D arrows as shown below:



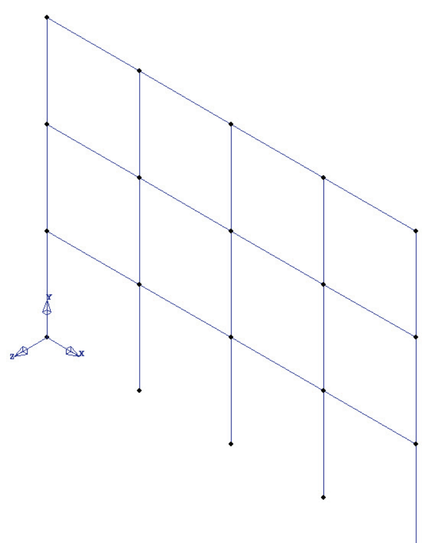
2. The location for the display of the global axes may be changed to the origin of the global coordinate system using the revised Set Display Options dialog as shown below:



The revised Display Options dialog is shown below where you may select to have the global axes displayed at the lower left corner of the screen (the default) or the structure (global) origin:

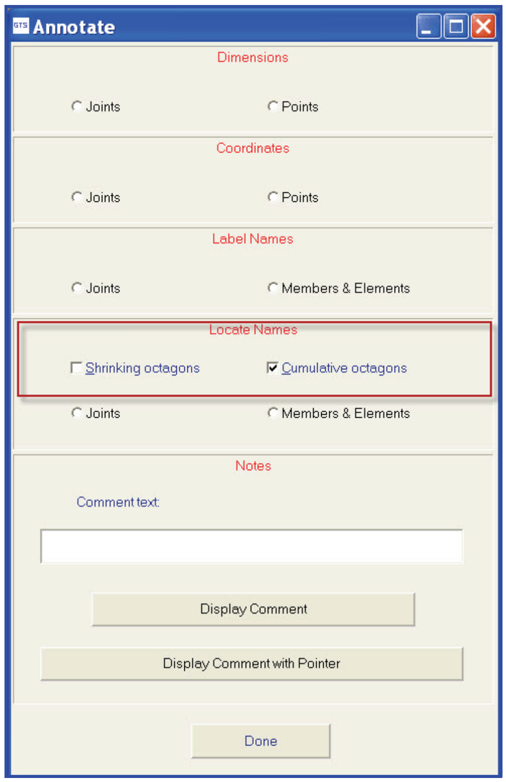


An example is shown below with the axes displayed at the global (Structure) origin which is located in the lower left corner of the structure:



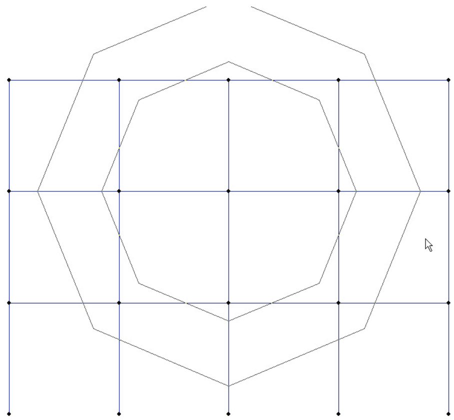
If a window or view is created which doesn't contain the global origin, the axes will be displayed in the window or view.

3. A new feature has been implemented to facilitate the location of a single joint, member or element on large models or on models with members and elements closely spaced. You may activate this feature either from the Annotate Model button on the Button Bar or by entering LJ, LE or LM when the Display Area is active. The revised Annotate dialog is shown below:

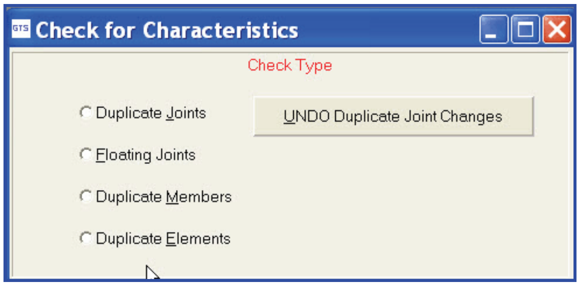


If the Shrinking octagons option is selected in the above dialog, the joint, member or element name will be located by drawing octagons which decrease in size centered on the entity. If the Cumulative octagons option is selected, the joint, member or element name will be located by a series of octagons which remain on the screen as they are drawn centered on the entity before they are erased.

An example of the cumulative octagons drawn around a joint in the model is shown below:

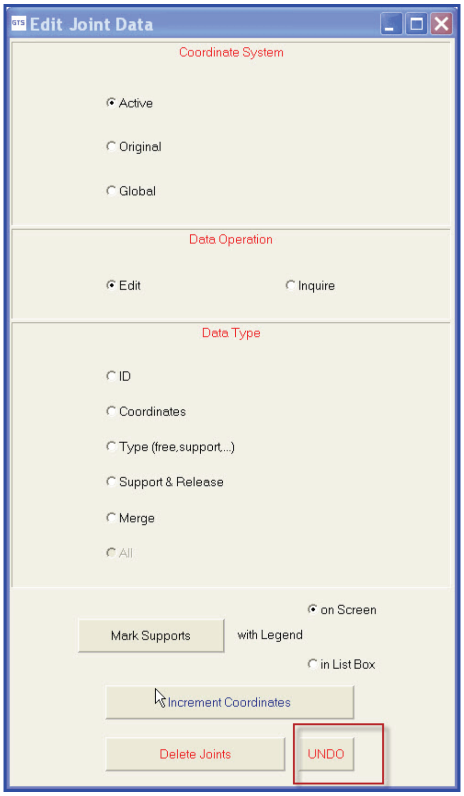


4. Undo functions have been added to the Check Duplicate s & Floating Joints functions under the Check Model pulldown. This feature has been requested by many users who have removed duplicate joints resulting in joints being removed which they didn't want removed. The revised Check Characteristics dialog is shown below where the UNDO Duplicate Joint Changes button appeared after deleting duplicate joints:



An UNDO button will also appear after deleting floating joints or duplicate members and elements.

5. UNDO functions have been added to the Edit dialogs for Joints, Members and Elements. The UNDO button will only appear in these dialogs if joints, members or elements have been deleted. An example of the UNDO button in the Edit Joint dialog is shown below:



6. The STOP box will now always appear in the Graphics Window when animating mode shapes or deformed shapes.

7. Redraw Solid now uses the element color for the cable element, IPCABLE, and the transmission line element, IPTL. Previously, the color for curves and lines was used for the cable and transmission line elements when using Redraw Solid.

8. Display Model will now display the cross section for members that have properties created using the Member Property Channel and Polygon cross-section specs. The Polygon cross sections will be drawn as pipes.

9. Point and curve labels are now placed closer to the location of the points and curves.

10. The time to display wire frame models which contain a large number of finite elements has been greatly reduced.
11. The time for Redraw Solid has been greatly reduced for large models.

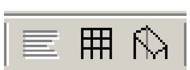
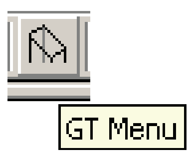
12. The time to Exit GTMenu and save the changes has been reduced for models which contain a large number of solid elements.

13. Display Model will now display the cross section for members that have properties created using the Member Property Channel and Polygon cross-section specs. The Polygon cross sections will be drawn as pipes.

14. Display Parameters will now display the parameters UNLCFTF and UNLCFBF.

2.5 GTSTRUDL Output Window

1. The Console and Scope Environment buttons have been removed from the Button Bar. The Scope Environment is no longer a supported feature and the Console button is no longer needed to return from the Scope Environment window. The GTMenu button will remain on the Button Bar, but note that it has been moved to the left of the message buttons.

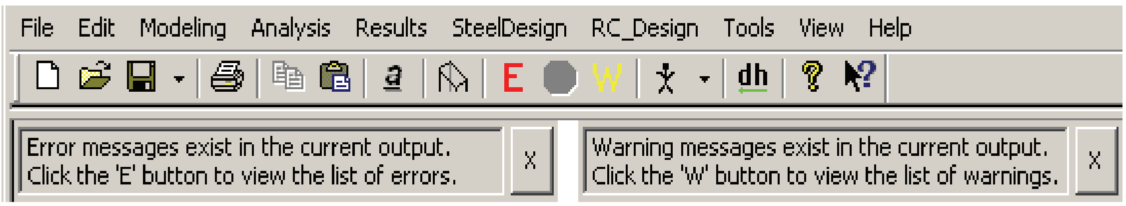
   Before:  Now: 

2. A new message alert system has been implemented. The words “ERROR” and “WARNING” in the output are detected and handled by the new system. The new system consists of two new buttons on either side of the Scan button (whose function is unchanged), two new alert windows that appear just below the Button Bar and a new message dialog that allows you to see the messages and to automatically scroll the output to the selected message. This new system alerts you to important output messages that may have scrolled out of view as commands are processed.

   If no Errors or Warnings have been found, the new buttons are inactive.

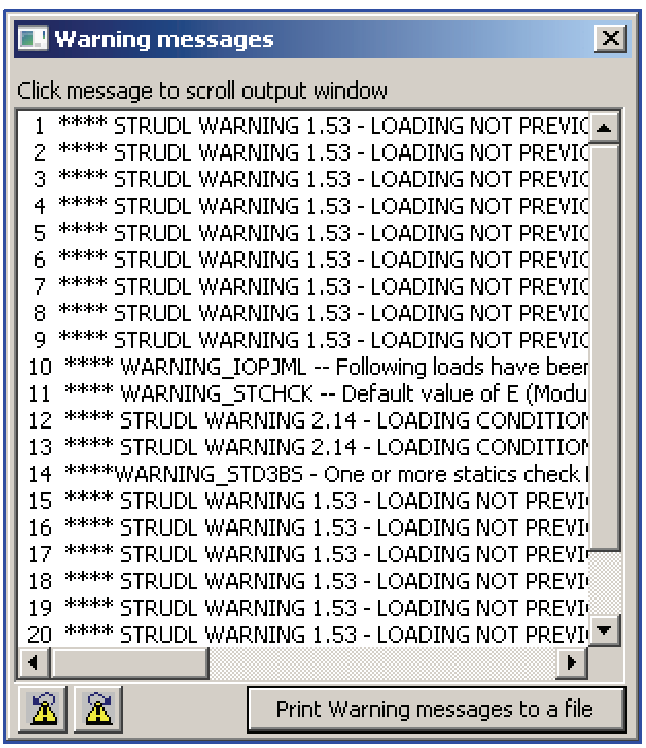
   

If Error or Warning messages are found in the output, the appropriate button becomes active, and in addition, an alert window is opened underneath the button.



The alert windows can be closed by clicking the "X" on the right side of the alert window. Also, if you click the message button ("E" or "W"), the associated alert window is automatically closed if it is open. The alert windows will re-open if a subsequent message (Error or Warning) is found after the alert window is closed.

<u>Message Dialog</u>



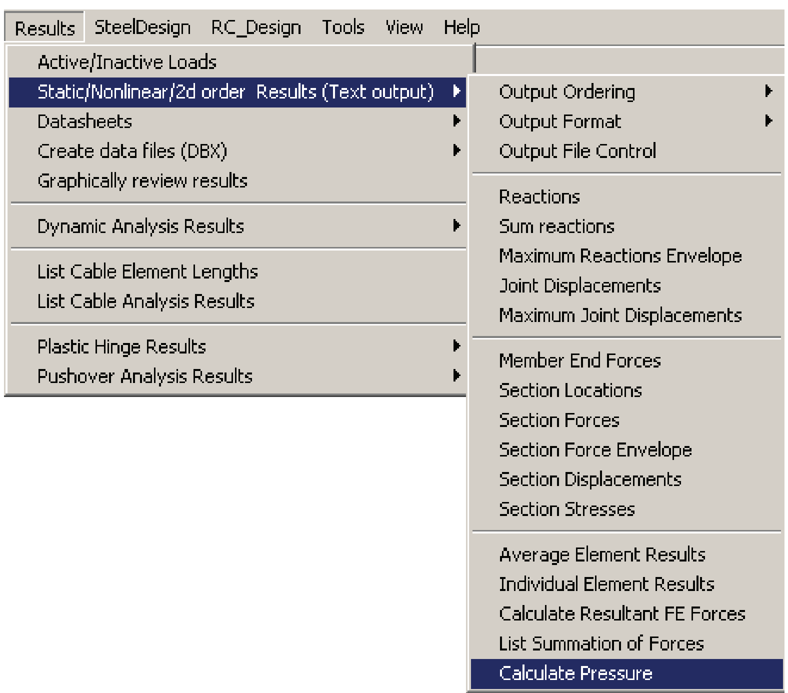
When you click an active message button (the "W" button for the above example), a message dialog will pop up. By default, it will appear in the upper, right corner of the Output window, but you can move it, even completely off the GTSTRUDL window. Each line in the output that contains the appropriate characters ("WARN" or "Warn" in this case) is inserted into the message dialog. The messages are numbered as they are inserted, but this is informative only.

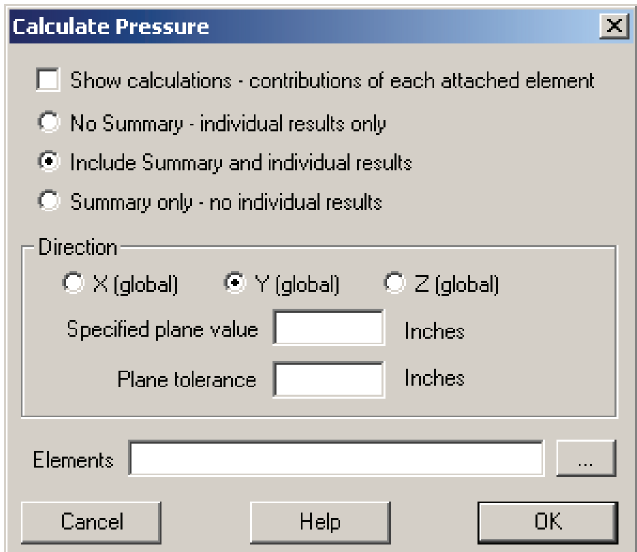
You can click on any message and the Output window will scroll to that line of the output. Also, you can click the "Previous" or "Next" buttons in the lower, left corner of the dialog to step through the messages sequentially.

The "Print Warning messages to a file" button will create a file WarningMessages.txt in the Working Directory and open the file in Notepad. Use 'Save As....' to save the file to a new name if you want to keep the file. If you are viewing Error messages, the file is named ErrorMessages.txt.

The Message Dialog may be closed with the "X" in the upper, right corner of the dialog and may be re-opened at any time by clicking the active message button "E" or "W".

3. A new dialog has been added to the 'Results - Text output' menu for the CALCULATE PRESSURE command. It is accessed by the menu pick shown below. Click the Help button in the Calculate Pressure dialog to learn more about its operation.

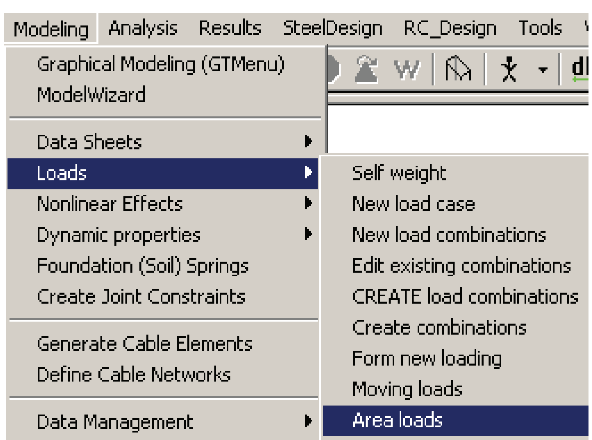


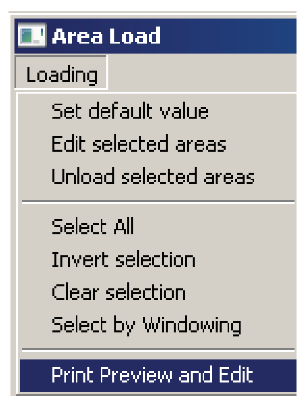


Documentation:

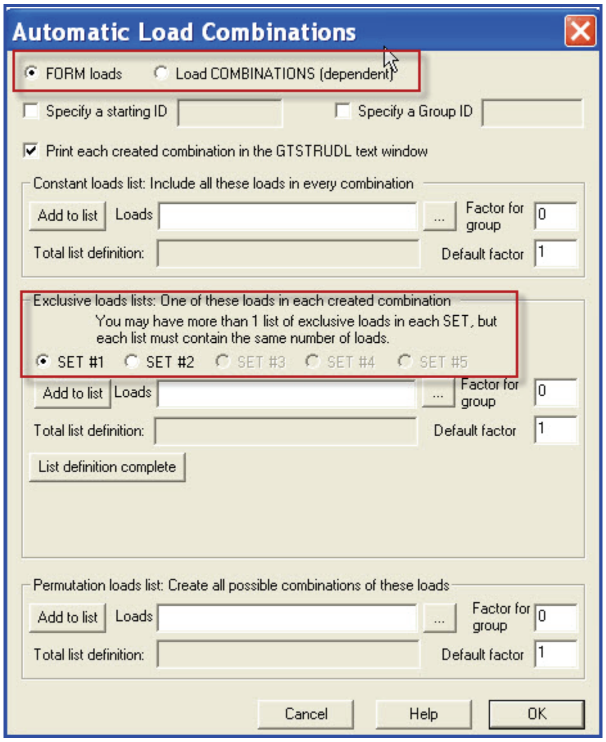
Section 2.1.12.19 The CALCULATE PRESSURE Command.

4. The Area Load dialog has a new option in the Loading menu: Print Preview and Edit. This option sends the Area Load display to the Scope Editor where you can edit and print the elevation display.





5. The dialog for Automatic Load Combinations and Automatic Form Loads has been revised to support the new features in the Create Automatic Load Combination and Form Load commands described in Section 2.3. The revised dialog is shown below:



2.6 Nonlinear

1. The NONLINEAR SPRING PROPERTIES command now accepts integer nonlinear spring curve point data where in previous versions only decimal curve point data were accepted.

Documentation:

Section 2.5.3.1.3, Volume 3, The NONLINEAR SPRING PROPERTIES Command

2.7 Offshore

1. The SAVE command now saves the data computed by the COMPUTE FATIGUE LJF command. This enhancement enables the COMPUTE FATIGUE LJF command and a subsequent PEFORM FATIGUE ANALYSIS command to be executed in consecutive jobs separated by SAVE and RESTORE commands, the first job executing the COMPUTE FATIGUE LJF command and ending with a SAVE command and the second job beginning with a RESTORE command and executing the PERFORM FATIGUE ANALYSIS command.

2. A new option has been added to the PRINT FATIGUE command as follows:

   PRINT FATIGUE NUMBER OF WAVES

   The PRINT FATIGUE NUMBER OF WAVES command prints a tabular report of the number of waves of the various distinct wave height-period pairs in each of the specified wave directions. The format of the new report is shown by the example below:

```
UNITS METERS
 PRINT FATIGUE NUMBER OF WAVES

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

     PROBLEM - FA101    TITLE - ST5825 -- PRINT FATIGUE WAVE OCCURENCES

     ACTIVE UNITS  M    N    DEG  DEGF SEC



     Numbers of Waves from WAVE PASSAGE Data
     =======================================

     Ordered by Height-Period and Direction

     Count (BASE) Period =  0.500 years

                                                             Directions (degrees)
                                                             --------------------
```

Height	Period	Sum	0.00	45.00	135.00	90.00	180.00	225.00	270.00	315.00
1.00	4.00	4196165	566137	201799	378150	338757	468618	687287	586870	968547
2.00	5.00	1738413	148038	63846	130293	136190	202183	318968	282168	456727
3.00	5.80	574695	33445	17105	41638	43300	59792	114137	110998	154280
4.00	6.50	227421	9604	5475	15741	16951	21699	48272	49642	60037
5.00	7.00	97752	3171	1900	6212	7267	8623	21537	23422	25620
6.00	7.50	44813	1163	692	2495	3262	3663	9991	11564	11983
7.00	8.00	21368	472	252	1004	1470	1624	4758	5875	5913
8.00	8.50	10485	213	89	405	653	739	2305	3056	3025
9.00	8.90	5261	104	30	166	285	341	1123	1624	1588
10.00	9.30	2692	54	9	70	122	159	546	880	852
11.00	9.70	1406	28	3	32	52	75	265	485	466
12.00	10.10	745	15	1	15	22	36	127	270	259
13.00	10.50	400	7	0	8	9	18	61	151	146
14.00	10.80	217	4	0	4	4	9	29	84	83
15.00	11.20	116	2	0	2	1	4	14	46	47
16.00	11.50	63	1	0	1	1	2	6	25	27

17.00	11.80	33	0	0	0	0	1	3	14	15
18.00	12.10	17	0	0	0	0	0	1	7	9
19.00	12.40	10	0	0	0	0	0	1	4	5
20.00	12.70	5	0	0	0	0	0	0	2	3
Totals:		6922077	762458	291201	576236	548346	767586	1209431	1077187	1689632

Documentation:

Section 5.7.1, Volume 8

3. A new option has been added to the LIST FATIGUE command as follows:

$$\underline{\text{LIS}}\text{T } \underline{\text{FAT}}\text{IGUE } \underline{\text{DAM}}\text{AGE} \begin{Bmatrix} \rightarrow \underline{\text{PERC}}\text{ENT} \\ \underline{\text{X1000}} \end{Bmatrix} (\text{points}_2)\ (\text{weld side})\ (\text{joints})$$

where

$$\text{points}_2 = (\underline{\text{POI}}\text{NTS } \underline{\text{CRI}}\text{TICAL})$$

$$\text{weld side} = ({}^{*}\begin{Bmatrix} \underline{\text{BRA}}\text{CE } \underline{\text{SID}}\text{E} \\ \underline{\text{CHO}}\text{RD } \underline{\text{SID}}\text{E} \end{Bmatrix})$$

$$\text{joints} = (\underline{\text{JOI}}\text{NTS list}_J)$$

This command produces a report of the fatigue damage in a table ordered on wave height/period and direction. The report is produced for the critical point (point with the highest damage) at the end of each brace member (FATIGUE MEMBER, Section 5.3.1, Volume 8, CHORDS FOR FATIGUE, Section 5.3.3, Volume 8) connected to each specified joint. This report also includes the center of damage with respect to wave height/period for each direction. The format of the new report is shown by the example below:

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

PROBLEM - FA105    TITLE - ST5830 -- Implement the LIST FATIGUE WAVE DAMAGE Command

ACTIVE UNITS  FEET N    DEG  DEGF SEC

Base Period =  0.500 years

Fatigue Damage Percentage at Critical Point, Wave H-T vs Direction
==================================================================

 Joint    Brace    Chord    Point    Side
-------- -------- -------- ------- --------
14       31       14           3    Chord
                                            Directions (degrees)
                                            --------------------
 H-T #     Height    Period         0.00      90.00     180.00    225.00    315.00     Sum
-------    ------    ------       /--------//--------//--------//--------//--------//--------/
    1        3.28      4.00          0.000     0.000     0.000     0.000     0.000     0.000
```

```
       2        6.56      5.00           0.000      0.000      0.000      0.000      0.000      0.000
       3        9.84      5.80           0.000      0.000      0.493      0.000      1.270      1.763
       4       13.12      6.50           0.589      0.000      1.165      0.000      3.214      4.968
       5       16.40      7.00           0.000      0.021      1.884      0.062      5.571      7.538
       6       19.68      7.50           0.000      0.030      2.504      0.090      8.139     10.762
       7       22.97      8.00           0.000      0.034      2.620      0.110      9.498     12.262
       8       26.25      8.50           0.000      0.034      1.923      0.119      7.837      9.913
       9       29.53      8.90           0.000      0.028      1.310      0.111      6.078      7.527
      10       32.81      9.30          16.548      0.022      0.886      0.099      4.700     22.255
      11       36.09      9.70           0.000      0.015      0.563      0.079      3.457      4.114
      12       39.37     10.10           0.000      0.009      0.321      0.050      2.282      2.661
      13       42.65     10.50           0.000      0.005      0.208      0.036      1.663      1.913
      14       45.93     10.80           0.000      0.003      0.130      0.025      1.187      1.346
      15       49.21     11.20           0.000      0.001      0.071      0.017      0.826      0.915
      16       52.49     11.50          10.589      0.002      0.043      0.009      0.572     11.215
      17       55.77     11.80           0.000      0.000      0.025      0.006      0.376      0.408
      18       59.05     12.10           0.000      0.000      0.000      0.002      0.264      0.266
      19       62.34     12.40           0.000      0.000      0.000      0.003      0.171      0.173
   -------    ------    ------       /--------//--------//--------//--------//--------//--------/
   Sums at Point:                       27.726      0.205     14.147      0.818     57.104    100.000
   Center of Damage (H-T#):                 10          8          7          9          8
```

Documentation:

Section 5.7.2, Volume 8

4. The following fatigue analysis commands have been enhanced to support the use of the GTSES sparse equation solver for static analysis when activated by the ACTIVE SOLVER GTSES/GT64M command: ASSEMBLE FOR FATIGUE, SOLVE FOR FATIGUE, COMPUTE FATIGUE LJF and PERFORM FATIGUE ANALYSIS.

5. A new GTSELOS feature has been implemented whereby additional tables express the drag parameters CD and CM as function(s) of one or more of the existing pipe member diameter, water depth, Reynolds number parameters, plus the additional parameters of roughness and Keulegan-Carpenters Number.

2.8 Steel Design

1. A new AISC 14th Edition (adopted on June 22, 2010) design code has been implemented as a released feature. Both LRFD (load and resistance factor design) and the ASD (allowable strength design) methods of the AISC Fourteenth Edition are implemented. This new code, AISC14, may be used to select or check any of the following shapes:

    Design for bi-axial bending and axial forces:

I shapes	Channel
Single Angles	Tees
Double Angles	Solid Round Bars
Solid Rectangular and Square Bars	

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

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

The documentation for the AISC14 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and AISC14 in the GTSTRUDL Output Windows.

2. The ANSI/AISC N690-12, specification for safety-related steel structures for nuclear facilities has been added as a released feature. This code is based on the AISC steel construction manual, Fourteenth Edition specification with a few modifications. Both LRFD (load and resistance factor design) and the ASD (allowable strength design) methods of the AISC Fourteenth 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 1.4 of the AISC14 code. The documentation for the N690-12 or AISC14 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and N690-12 or AISC14 in the GTSTRUDL Output Window.

3. The transmission tower provisions, slenderness ratio and computation of the number of bolts, are now implemented in AISC14 code. When a value of YES has been specified for parameter TowerCK, transmission tower provisions are checked in addition to the provisions of the AISC14 code. Applicable cross-sections are single and double angles. The documentation may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and AISC codes with Transmission Tower Provisions in the GTSTRUDL Output Windows.

4. Prerelease steel design codes (Chapter 5) will now output a warning message indicating they are in prerelease status. This change allows Quality Assurance customers a simple way to be aware if they are using prerelease features.

Example :

```
 {   76} > CHECK  MEMBERS  6  9


 **** WARNING_MSPCCK - LRFD3    is a prerelease steel design code.
                            User feedback and suggestions are welcome.

 **** Note:  Specify a value of 'NO' for parameter 'WarnMess' to prevent the
above
                      message to be printed
```

5. Elastic shear resistance of the EC3-2005 has been modified to use the first moment of area. The local shear stress, τ_{ED}, in the Equation 6.20 of the EN 1993-1-1:2005(E) is now computed using the first moment of area (e.g. S). The thickness, t, in the Equation 6.20

also has been modified to use the total thickness of the cross-section which resist the shear force.

$$\tau_{ED} = \frac{V_{ED}\ S}{I\,t} \qquad \text{EC3 Eq. 6.20}$$

Also Provisions ‘6.19 Y’ and ‘6.19Z’ have been modified as such that these provisions are only considered when the member is a Class 3 cross-section.

6. A new steel grade, A1011G33, has been added to the AISI89 code for the UNISTRUT cross-sections. This is the ASTM steel grade A1011 with a yield stress of 33 ksi.

7. Computation of the allowable bending stress for I Shapes, whose maximum unbraced length of the compression flange exceeds L_c, using the N690-94, NF-2004, and NF2007 codes has been modified. The new procedure uses $0.6F_yQ_s$ as the allowable bending stress when the maximum unbraced length of the compression flange is not long enough for the allowable bending stress to be reduced by equations Q1.5-5b in the N690-94 code or 22.1d517 in the NF-2004, or NF-2007 codes. This prevents the use of the allowable bending stress given by equations Q1.5-6 or 22.1d519, in their respective codes, which are grossly conservative for certain I Shapes with high d/A_f ratio.

2.9 Steel Tables

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

W-AISC14	W shapes from Table 1-1 of the AISC 14th Edition
M/S/HP14	M, S, and HP shapes from Tables 1-2, 1-3, and 1-4 of the AISC 14th Edition
C-AISC14	Channel C and MC shapes from Tables 1-5 and 1-6 of the AISC 14th Edition
L-ALL-14	Single angles from Table 1-7 of the AISC 14th Edition
L-EQ-14	Equal leg single angles from Table 1-7 of the AISC 14th Edition
L-UN-14	Unequal leg single angles from Table 1-7 of the AISC 14th Edition
WTAISC14	Tee WT, MT, and ST shapes from Tables 1-8, 1-9, and 1-10 of the AISC 14th Edition
RecHSS14	Rectangular and square HSS from Tables 1-11 and 1-12 of the AISC 14th Edition
RdHSS14	Round HSS from Table 1-13 of the AISC 14th Edition
2L-ALL14	Double angles from Table 1-15 of the AISC 14th Edition
2L-EQ-14	Equal legs double angles from Table 1-15 of the AISC 14th Edition

2L-LL-14	Long legs back-to-back double angles from Table 1-15 of the AISC 14th Edition
2L-SL-14	Short legs back-to-back double angles from Table 1-15 of the AISC 14th Edition
WBEAM-14	W shapes commonly used as beams from Table 3-6 of the AISC 14th Edition
WCOL-14	W shapes commonly used as columns from Table 4-1 of the AISC 14th Edition

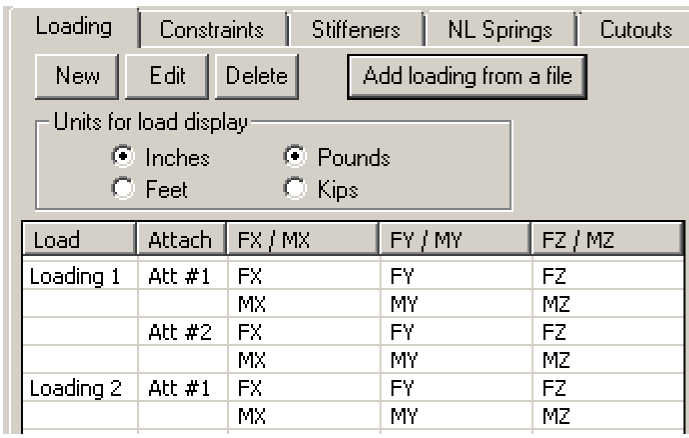
2. The following new Metric Tables from AISC Fourteenth Edition have been added to GTSTRUDL:

WAISC14M	W shapes from Table 1-1 of the AISC 14th Edition
MSHP-14M	M, S, and HP shapes from Tables 1-2, 1-3, and 1-4 of the AISC 14th Edition
CAISC14M	Channel C and MC shapes from Tables 1-5 and 1-6 of the AISC 14th Edition
L-ALL14M	Single angles from Table 1-7 of the AISC 14th Edition
L-EQ-14M	Equal leg single angles from Table 1-7 of the AISC 14th Edition
L-UN-14M	Unequal leg single angles from Table 1-7 of the AISC 14th Edition
WTAIS14M	Tee WT, MT, and ST shapes from Tables 1-8, 1-9, and 1-10 of the AISC 14th Edition
ReHSS14M	Rectangular and square HSS from Tables 1-11 and 1-12 of the AISC 14th Edition
RdHSS14M	Round HSS from Table 1-13 of the AISC 14th Edition
2LALL14M	Double angles from Table 1-15 of the AISC 14th Edition
2L-EQ14M	Equal legs double angles from Table 1-15 of the AISC 14th Edition
2L-LL14M	Long legs back-to-back double angles from Table 1-15 of the AISC 14th Edition
2L-SL14M	Short legs back-to-back double angles from Table 1-15 of the AISC 14th Edition
WBEAM14M	W shapes commonly used as beams from Table 3-6 of the AISC 14th Edition
WCOL-14M	W shapes commonly used as columns from Table 4-1 of the AISC 14th Edition

3. The tables for rectangular and circular concrete sections has been moved to a release status. The table names are CONCIR for circular concrete sections and CONRECT for rectangular concrete sections..

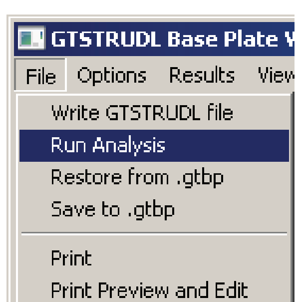
2.10 Base Plate Wizard

1. The Loading page has been changed, with a new layout and functionality. The load data display is no longer editable. Load data entry, editing and deletions is now done through dedicated dialogs available from buttons on the Loading page. A display units control has been added to allow you to see the loading data in units other than the default of inches and pounds or millimeters and Newtons. See Section 9 of the Base Plate Wizard Users Guide for more details.



2. Save to .gtbp no longer includes unreferenced load cases. A load case must have at least one value (FX, FY, FZ, MX, MY or MZ) to be included in the .gtbp file.

3. A new 'Run Analysis' selection has been added to the File menu. After loadings and boundary conditions (either anchors or constraints) have been added to the Base Plate or a .gtbp file with loadings and boundary conditions has been read, this menu selection will become active. When Run Analysis is selected, an input file including analysis and results will be generated and processed. See Section 10.1 of the Base Plate Wizard User Guide for more information.



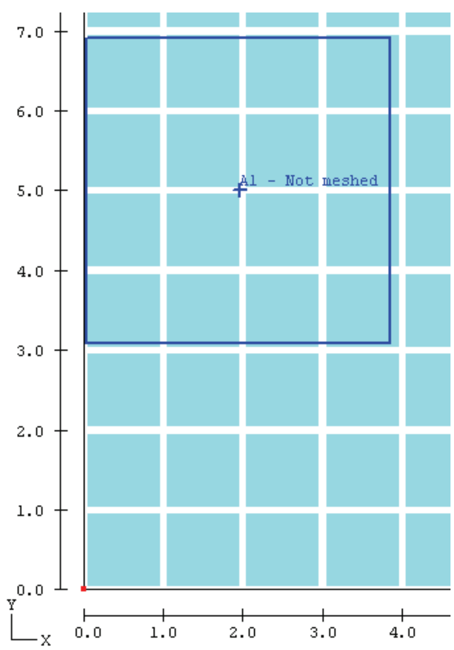
4. Two new pipe and hollow shape tables have been added to the GTStrudl\33\BasePlate folder: PIPES9.txt and RdHSS13.txt. These tables correspond to the GTSTRUDL

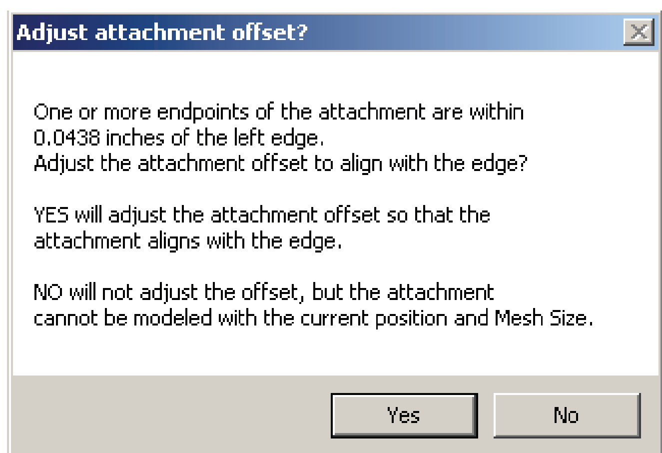
standard tables of the same name. The profiles can be accessed by using the Options - Attachment file menu pick or by concatenating one or both files to your current default attachment file.

5. When the Base Plate Wizard is minimized, the GTSTRUDL window is also minimized. This allows access to the Desktop without closing the Base Plate Wizard. Previously, because the Base Plate Wizard was a “modal” operation which prevented any interaction with the GTSTRUDL window, the GTSTRUDL window would not respond to a request to minimize.

6. When an attachment is added or edited and an endpoint of a line used to represent the attachment is within ½ standard mesh size of the plate edge, you will be queried if the attachment should be moved to fall exactly on the plate edge. Attachment points less than ½ of the mesh size from a plate edge cannot be modeled, so you must either move the attachment to fall on the plate edge or change the mesh size (on the FE Mesh tab) to be smaller than two times the distance from the attachment to the plate edge. An example is shown below:

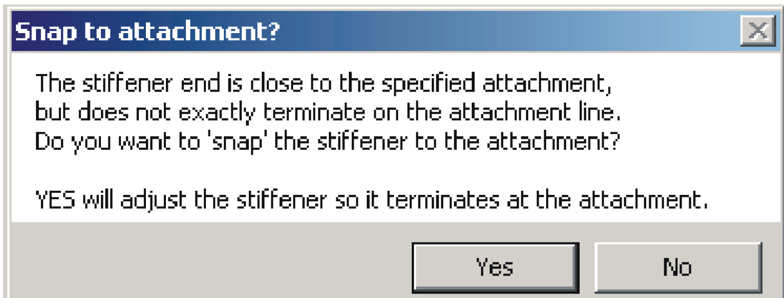
Example:



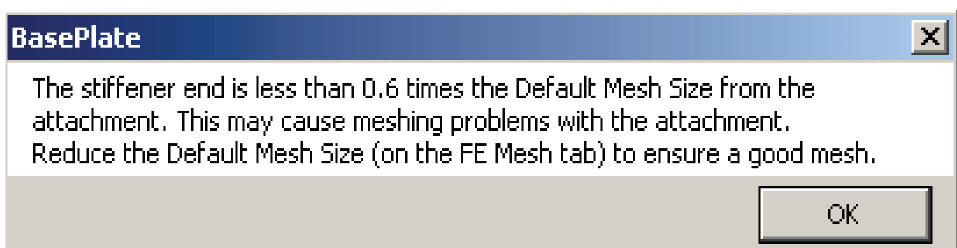


In this example, the tube is very close to the plate edge and the modeling algorithm cannot place joints along the plate edge and also align with the left side of the tube. This will generate the message shown above, reporting the 0.0438 inch separation. If you click the "Yes" button, the center of the tube will be shifted left (in the -X direction) so that the left side of the tube aligns exactly with the plate edge and modeling will be successful.

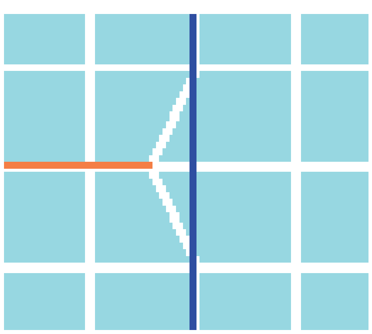
7. A new "Snap to attachment" feature has been added to Stiffeners. You will be alerted if the specified start or end of a stiffener is close to the attachment to which it is to be connected, but doesn't actually intersect the attachment center line. You will then have the option to have the start or end of the stiffener 'snap' to the attachment, which results in the start or end being adjusted to exactly intersect the attachment.



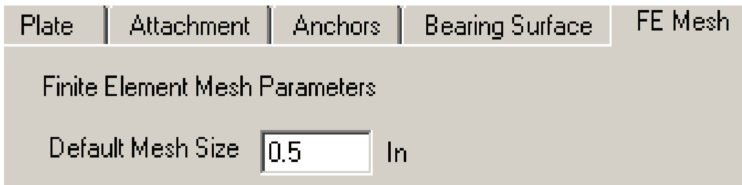
If you answer 'Yes' to the 'Snap to attachment' query, the end (or start) of the stiffener will be adjusted. If you answer 'No' to the above query, you will see the following message if the distance between the stiffener and the attachment is smaller than 0.6 times the standard Mesh Size of the base plate.



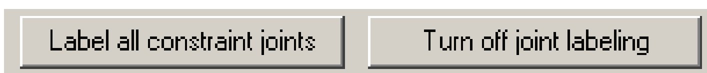
The reason for this warning is to inform you that the automated meshing algorithm of the Base Plate Wizard will adjust the plate mesh to match exactly with the end of the stiffener, which may cause the mesh to not line up with the attachment:



The user is responsible for deciding if the generated mesh is acceptable for analysis. The specified Default Mesh Size can be adjusted on the FE Mesh tab, as shown below:



8. The ability to display joint numbers for Constraints has been added. If results are requested for Constraints, they are reported by joint number. This new display will allow you to see which joint the listed values are associated with.



Label all constraint joints adds the joint ID to the display.

Turn off joint labeling returns to the standard display.

9. A new button has been added to the **Add a constraint** and **Edit selected constraint** dialogs, allowing you to fix translation DOF but leave rotation DOF unrestrained. Previously you had to set the translation DOF individually to Fixed.

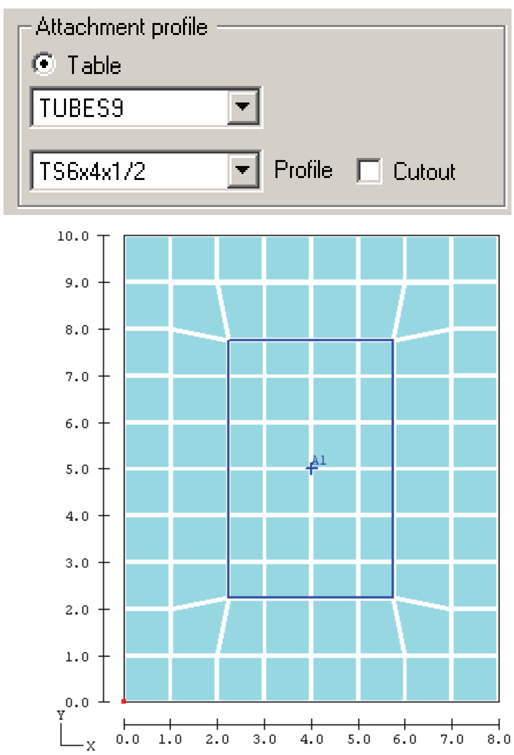
Set all DOF to Fixed

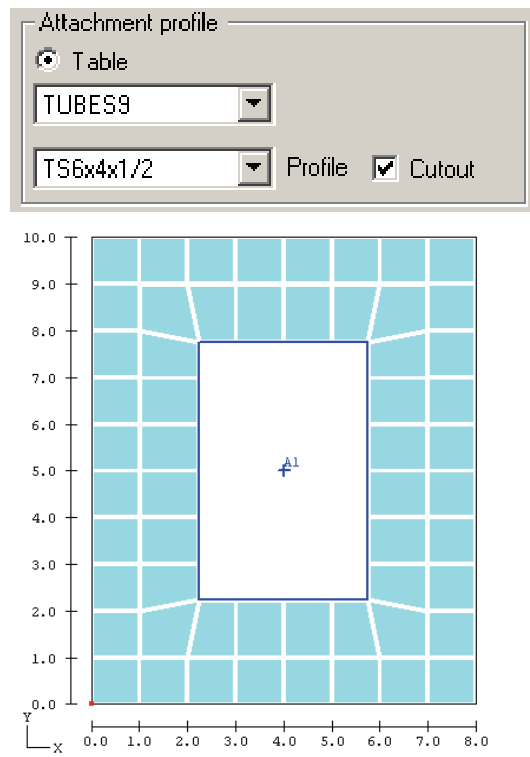
If you click this button, all joints associated with the Constraint will be declared as SUPPORT joints in GTSTRUDL, without any JOINT RELEASES. These joints are restrained for all translations and rotations.

Set only shear DOF to Fixed

If you click this button, all joints associated with the Constraint will be declared as SUPPORT joints in GTSTRUDL, but with JOINT RELEASES for MOMENT X Y Z. These joints are restrained for all translations, but rotations are unrestrained.

10. A new option is available for Tubes and Pipes, which allows the option of specifying that the interiors of the shape be removed from the base plate model in the same way as Cutouts (Section 2.3 of the Base Plate Wizard Users Guide). See "Section 3.4 Options for Angles, Tubes and Pipes" of the Base Plate Wizard Users Guide for more information.





11. A new results type has been added: Constraint Reactions.

Constraint Reactions are only applicable to constraints with restrained boundary conditions. Constraint Reactions are available through the Results - Constraint reactions menu pick, or as part of the Results Summary. Constraint Reactions consist of the force reactions FX, FY and FZ for each joint in the constraint, plus the SRSS (Square Root

Sum of the Squares) of the three shear values. You can request the reaction moments MX, MY and MZ if desired. Constraint Reactions are equivalent to the GTSTRUDL command “LIST REACTIONS JOINTS GROUP ‘CON1'” (for constraint #1).

12. A new results type has been added: Attachment Displacement.

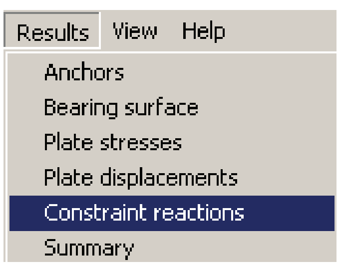
Attachment Displacement is the displacement of the control or load point of an attachment due to the applied loads. It is equivalent to the GTSTRUDL commands:

UNITS DEGREES $ Rotation units are degrees, translation units are inch

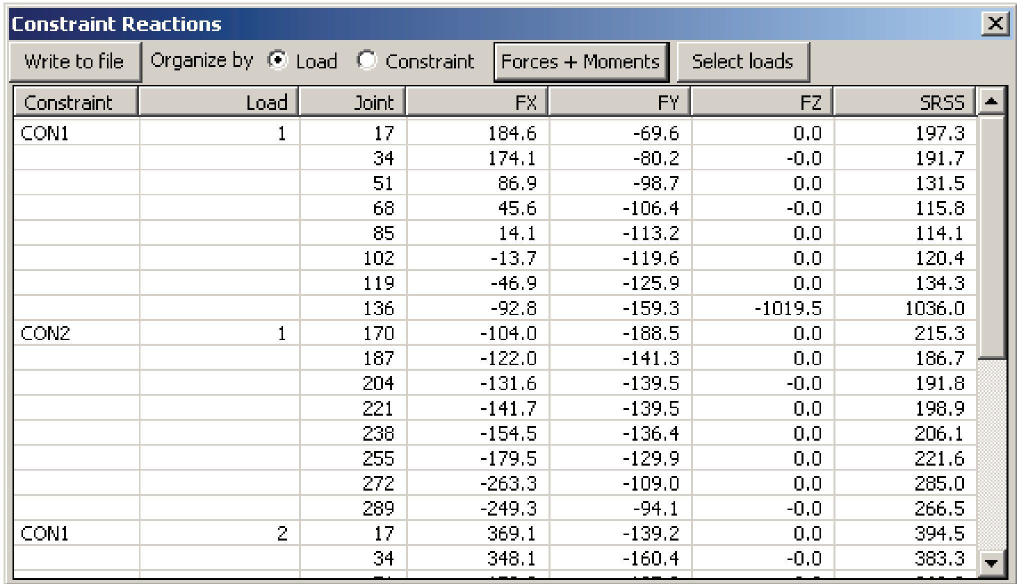
LIST DISPLACEMENTS JOINT ‘C1'

(for attachment #1). Attachment Displacement is only available through the Results Summary.

13. The Constraint Reactions results dialog has been added. It is activated through the Results menu.



You can use the “Forces + Moments” to include MX, MY and MZ in the new Constraint Reactions dialog shown below. The “Select loads” button allows you to select which loads are included in the display. By default all loads are included. Remember that only constraints with restrained boundary conditions are included.



Constraint	Load	Joint	FX	FY	FZ	SRSS
CON1	1	17	184.6	-69.6	0.0	197.3
		34	174.1	-80.2	-0.0	191.7
		51	86.9	-98.7	0.0	131.5
		68	45.6	-106.4	-0.0	115.8
		85	14.1	-113.2	0.0	114.1
		102	-13.7	-119.6	0.0	120.4
		119	-46.9	-125.9	0.0	134.3
		136	-92.8	-159.3	-1019.5	1036.0
CON2	1	170	-104.0	-188.5	0.0	215.3
		187	-122.0	-141.3	0.0	186.7
		204	-131.6	-139.5	-0.0	191.8
		221	-141.7	-139.5	0.0	198.9
		238	-154.5	-136.4	0.0	206.1
		255	-179.5	-129.9	0.0	221.6
		272	-263.3	-109.0	0.0	285.0
		289	-249.3	-94.1	-0.0	266.5
CON1	2	17	369.1	-139.2	0.0	394.5
		34	348.1	-160.4	-0.0	383.3

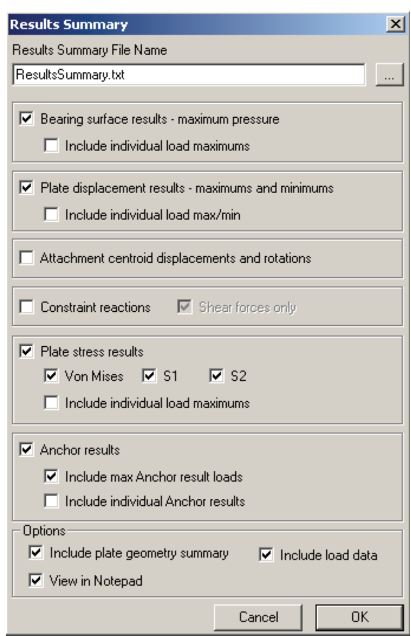
14. The Results Summary dialog and output have been changed.

Changes include:

• Individual load maximums for bearing surface pressure, plate displacement and plate stress.

• Selectable plate stress types.

• Attachment centroid displacements and rotations.



2.11 Scope Editor

1. As a result of the improvements to the wire frame display of models which contain a large number of finite elements, the time for Print Preview in the Scope Editor has been significantly decreased due to the reduced number of lines which are now being drawn.

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. When using the WRITE ELEMENT ATTRIBUTES command, the new elements introduced in version 32 (Q6CDRL, PBMITC, SBMITC and IPSLIM) now have assigned IELTYP (element type index) numbers as documented in “Notes for Table 1-3", Volume 5, GTSTRUDL Reference Manual. In version 32 these element type index numbers were written as 0 for the new elements, although the correct element type <u>names</u> were written to the character variable CELTYP.
(No GPRF issued)

3.2 Dynamic Analysis

1. In Versions 96.01 (1995) through 32, the presence of planar rigid bodies of the type RIGID PLANE, RIGID PLATE, and /or RIGID PIN in a model for which a dynamic eigenvalue analysis is executed could cause the LIST DYNAMIC MASS SUMMARY command to compute and report erroneous center of mass coordinates and erroneous gross mass moments of inertia. These errors have been corrected.
(GPRF 2012.03)

2. In Versions 96.01 (1995) through 32, the following errors related to the computation and reporting of mass participation factors could occur when planar rigid bodies of the type RIGID PLANE, RIGID PLATE, and /or RIGID PIN are defined in a structural model such that one or more structural members and/or finite elements are incident on any of the corresponding master joints:

   A. Erroneous dynamic participation factors could be computed and reported by the LIST DYNAMIC PARTICIPATION FACTOR command.

B. Erroneous response spectrum participation factors could be computed by the the PERFORM RESPONSE SPECTRUM ANALYSIS command. These mass participation factors are reported by the LIST RESPONSE SPECTRUM PARTICIPATION FACTORS command.

C. As a result of Item B, any response spectrum results computed by the COMPUTE RESPONSE SPECTRUM command could be in error.

These errors have been corrected. (GPRF 2012.04)

3. The CREATE TIME HISTORY no longer aborts or produces a time history file that contains only zero acceleration values when the specified joint is not defined or does not exist. If this condition is detected, the command now issues an error message as follows:

```
CREATE TIME HISTORY FILE 'MyTHFile' FROM JOINT 10000 -
TRANSLATION X LOADING 1

**** ERROR_QQCTH -- The specified joint does not exist. Scan mode
                    entered and analysis terminated.
```

In this example, the error message is issued because joint 10000, specified in the command, was found to be undefined and non-existent.
(GPRF 2013.03)

4. The COMPUTE MODAL DAMPING RATIOS command no longer aborts when damper elements are defined (the DAMPER ELEMENT DATA command) following the execution of the eigenvalue analysis that computes the modal data for the subsequent computation of the modal damping ratios.
(GPRF 2013.04)

5. The INERTIA OF JOINTS FROM LOADS command now correctly processes load lists when the following syntaxes are used:

A. INERTIA OF JOINTS FROM LOADS ‘ld1' TO ‘ld2'
B. INERTIA OF JOINTS FROM LOADS -
GROUP LIST ‘ldgrp1' ‘ldgrp2'...’ldgrpn’

(GPRF 2013.05)

6. The DYNAMIC ANALYSIS NONLINEAR... GTSES command no longer aborts when the ACTIVE SOLVER GTSES/GT64M and DYNAMIC PARAMETER commands are not given previously.
(GPRF 2013.07)

7. The COMPUTE HARMONIC STRESSES command for the SBMITC, Q6CDRL, PBMITC and IPSLIM elements will no longer print out extraneous information for each element in the model. Previously, one word such as MSBBKD, MQ6BKD, MPBBKD or STDBIM would be printed on a separate line for each element in the model.
(No GPRF issued)

8. The LIST STRESSES command no longer produces the message "RESULTS UNAVAILABLE DUE TO NUMERICAL INSTABILITY" for the planar finite element types SBMITC, PBMITC, and Q6CDRL under the following conditions:

   A. The desired element stresses and strains are computed by the COMPUTE HARMONIC STRESSES command.
   B. The eigenvalue analysis is performed using the GTLANCZOS or TRIDIAG or SUBSPACE eigenvalue analysis solver.

   (GPRF 2013.11)

9. The COMPUTE HARMONIC STRESSES command no longer aborts for the planar finite element types SBMITC, PBMITC, and Q6CDRL and when the eigenvalue analysis is performed using the GTSELANCZOS or GT64MLANCZOS eigenvalue analysis solver.
(GPRF 2013.11)

10. The LIST DYNAMIC PARTICIPATION FACTORS ORDER BY MODE ALL command will now output the mode number properly if more than 999 modes are present.
(No GPRF issued)

11. The CALCULATE AVERAGE command no longer reports erroneous average finite element stress results. This error occurred for PSEUDO STATIC loadings only if ***ALL*** of the following conditions were present:

   A. Both static and dynamic analyses are executed in the job using the standard (not GTSES or GT64M) solvers.

B. COMPUTE TRANSIENT / RESPONSE SPECTRUM / STEADY STATE / HARMONIC FORCES command(s) are executed only for a ***subset*** of the finite elements in the model.

C. COMPUTE TRANSIENT / RESPONSE SPECTRUM / STEADY STATE / HARMONIC STRESSES command(s) are not executed, i.e. no dynamic finite element stresses are computed for any of the elements in the model.

D. Pseudo static loads (CREATE PSEUDO STATIC command) are created from the dynamic loads that contain a subset of the dynamic element forces as computed according Item 2 above and do not contain dynamic element stresses according to Item 3 above.

E. A CALCULATE AVERAGE command is executed for one or more of the static loads for which a stiffness analysis was executed. The average element stress results reported by these CALCULATE AVERAGE commands are correct.

F. A CALCULATE AVERAGE command is subsequently executed (following Item 5) for one or more of the PSEUDO STATIC loads. These CALCULATE AVERAGE commands may report erroneous nonzero average element stress results. All average element stress results should be reported as 0.0.

(GPRF 2014.01)

12. The dynamic analysis assembly procedure, including assembly of the global stiffness and mass matrices, no longer produces superfluous messages of the following type:

**** WARNING_STJCLM -- Lumped mass not available for element 12345678 . Contribution to lumped mass matrix ignored.

when multiple FORM NOTIONAL LOAD commands are given prior to the INERTIA OF JOINTS FROM LOAD command and when the INERTIA OF JOINTS FROM LOAD command specifies the only type of mass matrix formulation.

(No GPRF issued)

13. The GT64MLANCZOS eigenvalue analysis no longer terminates with an error message indicating failure to allocate required memory when the product $(N_{modes})(N_{DOF}) \geq 2^{31}$, where N_{modes} is the number of modes requested and N_{DOF} is the total number of degrees of freedom in the model.

(GPRF 2014.02)

3.3 Finite Elements

1. The CALCULATE RESULTANT command will now process cuts that appear to be parallel to the Planar Y axis.
(GPRF 94.21)

2. The SUPERELEMENT DEFINITION command (Section 2.3.11.1, Volume 3) now enforces exclusivity with member and element names. If an attempt is made to name a superelement with an existing element or member name, an error message is generated and SCAN mode is entered.
(GPRF 95.08)

3. Curved elements, TYPE 'SCURV' and 'PCURV', will no longer accept CONCENTRATED or FORCE X UNIFORM loads. If these loads are applied to a curved element, a warning is issued during analysis and the incorrect load component is ignored.
(GPRF 99.13)

4. The PRINT MEMBER/ELEMENT PROPERTIES command will no longer abort when joint constraint rigid bodies are present in the model and rigidity matrix properties are specified for one or more finite elements.
(GPRF 2013.01)

5. GTSTRUDL error messages identified as **** STRUDL ERROR 4.06 – NO THICKNESS GIVEN FOR ELEMENT... and **** STRUDL ERROR 4.08 – INCORRECT NUMBER OF NODES FOR ELEMENT... , produced by the stiffness matrix computation operation for the SBMITC element, now report the correct information.

6. A temperature change loading applied to the SBMITC element now produces correct results.
(GPRF 2013.12)

3.4 General

1. The reported infinite loop for a GENERATE MEMBERS or GENERATE ELEMENTS command followed by a REPEAT command with an incorrect format has been corrected.
   (GPRF 93.22)

2. Rigid body elements may now be deleted as conventional finite elements. Rigid body elements may not be inactivated and are ignored if included in a list of elements to be inactivated.
   (GPRF 97.21)

3. The path plus file name for a SAVE or RESTORE command is documented as limited 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 be used in a GROUP LIST as part of a list or anywhere else GROUP may be used.
   (GPRF 99.26)

5. The Scope Environment is no longer supported and will not be included in Chapter 4 Known Deficiencies.
   (GPRF 96.19 is closed since the Scope Environment is no longer supported)

6. The error related to statics check failures and the incorrect analysis (all types) results caused by the use of large member eccentricities that produce ±90° member rotations between the joint-to-joint local member reference frame and the eccentric member reference frame has been corrected.
   (GPRF 2012.08)

7. The text-to-file functions of the GTSTRUDL Output window now do an automatic ‘Refresh text output’ before creating the file. This will prevent any text display anomalies (misaligned output, appended lines, etc.) from being sent to the generated file.
   (GPRF 2013.06)

8. The LIST SUM FORCES command no longer aborts when a pseudo static load created from a response spectrum ALGEBRAIC mode combination is processed

and the standard solvers (not the GTSES or GT64M solvers) are used to perform the analyses.
(GPRF 2013.08)

9. The CALCULATE PRESSURE command now reads the PLANE TOLERANCE and ELEMENT LIST as documented.
(No GPRF issued)

10. The REFORM (FORM LOAD) option of the STIFFNESS and NONLINEAR ANALYSIS command now works with the GTSES solver option.
(No GPRF issued)

11. Duplicate Construction Point names are no longer created as a result of processing commands which contain Construction Points or Curves with a gap in their naming.
(No GPRF issued)

3.5 GTMenu

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

1. The Refine Finite Element Mesh option has been modified to handle larger problems and to also correctly split framing members for a mesh containing two-dimensional finite elements.

2. GTMenu will no longer fail to open after the CHECK or SELECT commands have been used following the GTSES solver.

3. You may now specify a list of lines/curves in List Mode when creating joints and members or joints and elements. Previously, the list of lines/curves would not be recognized and an error message would appear on the screen stating "Too Few Curves to Define Surface".

4. You may now create a Surface of Revolution in List Mode. Previously, an error message would occur when entering the axis of revolution in List Mode stating "Point Not Found".

5. The incidence order may now be reversed correctly in the Edit Elements dialog for the new elements introduced in Version 32: Q6CDRL,PBMITC,SBMITC and IPSLIM.

6. Eliminating duplicate joints, members or elements no longer causes a load query to abort.

7. Rotation of a large structure now stops when the mouse button is released.

8. GTMenu no longer aborts if the structure has a joint object that doesn't have any members or elements.

9. A member's local axes are now drawn correctly for members with member eccentricities. Previously, the local x-axis would not be drawn on the member in its eccentric position.

10. The finite element aspect ratios output in Check Model now have the same values in the summary as in the detailed listing.

11. The text command input file produced by GTMENU now correctly includes the SBHQ, SBHQ6, SBHT, and SBHT6 planar finite elements that are specified as having geometric nonlinearity.

12. Command errors in the NONLINEAR SPRING PROPERTIES commands in the command text file produced by GTMENU have been corrected.

13. Label Member End Forces now works for pseudo-static loadings.

14. Display Joint Masses will now display "small" joint masses.

15. A long character string of up to 140 characters can now be pasted from the clipboard.

16. The Global Axes color is now saved between sessions. If the user wants to change the color used for the Global Axes, change the Annotation color in the Attributes dialog (select Options - Default Settings - Attribute Colors).

17. When entering GTMenu with a large model, the model is no longer drawn twice.

18. After extruding 2D elements to 3D elements, the generated input file no longer has a thickness for the newly created 3D elements. Previously, the thickness for the original 2D elements would be in the input file for the new 3D elements although it would be ignored when creating the 3D elements.

3.6 GTSTRUDL Output Window

1. The GTSTRUDL Output Window pulldown menu Results->Dynamic Analysis Results->Display Harmonic Analysis Results (vs Freq) no longer produces inconsistent/erroneous plots of harmonic analysis results for harmonic loads that contain duplicate frequencies.

3.7 Nonlinear Analysis

1. Pushover analysis no longer aborts when rigid bodies are present in the model and the pushover analysis encounters structural instabilities or convergence failures that cause the pushover analysis collapse search procedure to make adjustments to the incremental loading process.
   (GPRF 2013.10)

2. The PRINT NONLINEAR EFFECTS command now correctly includes the SBHQ, SBHQ6, SBHT, and SBHT6 planar finite elements that are specified as having geometric nonlinearity.
   (No GPRF issued)

3.8 Steel Design

1. The erroneous warning message issued after CHECK or SELECT command when a large number of load names specified for the parameters with the load list option (e.g. parameters ALSTRINC and DefLoads) has been corrected. (GPRF 2013.02)

2. The N690-94, NF-2004, and NF-2007 codes of GTSTRUDL have been corrected to accurately compute the Allowable Bending stress for Channel members when the ratio of Actual/Allowable values of the provision for laterally unsupported length of the compression flange is greater than 1.0 for such members.
   (GPRF 2013.09)

3. An abort no longer occurs when a polygonal shape is code checked with the ASCE4805 code and the resultant shear is zero, or the axis perpendicular to the direction of the shear resultant passes through one of the corners of the polygon.
   (No GPRF issued as this is a pre-release feature)

3.9 Welded Connection Design

1. The erroneous error message issued when the MEMBER ECCENTRICITIES are used to move a member to a new location for welded connection member-to-member configuration numbers 254 and 255 has been corrected.
   (GPRF 2012.09)

3.10 Base Plate Wizard

1. The header for the file generated by the ‘Write to File’ button in the Results - Bearing surface dialog has been corrected. Previously it read “Displacement Results” - it now reads “Pressure Results”. Also, the “Units:” that are printed are “psi + pounds” instead of “in” or “MPa + Newtons” instead of “mm”.
   (No GPRF issued)

2. When multiple base plates are analyzed without shutting down GTSTRUDL, the Maximum Pressure output in the Results Summary will now be correct. Previously, although the results in the Pressure Results dialog were correct, the maximum pressure calculation was for all base plates analyzed in the current GTSTRUDL session, not the current base plate.
   (GPRF 2012.05)

3. Previously, the units for the specified E value (Young’s modulus) for the bearing surface were shown as “ksi” in the dialog, but the specified value was used as “psi” in the base plate analysis. The generated input file contained a comment noting the units were “psi”, but this would not be obvious to the user without inspecting the input file. The generated input file now uses the specified value x 1000.0 to convert from ksi to psi.
   (GPRF 2012.06)

4. Attachments with Contact Only lines will now be displayed correctly in the Scope Editor when ‘File - Print preview and edit’ is used. Previously all attachment lines including Contact Only lines were drawn with the standard attachment color.
   (No GPRF issued)

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 33. 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)

4.2 General Input/Output

1. 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)

2. 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 for two-dimensional 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. GTMenu is limited to 1,000 views. If more than 1,000 views are created, incorrect displays may occur.

6. GTMenu is limited to 10,000 Member Property Groups. If more than 10,000 property groups are created, incorrect results may occur.

4.4 Offshore Analysis and Design – GTSELOS

1. The GTSELOS linear wave implementation produces incorrect wave kinematics and wave forces at water depths greater than approximately 525 feet (160 meters).
   (No GPRF issued)

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

5.2 Design Prerelease Features

- 5.2.1 LRFD3 Steel Design Code. Rather than use the LRFD3 code which is a prerelease feature, users should use the AISC13 or AISC14 codes which are released features.
- 5.2.2 ACI Concrete Code 318-99. Rather than use the 318-99 code which is a prerelease feature, users should use the 318-05 code which is now a released feature.
- 5.2.3 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)
- 5.2.4 ISO 19902:2007(E) code for fixed steel offshore structure.
- 5.2.5 ASCE4805 Steel Design Code. This code is for the ultimate strength design of steel transmission pole structures.

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

5.2 Design Prerelease Features

5.2.1 LRFD3 Steel Design Code

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. Rather than use the LRFD3 code which is a prerelease feature, users should use the AISC13 or AISC14 codes which are released features.

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.

5.2.2 ACI Code 318-99

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

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

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

Example:

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

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

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

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

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

Notes:

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

2. VU is multiplied by PHISH internally.

3. Calculations for V_c and T_c are modified by replacing $\sqrt{f_c'}$ with (RFSP/6.70) $\sqrt{f_c'}$ as per Section 11.2.1.1.

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

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

5.2.3 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:

```
DESIGN SLAB (REINFORCEMENT) (USING) -

  { WOOD (AND) (ARMER) { → AVERAGE }                }
  {                    {   MAXIMUM }                } (ALONG) -
  { CALCULATE (RESULTANT) (ELEMENT) (FORCES)        }

(CUT { 'a' }) { JOINTS } list1  ELEMENT list2  (TABLE { → ASTM   })-
     { i1  }  { NODES  }                              {   UNESCO }

* { TOP (FACE) (BARS i2) (SPACING v1)      }
  { BOTTOM (FACE) (BARS i3) (SPACING v2)   } -
  { BOTH (FACES) (BARS i4) (SPACING v3)    }

{ → INNER (LAYER) }
{   OUTER (LAYER) } (COVER v4) (LINEAR (TOLERANCE) v5) -

(TORSIONAL (MOMENT) (WARNING) v6)
```

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 5.2.3-1. 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.

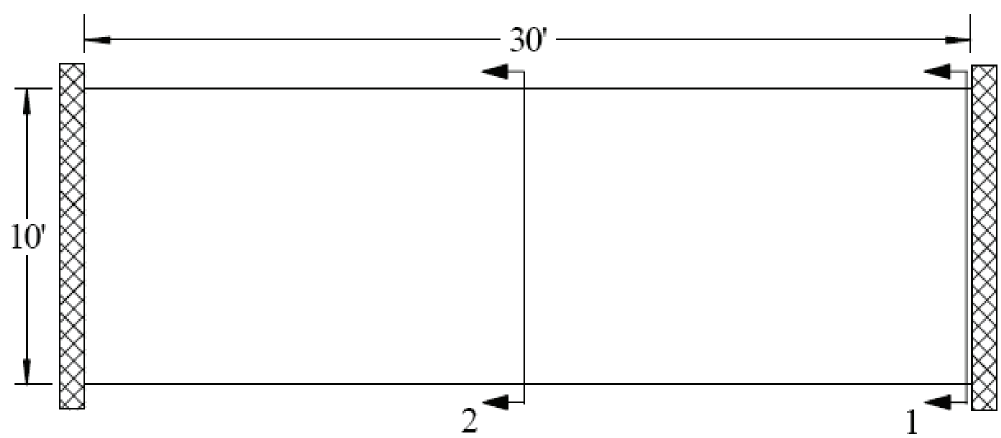


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

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.

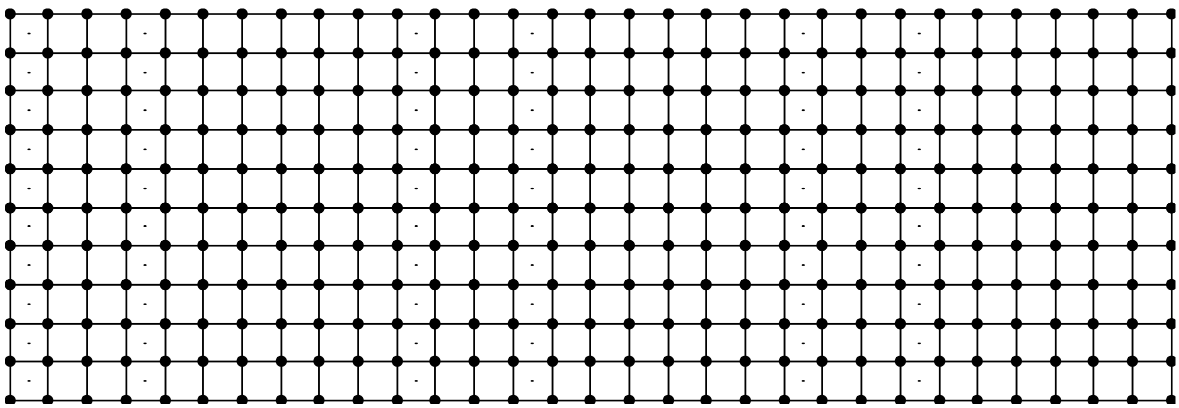


Figure 5.2.3-2 Example Finite Element Model

Definition of Cut Cross-Sections

Two "cuts" are considered for the verification example, as shown in Figure 5.2.3-1.

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
```

5.2.4 ISO 19902:2007 Code for the Design of Fixed Steel Offshore Structures

A new *International Standard ISO 19902:2007(E), First Edition, 2007-12-01, Petroleum and natural gas industries – Fixed steel offshore structures* has been implemented as a prerelease feature. The GTSTRUDL code name for this new offshore code is “19902-07”. The 19902-07 code performs *Basic, Hydrostatic Pressure, and Punching Shear stresses* check according to *International Standard ISO 19902:2007(E)*. This new code, 19902-07, may be used to select or check Circular Hollow Sections (Pipes). The prerelease documentation for the 19902-07 code may be found by selecting the help menu and then Reference Documentation, Reference Manuals, Offshore Loading, Analysis and Design, and 19902-07: ISO 19902:2007(E), First Edition in the GTSTRUDL Output Window.

5.2.5 ASCE4805 Code for the Design of Steel Transmission Pole Structures

The steel design code, ASCE4805, which is based on the 2005 edition of the ASCE/SEI, *Design of Steel Transmission Pole Structures* Specification has been implemented as a pre-release feature. The ASCE/SEI 48-05 Specification is based on ultimate strength methods using factored loads.

The ASCE4805 Code may be used to select or check any of the following shapes:

Design for axial force, bi-axial bending, and torsion:

Pipes

Regular Polygonal Tubes

Structural Tubing

The documentation for the ASCE4805 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and “ASCE4805" in the GTSTRUDL Output Window.

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

```
CALCULATE ERROR (ESTIMATE) (BASED ON)  -

  * { ENERGY (NORM)                                       }
    { MAX DIFFERENCE                                      }
    { DIFFERENCE FROM AVERAGE                             }
    { PERCENT MAX DIFFERENCE                              }  -
    { PERCENT DIFFERENCE FROM AVERAGE                     }
    { NORMALIZED PERCENT MAX DIFFERENCE                   }
    { NORMALIZED PERCENT DIFFERENCE FROM AVERAGE          }


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

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

The L_2 norm is given by:

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

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

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

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

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

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

The relative percentage error which is output for each element is given by:
The nodal error estimates estimate the accuracy of the data in a selected nodal output vector.

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

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
Normalized Percent Maximum Difference

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

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

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 CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

General form:

CALCULATE ECCENTRIC (MEMBER) (BETA) (ANGLES) (WITHOUT - COMMAND (LISTING))

Explanation:

Section 1.10.4 states that the member beta angle (the orientation of the member cross section principal axes) is defined with respect to the joint-to-joint position of the member before member eccentricities are applied. However, in certain structural modeling situations it may be more desirable to be able to specify a beta angle value that is defined with respect to the eccentric position of the member, after member eccentricities are applied. To this end, the CALCULATE ECCENTRIC MEMBER BETA ANGLES command has been implemented in order to provide beta angle information that can be used to construct CONSTANTS commands that specify beta angle values that reflect such a need. When issued, the CALCULATE ECCENTRIC MEMBER BETA ANGLES command produces a report that includes the member name, the member's originally-specified or -computed joint-to-joint beta angle value, and an adjusted joint-to-joint beta angle value that if specified, produces a member orientation and associated analysis behavior as if the original beta angle were defined with respect to the eccentric position of the member. The report also includes a listing of CONSTANTS/BETA commands for all affected members that can be easily copied and pasted into a GTSTRUDL command text file. If this command listing is not desired, it can be eliminated by giving the WITHOUT COMMAND LISTING option. An example of the report is reproduced below:

```
{  657} > CALCULATE ECCENTRIC MEMBER BETA ANGLES

**** WARNING_CHKECCBTA -- The CALCULATE ECCENTRIC MEMBER BETA ANGLES command is
                          a prerelease feature.  User feedback and suggestions
                          are welcome.


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

   PROBLEM - None

   ACTIVE UNITS  FEET KIP  RAD  DEGF SEC
```

```
The following report lists adjusted beta angle values that if specified, produce member
orientations, including corresponding analysis behavior, as if the ORIGINALLY-SPECIFIED
beta angles were defined with respect to the eccentric position of the member.  This
report is for information purposes only.  No computational action is taken.


              Eccentric Member Beta Angle Check Results
              =========================================
   Member            Original Beta Angle      Adjusted Beta Angle
  --------          ---------------------    ---------------------
  11002                        0.06655                  0.09484
  12002                       -0.02815                  0.00884
  11003                       -3.04469                 -3.06850
  13002                        1.26565                  2.52545
  14002                        1.16144                  2.31630
  15002                        1.05723                  2.10572
  16002                        0.95302                  1.89668
  13003                        1.26565                 -0.61557
  14003                        1.16144                 -0.79819
  15003                        1.05723                 -1.03473
  16003                        0.95302                 -1.24443
  17002                       -0.06191                  0.01547
  18002                       -0.44292                 -0.58340
  18003                        3.13987                  3.35983

CONSTANTS/BETA Commands for Adjusted Beta Angles
================================================

UNITS RAD
CONSTANTS
  BETA       0.09484  MEMBER  '11002   '
  BETA       0.00884  MEMBER  '12002   '
  BETA      -3.06850  MEMBER  '11003   '
  BETA       2.52545  MEMBER  '13002   '
  BETA       2.31630  MEMBER  '14002   '
  BETA       2.10572  MEMBER  '15002   '
  BETA       1.89668  MEMBER  '16002   '
  BETA      -0.61557  MEMBER  '13003   '
  BETA      -0.79819  MEMBER  '14003   '
  BETA      -1.03473  MEMBER  '15003   '
  BETA      -1.24443  MEMBER  '16003   '
  BETA       0.01547  MEMBER  '17002   '
  BETA      -0.58340  MEMBER  '18002   '
  BETA       3.35983  MEMBER  '18003   '
```

Note that members are listed only if they are active, they have global eccentricities, and the originally-specified beta angle and the adjusted beta angle differ by more than 1°.

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

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

Elements:

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

r_1, r_2, r_3 = values in current angle units of the load component rotation angles 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:

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

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

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:

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

Explanation:

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

5.4.3 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:

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

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

.
.
.

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

Elements:

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

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

Where,

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

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

End of Document