

GT STRUDL® Version 2017 Release Guide



PROCESS, POWER & MARINE

Release Guide

Release Date: April 2017



Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 2017 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

Copyright © 2017 Intergraph® Corporation. All Rights Reserved. Intergraph is part of **Hexagon**.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

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.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

- a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.
- b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, <http://www.bis.doc.gov/complianceand enforcement/liststocheck.htm>, the U.S. Department of Treasury Specially Designated Nationals List, <http://www.treas.gov/offices/enforcement/ofac/>, and the U.S. Department of State Debarred List, <http://www.pmddtc.state.gov/compliance/debar.html>.
- c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.
- d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Intergraph, the Intergraph logo, and GT STRUDL are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries. Other brands and product names are trademarks of their respective owners.

Table of Contents

Chapter	Page
NOTICES	iii
Table of Contents	v
CHAPTER 1	
Introduction	1-1
CHAPTER 2 NEW FEATURES IN VERSION 2017	
2.1 Steel Design	2-1
2.2 Seismic, Wind and Area Loadings	2-2
2.3 CAD Modeler	2-11
2.4 GTMenu	2-21
2.5 Base Plate	2-30
2.6 GTShell (GT STRUDL Output Window)	2-34
2.7 Steel Tables	2-35
2.8 Finite Elements	2-35
2.9 Reinforced Concrete Design	2-35
CHAPTER 3 ERROR CORRECTIONS	
3.1 CAD Modeler	3-1
3.2 DBX	3-1
3.3 General	3-1
3.4 GTMenu	3-3
3.5 GTShell (GT STRUDL Output Window)	3-5
3.6 Offshore/Fatigue Analysis	3-5
3.7 Reinforced Concrete Design	3-5
3.8 Steel Design	3-6

CHAPTER 4 KNOWN DEFICIENCIES

4.1	CAD Modeler	4-1
4.2	Finite Elements	4-1
4.3	General Input/Output	4-1
4.4	GTMenu	4-2

CHAPTER 5 PRERELEASE FEATURES

5.1	Introduction	5.1-1
5.2	Design Prerelease Features	5.2-1
5.2.1	Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)	5.2-1
5.2.2	ASCE4805 Code for the Design of Steel Transmission Pole Structures	5.2-11
5.3	Analysis Prerelease Features	5.3-1
5.3.1	Calculate Error Estimate Command	5.3-1
5.3.2	The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command	5.3-5
5.4	General Prerelease Features	5.4-1
5.4.1	Rotate Load Command	5.4-1
5.4.2	Reference Coordinate System Command	5.4-5
5.4.2-1	Printing Reference Coordinate System Command ..	5.4-8
5.4.3	GTMenu Point Coordinates and Line Incidences Commands ..	5.4-9
5.4.4	GTMenu Surface Definition Command	5.4-12

This page intentionally left blank.

Chapter 1

Introduction

Version 2017 covers GT STRUDL operating on PC's under the Windows 10 and 7 operating systems. For users who are accustomed to our older version numbering system, the version is internally known as Version 36.0.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2017. In particular, Chapter 2 briefly describes an extensive list of new features including the following new features:

- New Canadian steel design code - S16-14
- Seismic Loadings according to ASCE 7-05 and 7-10
- Numerous improvements to CAD Modeler and GTMenu
- Circular and notched base plate cutouts
- Chinese steel tables

Chapter 3 provides you with details regarding error corrections that have been made since the Version 2017 release. Chapter 4 describes known problems with Version 2017. 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 GT STRUDL 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 section numbers in Chapter 5 are shown below:

- 5.2 Design Prerelease Features
 - 5.2.1 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)
 - 5.2.2 ASCE4805 Steel Design Code. This code is for the ultimate strength design of steel transmission pole structures.
- 5.3 Analysis Prerelease Features
 - 5.3.1 Calculate Error Estimate Command
 - 5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

5.4 General Prerelease Features

5.4.1 Rotate Load Command

5.4.2 Reference Coordinate System Command

5.4.3 GTMenu Point Coordinates and Line Incidences Commands

5.4.4 GTMenu Surface Definition Command

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

Chapter 2

New Features in Version 2017

This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GT STRUDL in Version 2017. This release guide is also available online upon execution of GT STRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 Steel Design

1. A new Canadian Standards Association S16-14 design code (adopted in 2014) has been implemented. The GT STRUDL code name is called CSA-2014 which is primarily based on the *S16-14, Design of steel structures, Canadian Standards Association*, June 2014. This new code, CSA-2014, may be used to select or check any of the following shapes:

- I shapes
- Channels
- Rectangular Hollow Sections (Tubes)
- Circular Hollow Sections (Pipes)
- Tees
- Double Angles
- Solid Rectangular Bars
- Solid Round Bars

To use the CSA-2014 code, you specify the Code Parameter as shown below:

Parameters
Code CSA-2014 All

Additional parameters and full documentation for the CSA-2014 code may be found upon execution of GT STRUDL by selecting Help-Reference Documentation-Reference Manuals-Steel Design- CSA-2014.

2.2 Seismic, Wind and Area Loadings

1. A new SEISMIC LOAD command is now available for the purpose of creating a GT STRUDL independent load condition consisting of applied joint loads that represent equivalent static seismic forces computed according to the ASCE 7-05 and 7-10 standards. The SEISMIC LOAD command and the SEISMIC LOAD main shell dialog are shown below:

$$\text{SEISMIC LOAD } \left\{ \begin{matrix} i_{\text{SL}} \\ a_{\text{SL}} \end{matrix} \right\} ('descr_{\text{SL}}') \text{ standard}$$

General Structure Data

Seismic Ground Motion Data

Seismic Force Data

END (OF) (SEISMIC) (LOAD) (DATA)

$$\text{standard} = \left\{ \begin{matrix} \underline{\text{ASCE7-05}} \\ \underline{\text{ASCE7-10}} \end{matrix} \right\}$$

The General Structure, Seismic Ground Motion and Seismic Force Data are shown on the following pages.

general structure data =

$$\left\{ \begin{array}{l} \text{HEIGHT (AXIS)} \left\{ \begin{array}{l} \underline{Y} \\ \underline{Z} \end{array} \right\} \\ \text{DIRECTION} \left\{ \begin{array}{l} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{array} \right\} \\ \text{SEISMIC WEIGHT LOAD} \left\{ \begin{array}{l} i_{sw} \\ a_{sw} \end{array} \right\} \\ \text{STORY (HEIGHTS)} \left\{ \begin{array}{l} \text{list of heights} \\ \text{JOINTS list of joints} \end{array} \right\} (\text{TOLERANCE } v_{FTOL}) \\ (\text{FLOOR TOLERANCE } v_{FTOL}) \end{array} \right\} *$$

seismic ground motion data =

$$\left\{ \begin{array}{l} \text{SITE CLASS} \left\{ \begin{array}{l} \underline{A} \\ \underline{B} \\ \underline{C} \\ \underline{D} \\ \underline{E} \\ \underline{F} \end{array} \right\} \\ \text{OCCUPANCY (CATEGORY)} \left\{ \begin{array}{l} \underline{I} \\ \underline{II} \\ \underline{III} \\ \underline{IV} \end{array} \right\} \\ \text{SPECTRAL (ACCELERATION) } \textit{spectral acceleration data} \\ \text{LONG PERIOD (TL)} v_{TL} \\ \underline{CS} v_{CS} \quad \underline{SDS} v_{SDS2} \end{array} \right\} *$$

spectral acceleration data =

$$\left\{ \begin{array}{l} \text{MAP} \left\{ \begin{array}{l} \text{USA} \\ \text{ALASKA} \\ \text{HAWAII} \\ \text{VIRGIN (ISLANDS)} \\ \text{PUERTO (RICO)} \end{array} \right\} - \\ \text{LATITUDE } v_{\text{LAT}} \quad \text{LONGITUDE } v_{\text{LONG}} \\ \text{SS } v_{\text{SS}} \quad \text{S1 } v_{\text{S1,1}} \\ \text{SDS } v_{\text{SDS1}} \quad \text{SD1 } v_{\text{SD1}} \quad \text{S1 } v_{\text{S1,2}} \end{array} \right\}$$

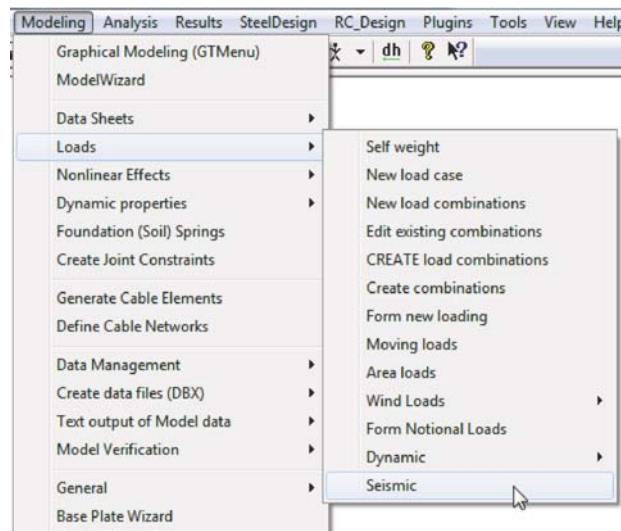
seismic force data =

$$\left\{ \begin{array}{l} \text{FUNDAMENTAL PERIOD} \left\{ \begin{array}{l} \text{ASCE 7 - 05 struc type} \\ \text{ASCE 7 - 10 struc type} \end{array} \right\} (\text{ANALYSIS MODE } i_{\text{M}}) \\ \text{TA } v_{\text{TA}} \\ (\text{RESPONSE MODIFICATION (COEFFICIENT)}) \text{ R } v_{\text{R}} \\ (\text{ACCIDENTAL (TORSION) (MOMENT) (FACTOR)}) \text{ ATMF } v_{\text{ATMF}} \\ (\text{VERTICAL (SEISMIC) (FACTOR)}) \text{ VSF } v_{\text{VSF}} \end{array} \right\}$$

$$\text{ASCE 7-05 struc type} = \left\{ \begin{array}{l} \text{SMR} \\ \text{CMR} \\ \text{EBS} \\ \text{CSMRLOW} \\ \text{MCSW} \\ \text{OTHER} \end{array} \right\}$$

$$ASCE\ 7-10\ struc\ type = \left\{ \begin{array}{l} \underline{SMR} \\ \underline{CMR} \\ \underline{SEB} \\ \underline{SBRB} \\ \underline{CSMRLOW} \\ \underline{MCSW} \\ \underline{OTHER} \end{array} \right\}$$

Dialogs are available in GTShell to create Seismic Loads as shown below:



Seismic Load Generation

Load X Name: SLX
Description: Seismic Load Global X

Standard:
☐ ASCE 7-05
☒ ASCE 7-10

General Data:
Height Axis: ☒ Y ☐ Z
Direction: ☒ X ☐ Z ☐ Both
Seismic Weight Load: DLY

Story Heights:
Story heights from: ☐ Values ☒ Joints Tolerance: 2.0000 Feet
EXISTING 37 57 77 97 117

☐ Specified Response Coefficient (Cs) SDS

☒ Response Coefficient from Standard

Ground Motion -- Spectral Response Accelerations:
Site Class: B: Rock Risk Category: IV: Essential Facilities
Long Period (TL): 12.0 Seconds
☒ SS, S1 from Map: USA Latitude: 36.0 Longitude: -89.0 Degrees
☐ SS S1 g
☐ SDS SD1 S1 g

Seismic Forces:
☒ Fundamental Period from Standard: SMR: Steel Moment-Resisting
Analysis Mode, Direction: X Y Z 1 to 0
☐ Specified Fundamental Period (Ta) Seconds
Response Modification Coefficient (R): 3.5
Accidental Torsional Factor (ATMF): 1.00 Vertical Seismic Factor (VSF): 1.00

OK Cancel Help

An example of the Seismic Load command is shown below:

SEISMIC LOAD 'SLX' 'Seismic Load Global X' STANDARD ASCE7-10

HEIGHT AXIS Y

DIRECTION X

SEISMIC WEIGHT LOAD 'DLY'

STORY HEIGHTS JOINTS EXISTING 37 57 77 97 117

FLOOR TOLERANCE 2.0000

SITE CLASS B

RISK CATEGORY IV

LONG PERIOD TL 12.0

SPECTRAL ACCELERATION MAP USA LATITUDE 37.0 -

LONGITUDE -89.0

FUNDAMENTAL PERIOD SMR

RESPONSE MODIFICATION COEFFICIENT R 3.5

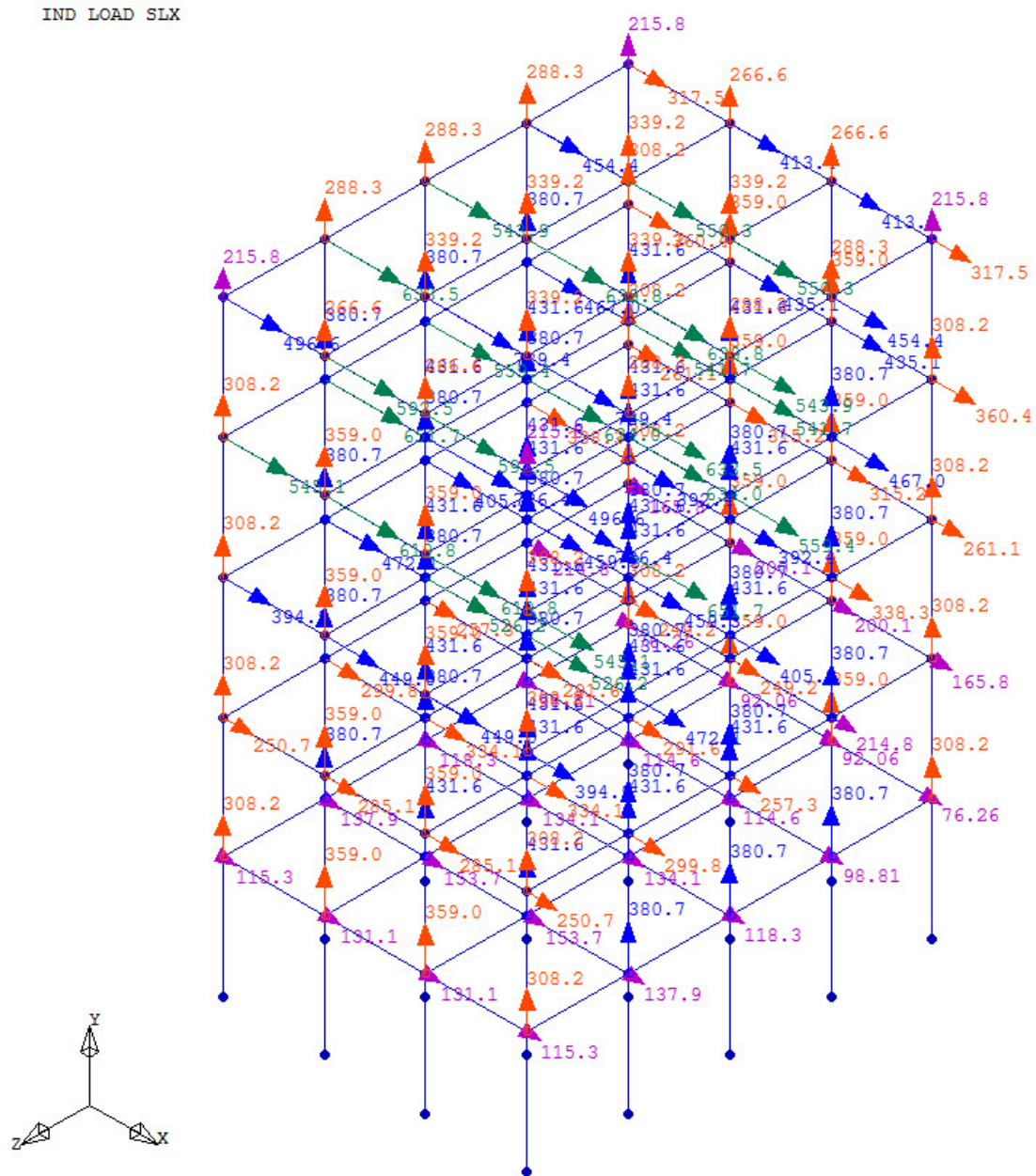
ACCIDENTAL TORSION FACTOR ATMF 1.00

VERTICAL SEISMIC FACTOR VSF 1.00

END OF SEISMIC LOAD DATA

Note that the seismic load is applied in the x-direction using the seismic weight load DLY and that an accidental torsion factor is applied as well as a vertical seismic factor.

The seismic loads created using these commands are shown in the figure on the next page:



2. In GT STRUDL version 2016 R2, the WIND LOAD command issues a warning message for every member for which a wind force projected area equal to 0.0 is detected. Version 2017 has been improved whereby only one such warning message is now issued, followed by a list of all members for which this condition is detected.
3. The WIND LOAD command has been modified to include a new general data option that provides for the optional specification of a minimum value of the velocity pressure q_z that is used for the computation of the design wind force. The specified minimum value of q_z overrides the prescribed value of 10 psf under ASCE 7-05 and 16 psf under ASCE 7-10. The new option command is MINIMUM VELOCITY PRESSURE QZMIN *value*, which is illustrated in the example WIND LOAD command below:

```

WIND LOAD 'WL3'
  UNITS FT KIPS DEG SEC StdMASS
  STANDARD ASCE7-10
  ELEVATION AXIS Y
  EXPOSURE CATEGORY B
  SPEED MPH 100.0000
  DIRECTION ANGLE -90.0000
  DIRECTIONALITY FACTOR KD 0.85
  IMPORTANCE FACTOR I 1.00
  TOPOGRAPHIC FACTOR KZT 1.00
  MINIMUM VELOCITY PRESSURE QZMIN 20.0
  GUST FACTOR G 0.85
  GROSS AREA AG 100.00
  ADDED FORCE AREA AFADD 0.0
  UNITS FT KIPS DEG SEC StdMASS
  MEMBERS EXISTING 1 7 TO 15 TYPE LATTICE FRAMEWORK -

      AREA AND FORCE AF 10.0000 -
      KZ 0.0000 CF 1.7000 QZ 0.0000 FLD 1.0000
END WIND LOAD DATA

```

See Section 2.1.11.3.9, GT STRUDL Reference Manual Volume 1.

4. The AREA LOAD Command has been improved to allow the use of a Joint to define the Elevation Value as shown below:

$$\underline{\text{ELEVATION}} \left\{ \begin{array}{c} v_e \\ \underline{\text{JOINT}} \text{ jld} \end{array} \right\} \dots$$

See Section 2.1.11.3.7.1 of the GT STRUDL Reference Manual Volume 1 for further information.

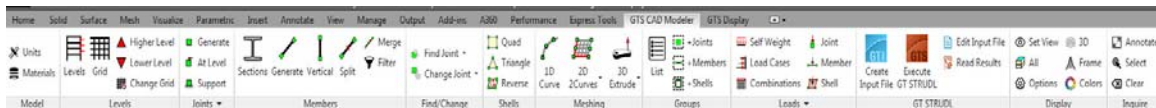
5. GT STRUDL now saves the information used to define Area Loads. This information can be printed at any time by using the new PRINT AREA LOAD PARAMETERS Command which is described in Section 2.1.11.3.7.2 of the GT STRUDL Reference Manual Volume 1.

Also, when the input file is generated in the Shell using Create a New Text File or by selecting Generate GT STRUDL Text input in GTMenu, the Area Loads are defined by the parameters used to create the loading rather than the applied members loads as shown in the following excerpt from a generated file:

```
AREA LOAD 1      DIRECTION Y PLANE TOLERANCE 0.167000  
                ELEVATION 12.000000 VALUE 0.100000 TWO WAY  
END AREA LOAD
```

2.3 CAD Modeler

1. CAD Modeler now supports BricsCAD Version 17 Pro and Platinum editions.
2. The GTS CAD Modeler Ribbon Bar has been improved and reorganized with the items in each panel aligned. The new 2017 GTS CAD Modeler Ribbon Bar is shown below followed by the one from Version 2016 R2:

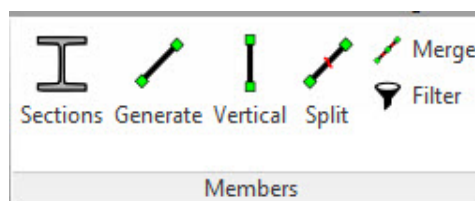


Version 2017 GTS CAD Modeler Ribbon Bar

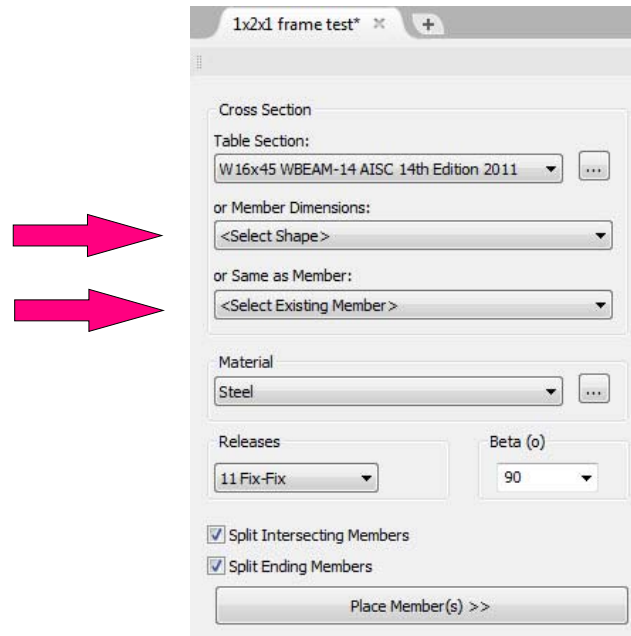


Version 2016 R2 GTS CAD Modeler Ribbon Bar

3. The following changes have been made to the GTS CAD Modeler Ribbon Bar and items in the Ribbon Bar:
 - a. The creation of Sections has been moved from the Model Panel to the Members panel as shown in the figure below:

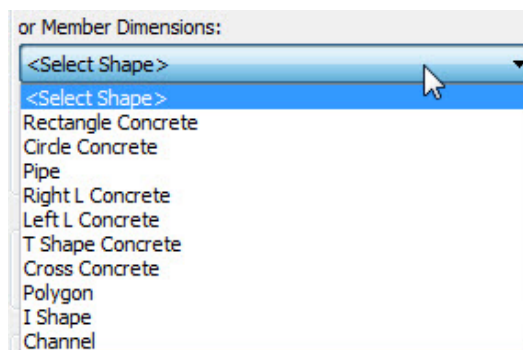


- b. The placement of members dialog that pops-up when you select the Generate or Vertical option has changed as shown below:

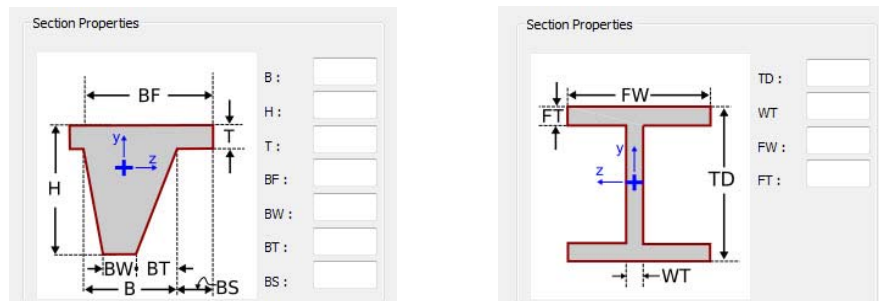


The workflow to place members has changed. Now, the dialog shown above appears when you select the Generate or Vertical option in the Members panel. Previously, you had to select a joint or grid point before the dialog would appear.

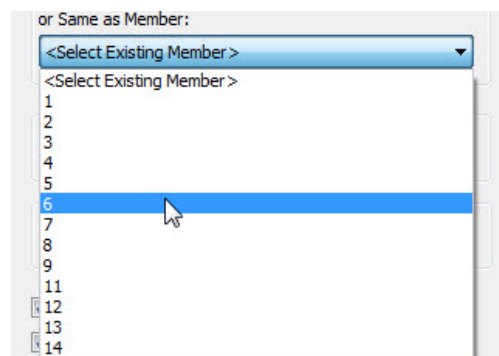
In addition, two new options indicated by the arrows placed to the left of the dialog shown above have been added to this dialog. You may now specify Member Dimension properties for the following shapes:



When you select one of the shapes from the Member Dimensions pulldown, the Place Members dialog changes at the bottom to show a figure of the shape with symbols indicating the dimensions that you must enter in the text boxes to define the cross section. Examples of the figures and text boxes are shown below for a T shape concrete section and a I Shape:



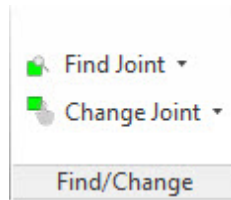
Now, you may also specify the properties for the members to be placed by choosing to use the properties that are the Same as an existing member in the structure. An example of this option on the dialog is shown below:



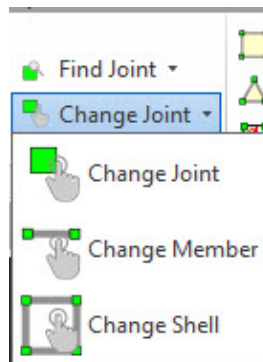
After selecting either an existing Table section, or by defining a cross section by specifying the member dimensions or choosing the cross section to be the same as an existing member, select the Place Members button. Then, begin placing members by clicking the two joints or grid position or by specifying the joint coordinates to define the start and end of the member.

Should you decide to change any item in the dialog, you may do so and then immediately resume selecting the joints or grid or specifying the coordinates to place the members. The dialog remains visible until you right click the mouse or hit the Esc key on the keyboard.

- c. The Find and Change options for Joints, Members and Shells have been moved to a new panel as shown below:

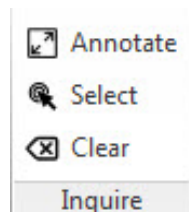


By clicking on the down arrow beside Find and Change, you may also Find or Change Members or Shells as shown in the pulldown below:

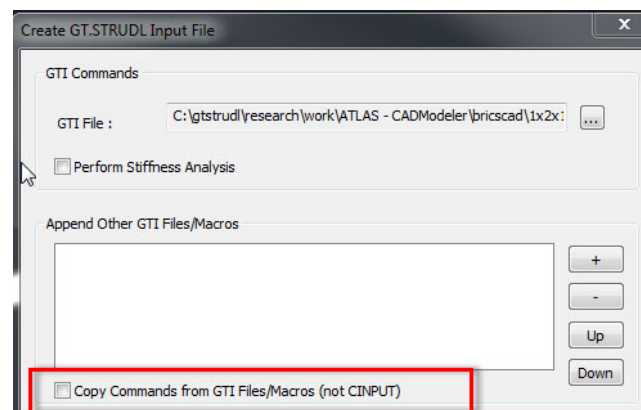


In previous versions, the Find and Change options appeared in the Joint, Member and Shell panels.

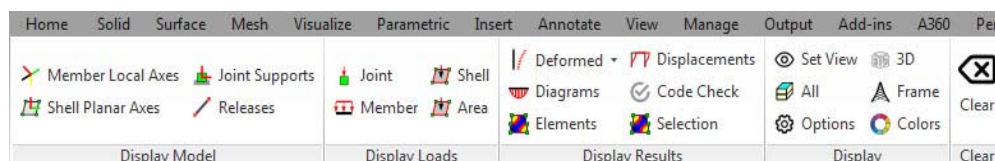
- d. The Clear feature which will clear the display of loads, supports, results,...,etc has been added to the Inquire panel. Previously, you had to go the GTS Display Ribbon Bar or use a command to use the Clear option. The modified Inquire panel is shown below:



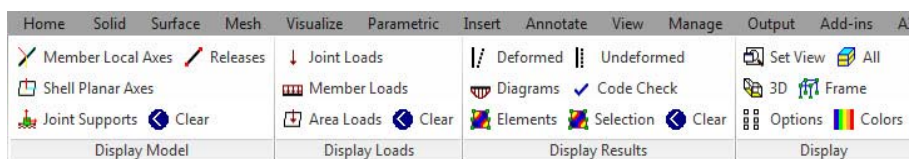
- e. A new option has been added to the Create Input File dialog which will allow you to copy the contents of another input file or macro to the file currently being created. This allows you to have all of the commands in one file. The new option is highlighted in the figure on the next page:



4. The GTS Display Ribbon Bar has been improved and reorganized with the items in each panel aligned. The new 2017 GTS Display Ribbon Bar is shown below followed by the one from Version 2016 R2:

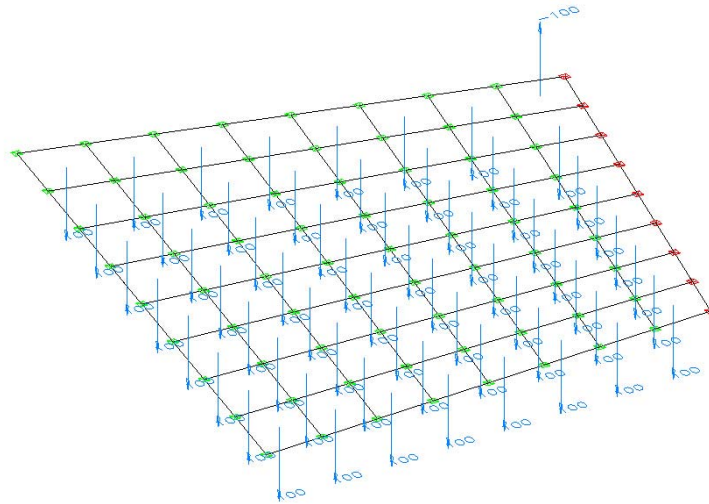


Version 2017 GTS Display Ribbon Bar



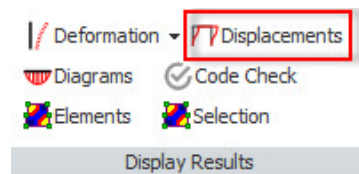
Version 2016 R2 GTS Display Ribbon Bar

5. The following changes have been made to the GTS Display Ribbon Bar and items in the Ribbon Bar:
- The Clear option has been placed in a separate Clear panel on the right side of the Ribbon Bar. Previously, a Clear item was in each panel of the Ribbon Bar.
 - A new Shell option has been added to the Display Loads panel enabling the display of shell element loads. An example of the display of shell element surface forces is shown on the next page:

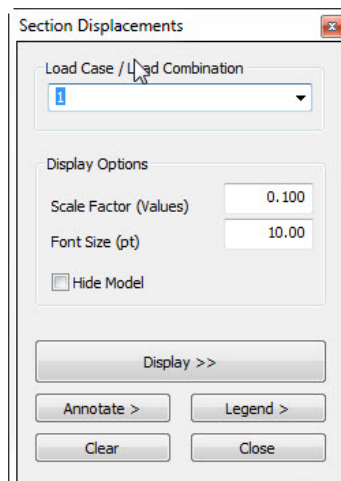


Skewed Plate with Shell Surface Forces Displayed

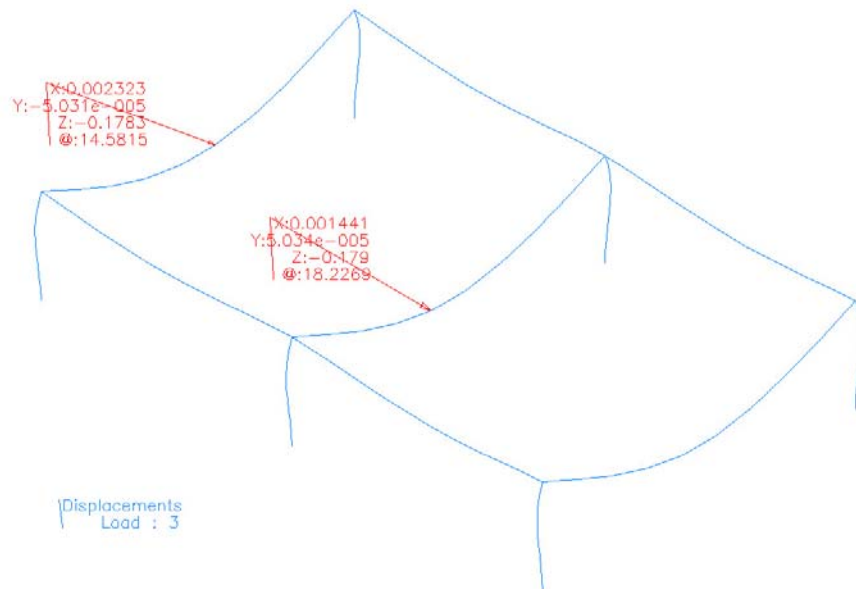
- c. The Display Results panel has an additional selection to allow you to display Section Displacements as highlighted below:



The new Section Displacements dialog which will draw the deformed shape of the members is shown below:

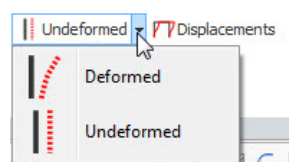


Using this new dialog, you can specify the loading and scale factor and also choose to hide the original model. Once you have displayed the deformed shape, you may also Annotate the displacements on the structure by picking on a position on the deformed shape and then selecting a location to annotate the global deformations. You may also place a Legend on the screen. An illustration of annotating the deformed shape is shown in the figure below:

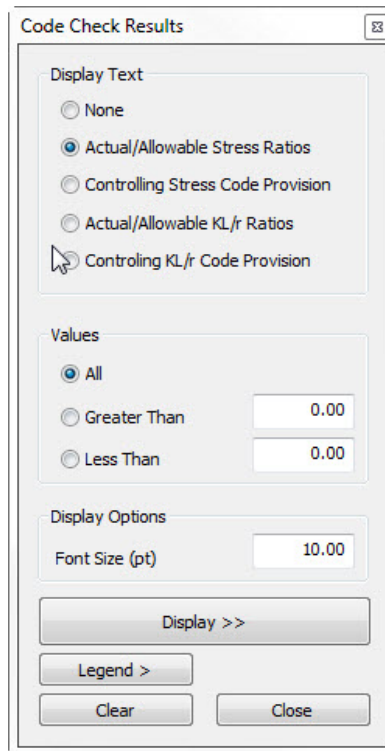


Frame with Section Displacements Annotated

The Deformation option on the Display Results panel will allow you to switch between the Deformed structure showing only joint displacements with a straight line drawn between the joints and the Undeformed structure as shown below:



- d. The display of Code Check Results now displays not only the Pass/Fail status of the selected members but also Actual/Allowable Stress and KL/r Ratios as well as the Provisions. You can also display values greater than or less than a value that you specify. The new Code Check Results dialog is shown below:

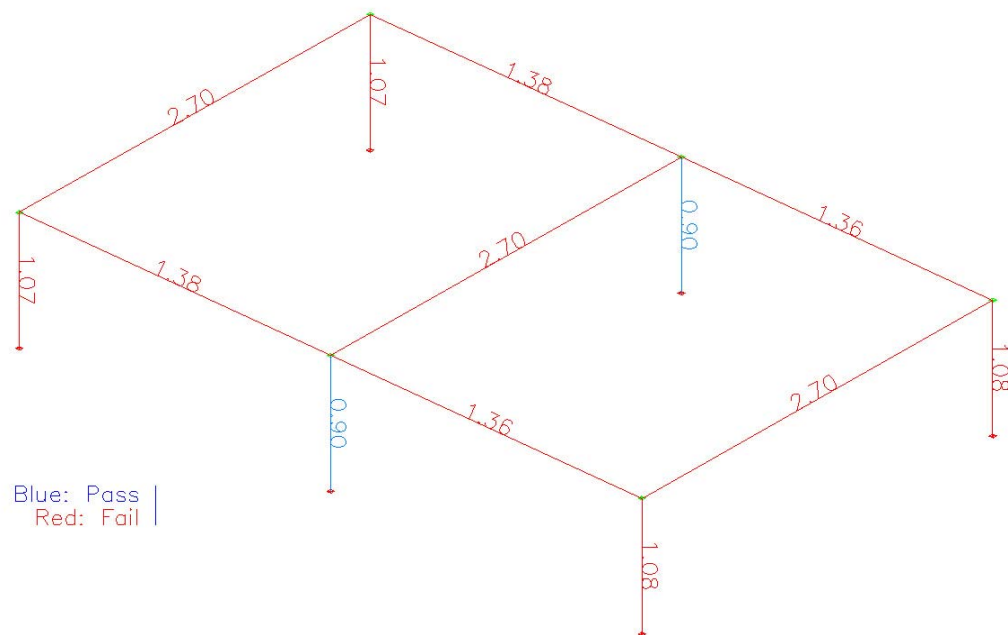


The image shows a dialog box titled "Code Check Results". It contains three main sections: "Display Text", "Values", and "Display Options".

- Display Text:** A group box containing five radio buttons:
 - ☐ None
 - ☒ Actual/Allowable Stress Ratios
 - ☐ Controlling Stress Code Provision
 - ☐ Actual/Allowable KL/r Ratios
 - ☐ Controlling KL/r Code Provision
- Values:** A group box containing three radio buttons and two text input fields:
 - ☒ All
 - ☐ Greater Than
 - ☐ Less Than
- Display Options:** A group box containing a text input field:
 - Font Size (pt)

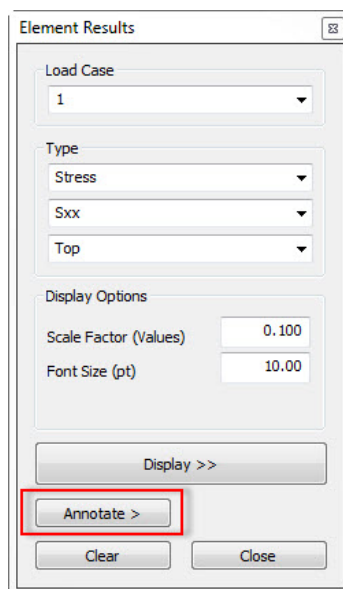
At the bottom of the dialog are four buttons: "Display >>", "Legend >", "Clear", and "Close".

The figure shown on the next page demonstrates the display of the Actual/Allowable Stress Ratios on the members with blue indicating the members that passed the code check and red indicating the members that failed the code check:



Frame with Actual/Allowable Code Check Values Displayed

- e. The Element Results dialog now has an Annotate button which allows you to annotate the values of the contour results at the joints. The modified dialog is shown below with the Annotate button highlighted:



An example of the Annotation of the stress contours is shown below:

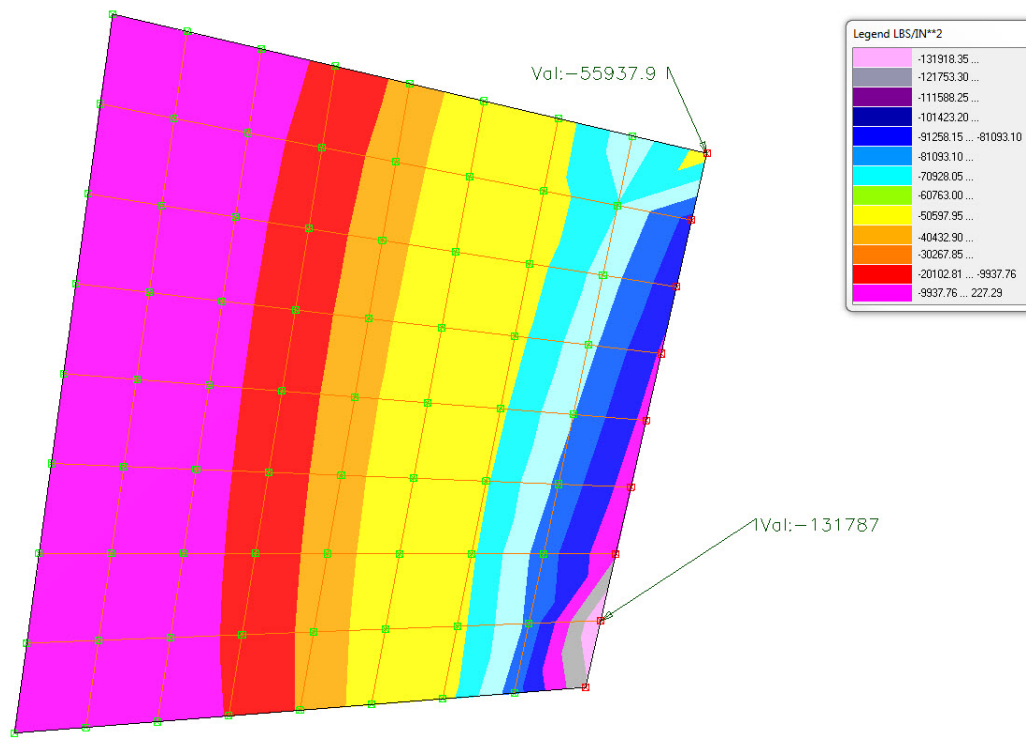
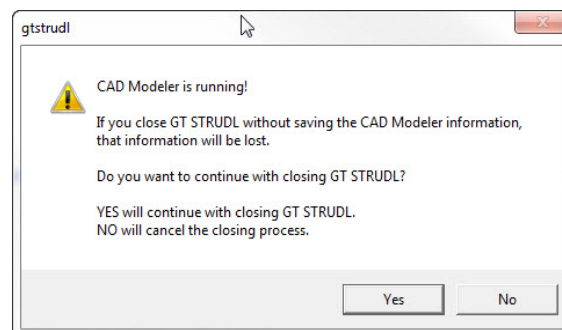


Plate with Contours values labeled

6. Joint loads and shell loads are now displayed as they are applied.
7. New Chinese steel tables have been added to CAD Modeler. These tables are described in Section 2.7.
8. If you close your GT STRUDL session while CAD Modeler is running, the following pop-up will appear informing you that you must save your CAD Modeler information or it will be lost as CAD Modeler requires GT STRUDL to be running:



2.4 GTMenu

1. Numerous improvements have been made to the Editing dialogs for Joints, Members and Elements. Now when editing a joint, member or element, the existing data will be shown in the dialog which will facilitate your changes and help reduce errors made when editing model data.

Shown below is an example of the dialogs that will pop-up when editing Joint Support & Release Data for Version 2017 and Version 2016R2:

Current selected Support & Release are:

D.O.F.	Restrained	Elastic Supports
FX:	☒	KFX: 900
FY:	☒	KFY: 900
FZ:	☒	KFZ: 9000
MX:	☐	KMX: 0
MY:	☐	KMY: 0
MZ:	☐	KMZ: 0

Support Rotations

TH1: 0 TH2: 0 TH3: 0

Cancel Done

Version 2017
with existing joint data shown

D.O.F.	Restrained	Elastic Supports
FX:	☒	KFX: 0
FY:	☒	KFY: 0
FZ:	☒	KFZ: 0
MX:	☒	KMX: 0
MY:	☒	KMY: 0
MZ:	☒	KMZ: 0

Support Rotations

TH1: 0 TH2: 0 TH3: 0

Cancel Done

Version 2016 R2
with no existing data shown

In the Version 2017 dialog above, the existing data for the Restraints, Elastic Supports and Support Rotations is shown. Also, if multiple joints are selected with the same data, the existing data will be shown and the top of the dialog will indicate that Joint Releases are being edited for multiple joints.

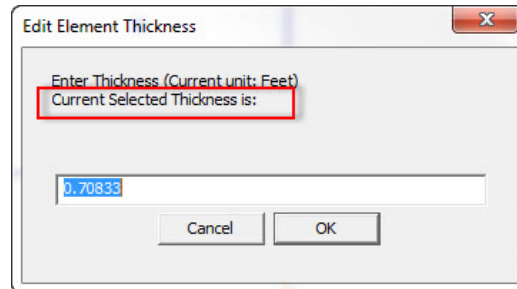
If multiple joints are selected with different data, the pop-up dialog will indicate that all DOF are restrained and the Elastic Supports will all be zero and the top of the dialog will again indicate that Multiple Joint Releases are being edited and that Multiple joints with different data are selected as shown on the next page:

Shown below is an example of the dialog that will now pop-up when you Edit Member Releases with the existing member releases highlighted:

Also, if multiple members are selected with the same data, the existing data will be shown and the top of the dialog will indicate that Member Releases are being edited for multiple members.

If multiple members are selected with different data, the pop-up dialog will indicate that all DOF are released and the Elastic Connections will all be zero and the top of the dialog will again indicate that Multiple Member Releases with different data are selected.

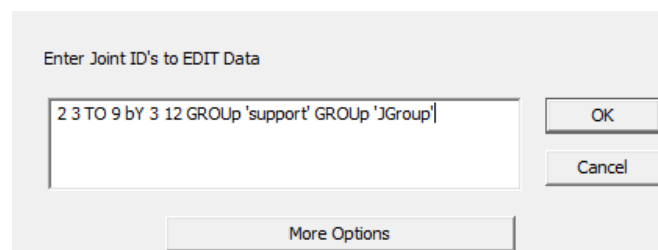
Shown below is the dialog that will now pop-up when you Edit Element Thickness data:



If multiple elements are selected with the same data, the existing data will be shown and the top of the dialog will indicate that you are editing the thickness for multiple elements.

If multiple elements are selected with different data, the pop-up dialog will indicate that the thickness is zero and the top of the dialog will again indicate that you are editing the thickness for multiple elements with different data.

2. In many instances when entering or editing data for your model, it is easier to specify a list of joints, members or elements using the List option from the Mode Bar especially if you have created Groups. In Version 2017, the List option on the Mode Bar has undergone the following significant improvements:
 - a. The input box to enter the list has been increased from 72 to 1080 characters to allow for longer lists for joint, member and element ID's. An example of the expanded List mode input box for Joint ID's is shown below:



- b. If you select the More Options button in the dialog shown above, you now have an option to see the available joints , members or element from which you can make selections to build the list. Furthermore, the

applicable Groups that you can choose from are also displayed in the dialog boxes. As you make selections, the list in the input box is updated dynamically.

An example of the expanded input box with the selected joints and groups to build the list is shown below:

Enter Joint ID's to EDIT Data

2 3 TO 9 bY 3 12 GROUp 'support' GROUp 'JGroup'

OK

Cancel

Less Options

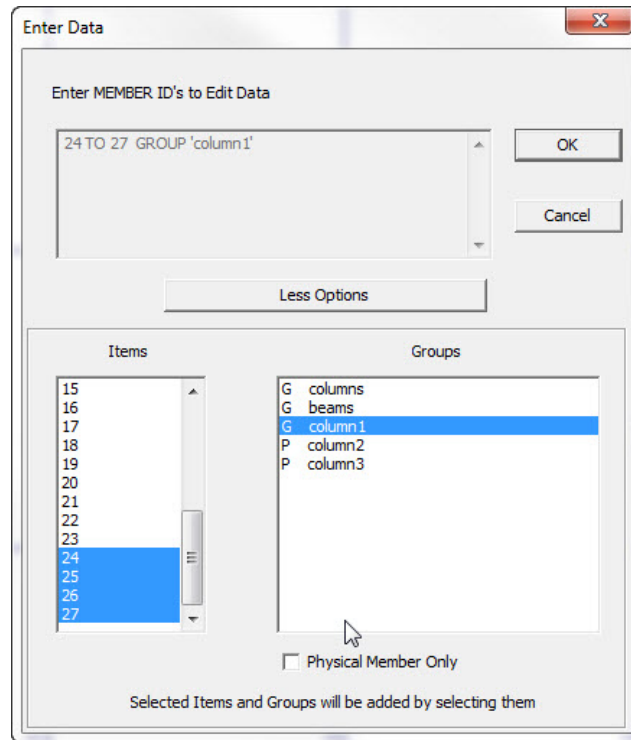
Items	Groups
1	support
2	JGroup
3	JointGro
4	CornerJo
5	
6	
7	
8	
9	
10	
11	
12	

Selected Items and Groups will be added by selecting them

As you select from the list of Items and Groups indicated in blue in the figure above, the list box is automatically updated with the selections.

- c. When building a list of members using the More Options feature of the new List dialog, the Group list will indicate which groups are General groups designated by a G to the left of the group name and which groups are Physical Member Groups designated by a P to the left of the group name. In addition, you have the option to see only Physical Member groups.

An example of the More Options feature when editing members, is shown on the next page. Note that some of the groups have a G (General) to the left of the group name and some have a P (Physical).



Again, as you select from the list of Items and Groups indicated in blue in the figure above, the list box is filled in with your selections.

The new List Mode dialog can be used with the following dialogs:

Create Loads (Joints, Members, Elements)

Edit

Joint Data Dialog

Member Data Dialog

Element Data Dialog

Loads (Joints, Members, Elements)

Refine Finite Elements Mesh

Split Members

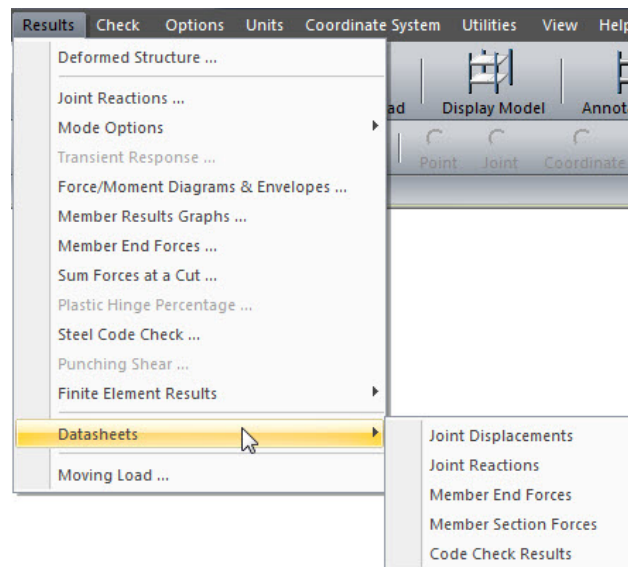
ID's

Move Model

Copy Model

Label Settings (List columns for Display and Label IDs for Joints, Members, and Elements)

3. New Datasheets options have been added to the Results pulldown which will bring up datasheets for Joint Displacements, Joint Reactions, Member End Forces, Member Section Forces and Code Check Results. The modified Results pulldown illustrating the new Datasheets option is shown in the figure below:



Shown below are examples of the Joint Displacement and Joint Reaction datasheets:

GTSTRUDL - Joint Displacement Datasheet, display Units: Inches Degrees

Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y	Rotation Z
1	1	0.0000	0.0000	0.0000	0.0034079	-0.0015114	0.0020799
1	2	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
1	3	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
1	4	0.0000	0.0000	0.0000	-1.1145505	0.0007505	-0.5897114
1	5	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
1	6	0.0000	0.0000	0.0000	-0.0005827	0.3264865	-0.0108161
1	7	0.0000	0.0000	0.0000	-0.0008427	-0.0000371	0.0002239
1	8	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
2	1	-0.0071	-0.0120	-0.0411	0.0033415	-0.0021526	0.0005768
2	2	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
2	3	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
2	4	-0.0003	0.0166	0.0014	1.1073195	-0.0008468	-0.2772933
2	5	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
2	6	0.0078	-0.0005	0.0008	0.0009584	-0.3216992	-0.0143220
2	7	0.0001	0.0030	0.0003	-0.0008210	0.0001262	-0.0006131
2	8	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
3	1	0.0000	0.0000	0.0000	0.0005736	0.0000000	0.0000000
3	2	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
3	3	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
3	4	0.0000	0.0000	0.0000	-1.3424299	0.0000000	0.0000000
3	5	0.0000	0.0000	0.0000	0.0000000	0.0000000	0.0000000
3	6	0.0000	0.0000	0.0000	0.0010058	0.0000000	0.0000000

GTSTRUDL - Joint Reactions Datasheet, display Units: Feet Kips

Joint	Load	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z
1	1	6.498	0.000	45.115	-0.000	-0.000	0.000
1	2	0.000	0.000	0.000	0.000	0.000	0.000
1	3	0.000	0.000	0.000	0.000	0.000	0.000
1	4	0.069	-2.978	-1.346	-0.000	0.000	0.000
1	5	0.000	0.000	0.000	0.000	0.000	0.000
1	6	-5.367	-0.000	-1.572	-0.000	0.000	0.000
1	7	-0.008	-0.000	-0.266	-0.000	-0.000	0.000
1	8	0.000	0.000	0.000	0.000	0.000	0.000
3	1	-1.942	-0.000	47.870	0.000	0.128	-0.001
3	2	0.000	0.000	0.000	0.000	0.000	0.000
3	3	0.000	0.000	0.000	0.000	0.000	0.000
3	4	0.073	-4.248	-0.250	0.000	0.044	-0.322
3	5	0.000	0.000	0.000	0.000	0.000	0.000
3	6	-2.225	0.000	0.295	0.000	-0.793	-0.005
3	7	0.014	0.000	-0.020	0.000	-0.001	-0.000
3	8	0.000	0.000	0.000	0.000	0.000	0.000
5	1	-3.597	0.000	8.890	0.000	0.000	-0.000
5	2	0.000	0.000	0.000	0.000	0.000	0.000
5	3	0.000	0.000	0.000	0.000	0.000	0.000
5	4	0.070	-2.973	-1.431	-0.000	-0.000	0.000
5	5	0.000	0.000	0.000	0.000	0.000	0.000
5	6	-1.154	-0.000	1.302	0.000	0.000	0.000
5	7	-0.008	0.000	0.002	0.000	0.000	-0.000
5	8	0.000	0.000	0.000	0.000	0.000	0.000
6	1	-0.000	0.000	0.000	-0.000	0.014	0.000
6	2	0.000	0.000	0.000	0.000	0.000	0.000
6	3	0.000	0.000	0.000	0.000	0.000	0.000
6	4	0.000	0.000	0.000	0.000	0.003	0.031
6	5	0.000	0.000	0.000	0.000	0.000	0.000
6	6	0.000	-0.000	0.000	0.000	-0.027	-0.000

Examples of the Member End Forces and Section Force datasheets are shown below and on the next page:

GTSTRUDL - Member End Force Datasheet, display Units: Feet Kips

Member	Load	Joint	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z
1	1	1	41.3757	0.0018	0.0000	0.0000	0.0001	0.0259
1	1	2	-41.3757	-0.0018	-0.0000	-0.0000	-0.0004	0.0040
1	2	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	2	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	3	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	3	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	4	1	-1.3947	0.0004	-2.9778	-0.0086	0.0458	0.0306
1	4	2	1.3947	-0.0004	-2.9722	-0.0086	0.0012	-0.0241
1	5	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	5	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	6	1	-0.7557	-4.2477	-0.0001	0.0001	-0.0046	-0.5253
1	6	2	0.7557	-4.2523	0.0001	-0.0001	0.0071	0.9647
1	7	1	-0.2610	0.0003	-0.0000	0.0000	-0.0000	0.0001
1	7	2	0.2610	-0.0003	0.0000	-0.0000	0.0001	0.0057
1	8	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	8	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	1	3	44.1469	0.0079	-0.0000	0.0000	0.0006	0.1051
2	1	4	-44.1469	-0.0079	0.0000	-0.0000	0.0000	0.0299
2	2	3	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	2	4	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	3	3	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	3	4	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	4	3	-0.0015	0.0028	-4.2592	-0.0314	0.2454	0.0198
2	4	4	0.0015	-0.0028	-4.2408	0.0314	-0.0897	0.0270
2	5	3	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	5	4	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	6	3	0.0316	-0.0462	-0.0002	-0.0004	-0.0013	-0.3309
2	6	4	-0.0316	0.0462	0.0002	0.0004	0.0041	-0.4538
2	7	3	-0.0003	-0.0001	-0.0000	-0.0000	0.0002	-0.0008
2	7	4	0.0003	0.0001	0.0000	0.0000	0.0001	-0.0001
2	8	3	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	8	4	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
8	1	16	4.8937	-0.0002	-0.2122	0.0032	0.0041	-0.0043
8	1	16	-4.8937	0.0002	0.2122	-0.0032	8.6039	0.0006
8	2	15	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
8	2	16	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000

GTSTRUDL - Section Force Datasheet, display Units: Feet Kips

Member	Load	Section	Axial	Y shear	Z shear	Torsion	Y bending	Z bending
1	1	0.000	-41.3757	-0.0018	-0.0000	-0.0000	-0.0001	-0.0259
1	1	0.250	-41.3757	-0.0018	-0.0000	-0.0000	-0.0002	-0.0185
1	1	0.500	-41.3757	-0.0018	-0.0000	-0.0000	-0.0003	-0.0110
1	1	0.750	-41.3757	-0.0018	-0.0000	-0.0000	-0.0003	-0.0035
1	1	1.000	-41.3757	-0.0018	-0.0000	-0.0000	-0.0004	0.0040
1	2	0.000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	2	0.250	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	2	0.500	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	2	0.750	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	2	1.000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	3	0.000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	3	0.250	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	3	0.500	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	3	0.750	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	3	1.000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	4	0.000	1.3947	-0.0004	2.9778	0.0086	-0.0458	-0.0306
1	4	0.250	1.3947	-0.0004	1.4903	0.0086	9.4488	-0.0290
1	4	0.500	1.3947	-0.0004	0.0028	0.0086	12.6214	-0.0273
1	4	0.750	1.3947	-0.0004	-1.4847	0.0086	9.4722	-0.0257
1	4	1.000	1.3947	-0.0004	-2.9722	0.0086	0.0012	-0.0241
1	5	0.000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	5	0.250	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	5	0.500	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	5	0.750	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	5	1.000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	6	0.000	0.7557	4.2477	0.0001	-0.0001	0.0046	0.9253
1	6	0.250	0.7557	2.1227	0.0001	-0.0001	0.0052	-12.6117
1	6	0.500	0.7557	-0.0023	0.0001	-0.0001	0.0058	-17.1175
1	6	0.750	0.7557	-2.1273	0.0001	-0.0001	0.0065	-12.5921
1	6	1.000	0.7557	-4.2523	0.0001	-0.0001	0.0071	0.9647

An example of the Code Check datasheet is shown below:

GT STRUDL Code Check Results

Member	P/F	Load	Section Loc	Crit Ratio	Crit Prov	Stress Ratio	Stress Prov	KL/r Ratio	KL/r Prov	Properties
1	Pass	4	8.500	1.006	H2-1 TEN	1.006	H2-1 TEN	0.634	B7 COMP	W8x24
2	FAIL	4	8.500	1.200	H1-3 COM	1.200	H1-3 COM	0.630	B7 COMP	W8x28
8	Pass	7	17.000	0.630	B7 COMP	0.462	H1-3 COM	0.630	B7 COMP	W8x28
3	FAIL	4	8.500	1.866	H2-1 TEN	1.866	H2-1 TEN	0.699	B7 COMP	W6x15
6	Pass	1	17.000	0.699	B7 COMP	0.394	H1-1 COM	0.699	B7 COMP	W6x15
7	FAIL	6	8.500	1.269	AKT CBEN	1.269	AKT CBEN	0.699	B7 COMP	W6x15
9	Pass	1	17.000	0.699	B7 COMP	0.112	H1-3 COM	0.699	B7 COMP	W6x15
4	Pass	6	8.500	0.555	H2-1 TEN	0.555	H2-1 TEN	0.505	B7 COMP	W8x31
19	Pass	1	15.000	0.891	B7 COMP	0.677	AKT CBEN	0.891	B7 COMP	W8x31
20	Pass	1	25.000	0.891	B7 COMP	0.382	H2-1 TEN	0.891	B7 COMP	W8x31
21	Pass	1	0.000	0.891	B7 COMP	0.348	H2-1 TEN	0.891	B7 COMP	W8x31
69	Pass	1	5.000	0.891	B7 COMP	0.230	H2-1 TEN	0.891	B7 COMP	W8x31
70	Pass	1	5.000	0.891	B7 COMP	0.367	H2-1 TEN	0.891	B7 COMP	W8x31
71	Pass	1	5.000	0.891	B7 COMP	0.414	H2-1 TEN	0.891	B7 COMP	W8x31
72	Pass	1	0.000	0.891	B7 COMP	0.414	H2-1 TEN	0.891	B7 COMP	W8x31
73	Pass	1	0.000	0.891	B7 COMP	0.368	H2-1 TEN	0.891	B7 COMP	W8x31
74	Pass	1	0.000	0.891	B7 COMP	0.231	H2-1 TEN	0.891	B7 COMP	W8x31
84	Pass	1	20.000	0.891	B7 COMP	0.539	H2-1 TEN	0.891	B7 COMP	W8x31
85	Pass	1	0.000	0.891	B7 COMP	0.539	H2-1 TEN	0.891	B7 COMP	W8x31
87	Pass	1	20.000	0.891	B7 COMP	0.539	H2-1 TEN	0.891	B7 COMP	W8x31
88	Pass	1	0.000	0.891	B7 COMP	0.539	H2-1 TEN	0.891	B7 COMP	W8x31
5	Pass	1	17.000	0.616	H1-1 COM	0.616	H1-1 COM	0.515	B7 COMP	W10x39
42	Pass	7	0.000	0.909	B7 COMP	0.076	H1-3 COM	0.909	B7 COMP	W10x39
43	Pass	1	5.000	0.909	B7 COMP	0.149	H1-3 COM	0.909	B7 COMP	W10x39
44	Pass	7	5.000	0.909	B7 COMP	0.237	H1-3 COM	0.909	B7 COMP	W10x39
45	Pass	7	5.000	0.909	B7 COMP	0.335	H1-3 COM	0.909	B7 COMP	W10x39
46	Pass	7	5.000	0.909	B7 COMP	0.437	H1-3 COM	0.909	B7 COMP	W10x39

The datasheets are used to display, review and organize results from a previous analysis or steel code check. You can not edit the the display values. Only active joints or members and loads are included in the display. You can double-click a column heading to sort the data based on the values in that column.

After the data is arranged in a convenient manner by using the Columns, Filter and Sort selections, you can request a text copy to be printed in the GT STRUDL

text output ('File -> Write display to text window') so you can highlight-and-print or copy-and-paste to another program such as Microsoft Excel®.

You can also put selected data on the Windows clipboard with the 'Edit -> Copy' selection and then paste into another program, such as a spreadsheet. You can also print directly from the datasheet to your printer using 'File -> Print', or create a "tab-delimited" file (for spreadsheets) by selecting 'File -> Write 'Tab' file'.

4. The input file created in GTMenu now contains the following additional data:
 - a. As mentioned previously in Section 2.2, the input file created in GTMenu now contains the data used to describe an Area load. Previously only the computed member loads were created in the input file making it difficult to modify the area loads as you had to recreate the Area Load commands. Shown below is an example of the Area Load command now created by GTMenu:

```

UNITS  MET.  NEW.  DEG  FAH
AREA LOAD  1          DIRECTION X
ELEVATION 3.000000
IGNORE MEMBERS  91          142          193          196          145          -
94
EXCEPT
LIMITS MEMBERS  194          163          143          159          -
LIMITS JOINTS   70          74          54          50          -
VALUE -25.000002 ONE WAY Y 1.000000 Z 1.000000
END AREA LOAD

```

- b. The input file will now contain the property specification for I-shapes, channels and polygons that were specified by commands as described in Section 2.1.9.2.6-8 of Volume 1 of the GT STRUDL Reference manuals. An examples of the commands that are created are shown below:

```

MEMBER PROPERTIES  CHANNEL TOTAL DEPTH 2.4000000E+01  -
WEB THI 7.2000003E-01  FLANGE WIDTH  8.0000000E+00  -
FLANGE THI  8.9999998E-01
7

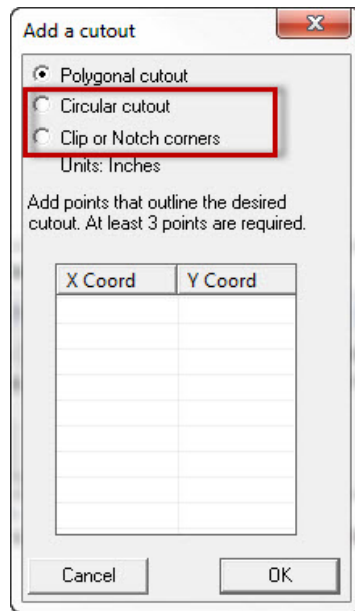
MEMBER PROPERTIES  I-SHAPE TOTAL DEPTH 2.0000000E+01  -
WEB THI 8.0000001E-01  FLANGE WIDTH  1.0000000E+01  -
FLANGE THI  1.0000000E+00
8

MEMBER PROPERTIES  POLYGON DIAMETER BET CORNERS  2.4000000E+01  -
SIDES  6.0000000E+00  THICKNESS  2.0000000E+00
9

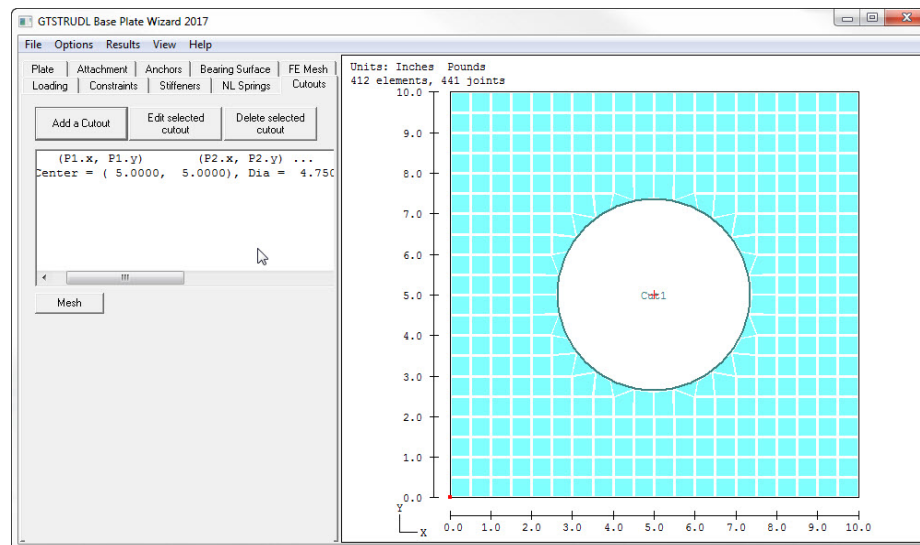
```

2.5 Base Plate

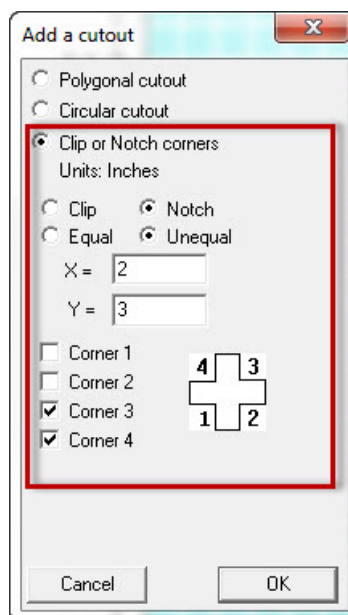
1. The Add a Cutout dialog has been enhanced to allow the user to specify circular cutouts in the Base Plate. In addition, a 'Clip Corners' option has been incorporated into the Add a Cutout dialog to include rectangular corner notches which are useful when you have tabs added to a base plate. The enhanced Add a Cutout dialog is shown below:



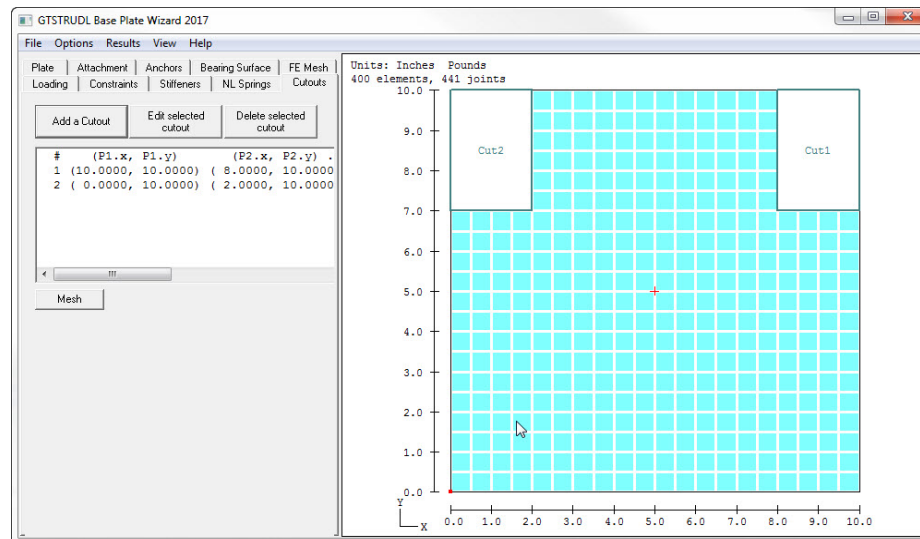
If the Circular cutout option is selected, you enter the x and y coordinates for the center of circular cutout and the diameter of the circular cutout. For example, enter the center of the cutout as x=5", y=5" and the diameter of the cutout as 4.75", results in the plate mesh shown on the next page:



To specify a rectangular (notch) cutout, you specify if the cutout has equal or unequal x and y cuts and the values for these cuts as well as the corners of the plate where you want to apply the cutout. In the cutout dialog shown below, unequal notches with $x=2$ " and $y=3$ " were applied at corners 3 and 4 of the base plate.



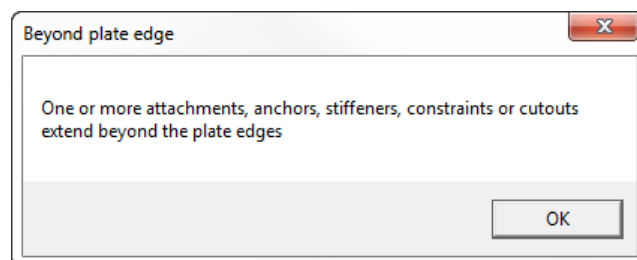
The resulting plate mesh is shown on the next page:



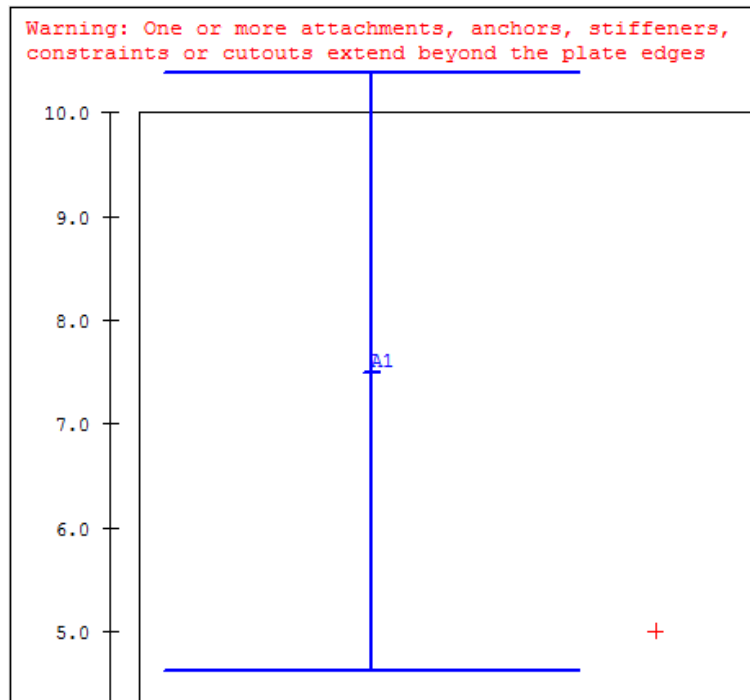
More information on these enhancements may be found in Section 2.3 The Cutouts Tab in the GT STRUDL Base Plate Wizard Users Guide.

2. The “off the plate” warning has been changed from a popup Windows Message Box to a red text warning in the upper, left corner of the plate display window. No meshing can be done until all components are inside the plate boundaries.

In Versions 2016R2 and earlier, the following warning pop-up Message Box appeared when one of the components was off the plate:



In Version 2017, a warning message in red is shown at the top of the plate display window as shown on the next page:



2.6 GTShell (GT STRUDL Output Window)

1. GTShell and GTMenu now use the same results datasheets for Joint Displacements, Joint Reactions, Member End Forces, Section Forces and Code Check results. The Joint Reactions datasheet is new to GTShell. Examples of these datasheets may be found in Section 2.4.
2. The Steel Design Parameters dialog has been modified to include the new Canadian steel design code CSA-2014 presented in Section 2.1 and the additional code parameters. An illustration of the modified Parameters dialog is shown on below:

Steel Design PARAMETERS Command

CODE Name: **CSA-2014** Units: **FEET** POUNDS

Commonly Used | All Parameters | Alphabetic

Parameter Names	Parameter Values (default values)
Class4	NO
Cls13.5Z	YES
PrintLim	NO
Material Property Parameters	
STEELGRD	A36
Fy	Computed
Fu	Computed

Member List Processing

All Members | Member List Selection | Same as Previous Member List

OK | Cancel | Help

Help

CSA-2014

Identifies the code to be used for member checking or member selection. Specify CSA-2014 for the code name. CSA-2014 code is applicable to hot rolled I-shapes, Channels, Rectangular Hollow Sections, Circular Hollow Sections, Tees, Double Angles, Solid Rectangular Bars, and Solid Round Bars. See Sections 2 and 3 for a more detailed description

Selected Order | Alphabetic

Parameter Names	Parameter Values	Member, Joint, or Load List	UNITS
CODE	CSA-2014	ALL MEMBERS	
STEELGRD	350W	ALL MEMBERS	

2.7 Steel Tables

1. New Chinese steel tables have been added based on Chinese National Standards Press publications. The following 11 new tables have been added:

GBI	rolled I shapes
GBH	rolled H and W shapes
GBWELDH	welded H shapes
GBLITEH	welded light gauge H shapes
GBTEE	rolled Tee shapes
GBCHAN	rolled Channel shapes
GBL-EQ	rolled equal leg single angles
GBL-UN	rolled unequal leg single angles
GBRBAR	round bars
GBSQBAR	square bars
GBTUBE	seamless circular steel tubes

A summary of these tables and the profiles in the tables is available in Appendix C of Volume 2A of the GT STRUDL Reference Manuals. You may also view the tables and profiles using the Profile Browser under the Steel Design pulldown or in GTMenu. The new tables and profiles are also available in CAD Modeler and the Model Wizard.

2.8 Finite Elements

1. Modifications have been made to improve the numerical precision of all static and dynamic analysis results computed for two- and three-dimensional finite elements. A comparison between such results computed under Version 2017 and those of previous versions can be expected to show differences in the range of 0.01% or less.

2.9 Reinforced Concrete Design

1. The algorithm that is used by the CHECK MEMBER COLUMN command for the computation of the reinforced concrete column axial force and bending moment capacities is improved in Version 2017. The CHECK MEMBER COLUMN command now computes and reports more consistently accurate axial

force and bending moment capacity results, particularly when the eccentricity associated with applied axial and bending moment loads is smaller than 3.0 inches.

Chapter 3

Error Corrections

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

3.1 CAD Modeler

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

1. CAD Modeler Setup now includes the following path:
"Drive:\Program Files\Autodesk\AutoCAD xxxx\Support"
for the corresponding AutoCAD version in the BAT file in order to avoid the annoying message "Unable to find main dictionary".

3.2 DBX

1. The following DBX data types previously limited the specified file name (including the path) to 80 characters. Now these types support 256 character file names.
MEMBER PROPERTIES, MEMBER RESULTS, RC DESIGN and
SUPERELEMENTS
(No GPRF issued)
2. The DBX output now produces correct dates. The dates were having the Y2K problem (i.e., 1917 instead of 2017).
(No GPRF issued)

3.3 General

1. The specification of channel and polygon member properties in the MEMBER PROPERTIES header will no longer produce an error. Previously, the following error message would occur as shown in the example below:

```
MEMBER PROPERTIES CHANNEL TOTAL DEPTH 2.4000000E+01 -  
WEB THI 7.2000003E-01 FLANGE WIDTH 8.0000000E+00 -
```

FLANGE THI 8.9999998E-01

1

**** STRUDL ERROR 1.614 - MEMBER TYPE NOT SPECIFIED

(GPRF 2016.06)

2. The WIND LOAD command now produces correct, non-zero member loads under when the following conditions are present:
 - a. Member properties for the affected members are defined by the MEMBER PROPERTIES POLYGON command in which the DIAMETER is specified explicitly as (BETWEEN) FLATS, or not specified, in which case (BETWEEN) FLATS is assumed. The following MEMBER PROPERTIES command illustrates the previously problematic MEMBER PROPERTIES POLYGON data:

```

UNITS INCHES
MEMBER PROPERTIES
'C1' POLYGON TUBE DIAM 19.7000 SIDES 8 THICK 0.18750
'C2' POLYGON TUBE DIAM 19.1000 SIDES 8 THICK 0.18750
'C3' POLYGON TUBE DIAM 18.5000 SIDES 8 THICK 0.18750
'C4' POLYGON TUBE DIAM 17.9000 SIDES 8 THICK 0.18750
'C5' POLYGON TUBE DIAM 17.3000 SIDES 8 THICK 0.18750
'C6' POLYGON TUBE DIAM 16.7000 SIDES 8 THICK 0.18750
'C7' POLYGON TUBE DIAM 16.1000 SIDES 8 THICK 0.18750
'C8' POLYGON TUBE DIAM 15.5000 SIDES 8 THICK 0.18750
'C9' POLYGON TUBE DIAM 14.9000 SIDES 8 THICK 0.18750
'C10' POLYGON TUBE DIAM 14.3000 SIDES 8 THICK 0.18750

```

The DIAMETER values (DIAM) refer by default to the distance between diametrically opposite flat sides.

- b. In the WIND LOAD command, the AREA AND FORCE PROPERTIES option is specified for the affected members as illustrated by the following example in which the AREA AND FORCE PROPERTIES option is highlighted:

```

WIND LOAD 'WX' '140 MPH Wind on Structure in +X Direction'
UNITS FT LBS DEG SEC StdMASS
STANDARD ASCE7-05
ELEVATION AXIS Y
EXPOSURE CATAGORY C
SPEED MPH 140.0000
DIRECTION ANGLE 0.0000
DIRECTIONALITY FACTOR KD 1.00
TOPOGRAPHIC FACTOR KZT 1.00
GUST FACTOR G 0.96
GROSS AREA AG 999999.00

```

```

ADDED FORCE AREA AFADD 0.0000
UNITS FT LBS DEG SEC StdMASS
MEMBERS EXISTING GROUP LIST 'COLUMN' TYPE LATTICE FRAMEWORK -
  AREA AND FORCE PROPERTIES THICE 0.0000 THINS 0.0000 -
  KZ 0.8490 CF 1.9000 QZ 53.8809 FLD 1.0000

```

```

MEMBERS EXISTING GROUP LIST '0201' '0301' '0302' -
  TYPE LATTICE FRAMEWORK -
  AREA AND FORCE PROPERTIES THICE 0.0000 THINS 0.0000 -
  KZ 0.8490 CF 1.6000 QZ 53.8809 FLD 1.0000
END WIND LOAD DATA

```

Documentation

The MEMBER PROPERTIES Command (cross-section specs), Volume 1, Section 2.1.9.2, GT STRUDL Reference Manual

The WIND LOAD Command, Volume 1, Section 2.1.11.3.9.1, GT STRUDL Reference Manual

(GPRF 2017.02)

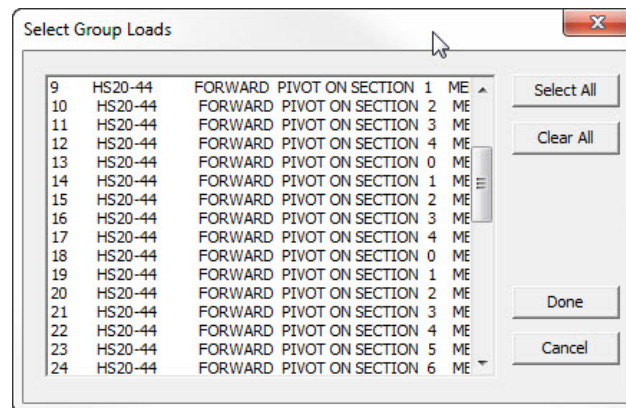
3.4 GTMenu

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

1. All Mode Bar settings are now available in the Display Load dialog for the display using the Joint, Member and Element options in the Selection panel. The ALL setting on the Mode Bar is the only option shown if All is chosen in the Selection panel of the Display Load dialog. Previously, other Mode Bar settings were shown but did not work.
2. When you switch between the Display Load dialog and other dialogs, the Mode Bar setting is now remembered when you return to the dialog.
3. Loading and element descriptions are now completely able to be seen in many dialogs due to the addition of a horizontal scroll bar. Previously, long loading descriptions or the descriptions of finite elements was truncated.
4. After completing Edit member or Edit joint functions, the member or joint selection is now erased. Previously, the members or joints were shown as still being selected after editing them.

5. If you right click the mouse before selecting anything when deleting joints, members or elements, the Undo Deleted button is now grayed out.
6. The text in the Edit Member Releases dialog has been corrected to say “Elastic Connections” instead of “Elastic Supports.”
7. The text in the Edit Member Data dialog has been corrected to say “End Joint Size” instead of “Edit Joint Size.”
8. List mode is now available in the Generate Display List option in the Display Model dialog. Previously if List mode was active, no dialog would appear to allow you to create a list.
9. The All Mode Bar option is now working in the Create and Edit Load dialogs.
10. When creating or editing data in Hit or List mode, the text on the right of the Mode Bar would always indicate Joints. The text now reflects the entities being created or edited.
11. Dialogs are no longer popped up when editing data in Hit, List or All modes if no selection is made. For example, in previous versions if you right clicked the mouse without making a selection in a dialog such as the Edit Coordinates dialog, a dialog would be popped up asking you to enter coordinates.
12. The All Mode Bar option now works in the Edit Joint dialog when you select Inquire and then the All button in the Data Type panel.
13. The active loads are now retained when leaving GTMenu after executing the functions in the Moving Load dialog. Previously, some loads would be marked as inactive which was incorrect.
14. An empty dialog is no longer popped up when Editing ID's and you select the Manual option in the Edit Action panel but right clicking the mouse before making any selection.
15. The Display Moving Load function now works when the GTSES or GT64M solvers are used. Previously, the Draw Deformed Shape and Draw Member Diagrams options were grayed out in the dialog when you had selected the GTSES or GT64M solver.
16. You can now create Groups which contain Loads. Previously, you could never create a group which contained loads. Either nothing would happen or an abort

would occur after clicking the Loads button in the Create Groups dialog. Now, a dialog such as the one shown below will pop-up which will allow you to select the loads to be placed in the Load group.



3.5 GTShell (GT STRUDL Output Window)

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

1. The Wind Load dialog for ASCE 7-05 now creates the IMPORTANCE FACTOR command.

3.6 Offshore/Fatigue Analysis

(GPRF's are **not** issued for offshore and fatigue analyses unless specifically noted below)

1. Version 2016 introduced a problem with the READ WAVE LOADS command when the LOADS option is included to where the command can process only the first 10 loadings in the list of loads. This problem has been corrected.

Documentation

The READ WAVE LOAD command when the LOADS option is included, Section 5.2.1, Volume 8 of the Reference Manual.

3.7 Reinforced Concrete Design

1. The reinforced concrete CHECK MEMBERS command now supports the static analysis results that are created and stored in hard drive files when the GTSES and GT64M solvers are used for static and dynamic analyses. In previous versions the CHECK MEMBERS command issued an error message when

expected analysis results were created and stored by the GTSES and GT64M solvers.

(GPRF 2016.04)

Documentation

The CHECK MEMBERS command, Section 7.3, Volume 4 of the Reference Manual.

2. The following properties are now correct for the RL, LL, Tee and Cross shapes calculated using the MEMBER DIMENSION command:

RL and LL - ZC

Tee - ZC

Cross - YC, YD, ZC and SZ

(GPRF 2017.01)

3.8 Steel Design

2. The following error message which may have been given during the CHECK WELD or SIZE WELD command has been fixed.

CI-w-cmd23, ERROR: Commands longer than 23 characters are not processed.

(GPRF 2016.05)

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 2017. 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 CAD Modeler

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

1. Loads are not copied or mirrored when using the Copy or Mirror commands.
2. The Beta angles and Loads are not rotated or mirrored when using the Rotate or Mirror commands.

4.2 Finite Elements

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

4.3 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 when all of the following conditions are present:
 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 GT STRUDL.

(GPRF 2004.14)

4.4 GTMenu

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

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 GT STRUDL 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 GT STRUDL Reference Manual.

(GPRF 95.21)

3. The Check Load option in CHECK MODEL dialog 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 GT STRUDL 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.

(No GPRF issued)

6. The Deformed Structure display with the Deform between Joints option may produce inconsistent results for nonlinear geometric frame members. The deformed structure may show a discontinuity at the joints.

(No GPRF issued)

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

(No GPRF issued)

Chapter 5

Prerelease Features

5.1 Introduction

This chapter describes new features that have been added to GT STRUDL 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 GT STRUDL 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 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)
- 5.2.2 ASCE4805 Steel Design Code. This code is for the ultimate strength design of steel transmission pole structures.

5.3 Analysis Prerelease Features

- 5.3.1 Calculate Error Estimate Command
- 5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

5.4 General Prerelease Features

- 5.4.1 Rotate Load Command
- 5.4.2 Reference Coordinate System Command
- 5.4.3 GTMenu Point Coordinates and Line Incidences Commands

5.4.4 GTMenu Surface Definition Command

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

5.2 Design Prerelease Features

5.2.1 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) -
 $\left\{ \begin{array}{l} \text{WOOD (AND) (ARMER)} \left\{ \begin{array}{l} \rightarrow \text{AVERAGE} \\ \text{MAXIMUM} \end{array} \right\} \\ \text{CALCULATE (RESULTANT) (ELEMENT) (FORCES)} \end{array} \right\} (\text{ALONG}) -$
 $(\text{CUT } \left\{ \begin{array}{l} \text{'a'} \\ i_1 \end{array} \right\}) \left\{ \begin{array}{l} \text{JOINTS} \\ \text{NODES} \end{array} \right\} \text{list}_1 \text{ ELEMENT list}_2 (\text{TABLE } \left\{ \begin{array}{l} \rightarrow \text{ASTM} \\ \text{UNESCO} \end{array} \right\}) -$
 $* \left\{ \begin{array}{l} \text{TOP (FACE) (BARS } i_2 \text{) (SPACING } v_1 \text{)} \\ \text{BOTTOM (FACE) (BARS } i_3 \text{) (SPACING } v_2 \text{)} \\ \text{BOTH (FACES) (BARS } i_4 \text{) (SPACING } v_3 \text{)} \end{array} \right\} -$
 $\left\{ \begin{array}{l} \rightarrow \text{INNER (LAYER)} \\ \text{OUTER (LAYER)} \end{array} \right\} (\text{COVER } v_4) (\text{LINEAR (TOLERANCE) } v_5) -$
 $(\text{TORSIONAL (MOMENT) (WARNING) } v_6)$

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, MY, 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.

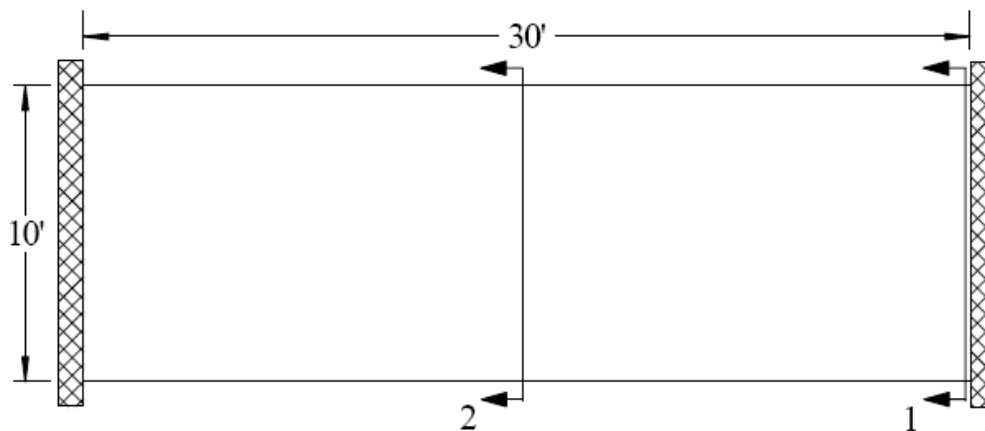


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

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.

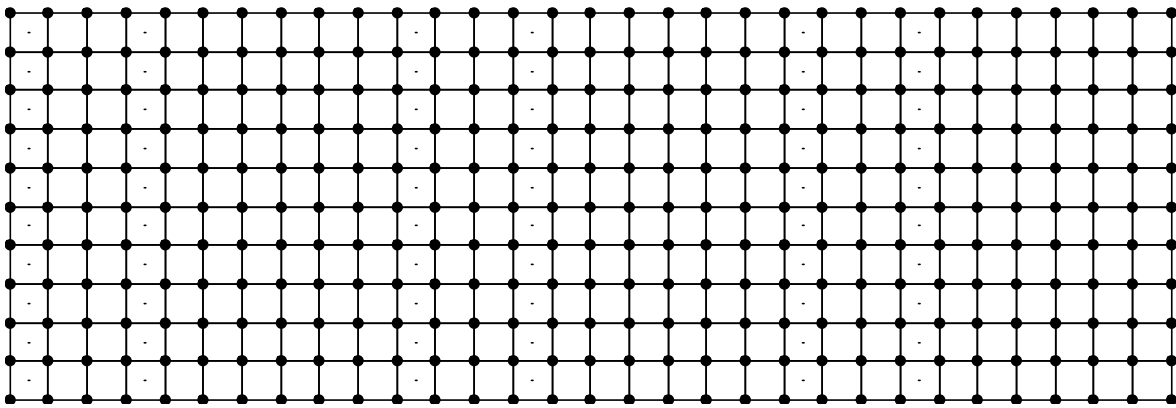


Figure 5.2.3-2 Example Finite Element Model

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 GT STRUDL 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

```

5.2.2 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 GT STRUDL Output Window.

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

$$\begin{aligned} & \text{CALCULATE ERROR (ESTIMATE) (BASED ON) -} \\ & * \left\{ \begin{array}{l} \text{ENERGY (NORM)} \\ \text{MAX DIFFERENCE} \\ \text{DIFFERENCE FROM AVERAGE} \\ \text{PERCENT MAX DIFFERENCE} \\ \text{PERCENT DIFFERENCE FROM AVERAGE} \\ \text{NORMALIZED PERCENT MAX DIFFERENCE} \\ \text{NORMALIZED PERCENT DIFFERENCE FROM AVERAGE} \end{array} \right\} - \\ & (\text{AT}) * \left\{ \begin{array}{l} \text{TOP} \\ \text{MIDDLE} \\ \text{BOTTOM} \end{array} \right\} (\text{SURFACES}) (\text{FOR}) \left\{ \begin{array}{l} \rightarrow \text{ALL} \\ \text{ELEMENT list} \end{array} \right\} \end{aligned}$$

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:

$$\|e_\sigma\|_{L_2} = \left(\int_{\Omega} (e_\sigma)^T (e_\sigma) d\Omega \right)^{1/2}$$

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,

$$\|e_\sigma\|_{L_2} = \left(\int_{\Omega} (\sigma^* - \sigma)^T N^T \cdot N (\sigma^* - \sigma) d\Omega \right)^{1/2}$$

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:

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

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 from the individual element stress output vector (maximum value from LIST STRESS output for all nodes). 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 can become 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_CHKECBTA -- 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 GT STRUDL User Reference Manual.

2.1.11.4.6 The ROTATE LOAD Command

General form:

$$\underline{\text{ROTATE}} \quad \underline{\text{LOADING}} \quad \left\{ \begin{array}{c} \underline{i_R} \\ \\ \underline{'a_R'} \end{array} \right\} (\underline{\text{ANGLES}}) [\underline{T1}] r_1 [\underline{T2}] r_2 [\underline{T3}] r_3$$

Elements:

$\underline{i_R}/\underline{'a_R'}$ = 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 $\theta_1, \theta_2, \theta_3$ 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 GT STRUDL 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:

$$\text{REFERENCE (COORDINATE) (SYSTEM) } \left\{ \begin{array}{c} i_1 \\ 'a_1' \end{array} \right\} - \left\{ \begin{array}{c} (\text{ORIGIN } [\underline{X}] v_x [\underline{Y}] v_y [\underline{Z}] v_z) (\text{ROTATION } [\underline{R1}] v_1 [\underline{R2}] v_2 [\underline{R3}] v_3) \\ \left\{ \begin{array}{c} \text{JOINT } \left\{ \begin{array}{c} i_2 \\ 'a_2' \end{array} \right\} \\ \left[\begin{array}{ccc} \underline{X} v_4 & \underline{Y} v_5 & \underline{Z} v_6 \end{array} \right] \end{array} \right\} \left\{ \begin{array}{c} \text{JOINT } \left\{ \begin{array}{c} i_2 \\ 'a_2' \end{array} \right\} \\ \left[\begin{array}{ccc} \underline{X} v_4 & \underline{Y} v_5 & \underline{Z} v_6 \end{array} \right] \end{array} \right\} \left\{ \begin{array}{c} \text{JOINT } \left\{ \begin{array}{c} i_2 \\ 'a_2' \end{array} \right\} \\ \left[\begin{array}{ccc} \underline{X} v_4 & \underline{Y} v_5 & \underline{Z} v_6 \end{array} \right] \end{array} \right\} \end{array} \right\}$$

Explanation:

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

At the present time, this command is used to specify additional coordinate systems which may be used in GTMenu (see Volume 2 of the GT STRUDL 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 GT STRUDL 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 'a₂' 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 'a₃' 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 'a₄' 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 GT STRUDL. 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:

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

Explanation:

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

5.4.3 GTMenu POINT COORDINATE and LINE INCIDENCES Commands

GTMenu can now write construction geometry commands to an input file, which can be read later into GT STRUDL 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

$$\begin{array}{l} \{ 'a_1' \} \text{ coordinate-specs}_1 \\ \cdot \\ \cdot \\ \cdot \\ \{ 'a_n' \} \text{ coordinate-specs}_n \end{array}$$

Elements:

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

where,

$'a_1', 'a_2', \dots, 'a_n'$ = 1 to 8 character alphanumeric Point identifiers beginning with P (i.e. P1 P2 ...)

v_1, v_2, v_3 = Cartesian Point coordinates (integer or real)

(2) **GTMenu LINE INCIDENCES****General Form:**GTMenu LINE INCIDENCES $\{ 'a_1' \} \text{ type}_1 \text{ incidence-specs}_1$ $\cdot$
 $\cdot$
 $\cdot$ $\{ 'a_n' \} \text{ type}_n \text{ incidence-specs}_n$ **Elements:**

$$\text{type} = \left\{ \begin{array}{l} \rightarrow \underline{\text{LINE}} \\ \underline{\text{POLYNOMINAL}} (\underline{\text{CURVE}}) \\ \underline{\text{ARC}} (\underline{\text{TEMPLATE}}) \\ \underline{\text{CENTERED}} (\underline{\text{ARC}}) \underline{\text{PERCENT}} v_1 \\ \underline{\text{BEZIER}} (\underline{\text{CURVE}}) \\ \underline{\text{SPLINE}} (\underline{\text{CURVE}}) (\underline{\text{ORDER}} k_2) \end{array} \right\}$$

$$\text{incidence-specs} = \{ \text{point}_1' \} \{ \text{point}_2' \} . . . \{ \text{point}_p' \}$$

where,

'a₁', 'a₂', ..., 'a_n' = 1 to 8 character alphanumeric Line/Curve identifiers beginning with C (i.e. C1, C2).

'point₁', ..., 'point_p' = 1 to 8 character alphanumeric Point identifiers beginning with P (i.e. P1, P2).

v₁ = positive number (integer or real).

k₂ = 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

5.4.4 GTMenu SURFACE DEFINITION Command

GTMenu construction geometry commands that are written to an input file have been enhanced with the ability to write/read Surface Definitions. Although this prerelease feature is intended mainly to support the save/restore of Surfaces defined through the GTMenu Graphical Interface, users may be able to edit or create new Surfaces through commands provided the point, curve and surface naming rules are followed.

General Form:

GTMenu SURFACE DEFINITION

$$\begin{array}{c} \{ 'a_1' \} \text{ surface-specs}_1 \\ \vdots \\ \{ 'a_n' \} \text{ surface-specs}_n \end{array}$$

Elements:

$$\text{surface-specs} = \left\{ \begin{array}{l} (\underline{\text{PATCH}} \underline{\text{SURFACE}} \underline{\text{SPACING}}) \text{ } iu \text{ } iv \text{ } \text{patch-specs} \\ (\underline{\text{SURFACE OF}}) \underline{\text{REVOLUTION}} (\underline{\text{SPACING}}) \text{ } iu \text{ } iv \text{ } \text{sor-specs} \end{array} \right\}$$

$$\text{patch-specs} = U (\underline{\text{CURVES}}) 'b_1' \dots 'b_n' \text{ } V (\underline{\text{CURVES}}) 'c_1' \dots 'c_m'$$

$$\text{sor-specs} = (\underline{\text{REVOLUTION}} \underline{\text{ANGLE}}) \text{ } v \text{ } \text{axis-specs} \text{ } U (\underline{\text{CURVE}}) 'b_1'$$

$$\text{axis-specs} = (\underline{\text{AXIS}}) \left\{ \begin{array}{l} \underline{\text{POINTS}} 'd_1' 'd_2' \\ \underline{\text{COORDINATES}} \underline{\text{START}} x_1 \text{ } y_1 \text{ } z_1 \text{ } \underline{\text{END}} x_2 \text{ } y_2 \text{ } z_2 \end{array} \right\}$$

where,

'a ₁ ', 'a ₂ ', ..., 'a _n '	=	1 to 8 character alphanumeric Surface IDs beginning with S (i.e. S1, S2).
iu, iv	=	integer values representing the number of drawing segments to use in directions U and V respectively.
'b ₁ ', 'b ₂ ', ..., 'b _n '	=	1 to 8 character alphanumeric Line/Curve IDs for U direction. n must be greater than or equal to 1 and less than or equal to 10. Line/Curve IDs begin with C (i.e. C1,C2).
'c ₁ ', 'c ₂ ', ..., 'c _m '	=	1 to 8 character alphanumeric Line/Curve IDs for V direction. m must be greater than or equal to 1 and less than or equal to 10. Line/Curve IDs begin with C (i.e. C1,C2).
v	=	real number representing the angle of revolution.
'd ₁ ', 'd ₂ '	=	1 to 8 character alphanumeric Point IDs for start and end points of the axis of revolution respectively. Point IDs begin with P (i.e. P1,P2).
X _i , Y _i , Z _i	=	real values representing coordinates for global directions X, Y, Z respectively of the start and end points of the axis of revolution.

Examples:

```
GTMenu SURFACE DEFINITION
'S1' PATCH SURFACE SPACING 10 20 -
  U CURVES 'C1' -
  V CURVES 'C2'

'S2' SURFACE OF REVOLUTION SPACING 10 20 -
  REVOLUTION ANGLE 60.5 -
  AXIS POINTS 'P1' 'P6' -
  U CURVE 'C2'
```

```
'S3' SURFACE OF REVOLUTION SPACING 10 20 -  
    REVOLUTION ANGLE 360 -  
    AXIS COORDINATES START 10.0 0.0 10.0 -  
                           END 20.0 0.0 0.0 -  
U CURVE 'C2'
```

End of Document