

GT STRUDL® Version 2016 R2 Release Guide



PROCESS, POWER & MARINE

Release Guide

Release Date: October 2016



Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 2016 R2 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

Copyright © 2016 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- Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States.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Unpublished - rights reserved under the copyright laws of the United States.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 2016 R2	
2.1 CAD Modeler	2-1
2.2 General	2-5
2.3 GTMenu	2-10
2.4 Finite Elements	2-20
2.5 Base Plate	2-21
2.6 Scope Editor	2-22
 CHAPTER 3 ERROR CORRECTIONS	
3.1 CAD Modeler	3-1
3.2 DBX	3-1
3.3 File -> Export -> CIS/2	3-1
3.4 Finite Elements	3-1
3.5 GTMenu	3-2
3.6 GTShell (GT STRUDL Output Window)	3-4
3.7 SPLM Licensing	3-4
3.8 Steel Design	3-4

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)	5.2-4
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 2016 R2 covers GT STRUDL operating on PC's under the Windows 10, 8 and 7 operating systems. For users who are accustomed to our older version numbering system, the version is internally known as Version 35.2.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2016 R1. In particular, Chapter 2 briefly describes an extensive list of new features including the following significant new features:

- CAD Modeler now runs on BricsCAD Pro and Platinum editions (Version 16.2.09) in addition to AutoCAD.
- A new WIND LOAD command has been implemented to create GT STRUDL independent loading conditions comprised of applied member and joint loads computed according to ASCE Standards 7-05 and 7-10

Chapter 3 provides you with details regarding error corrections that have been made since the Version 2016 R1 release. Chapter 4 describes known problems with Version 2016 R2. 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 2016 R2

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 2016 R2. This release guide is also available online upon execution of GT STRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 CAD Modeler

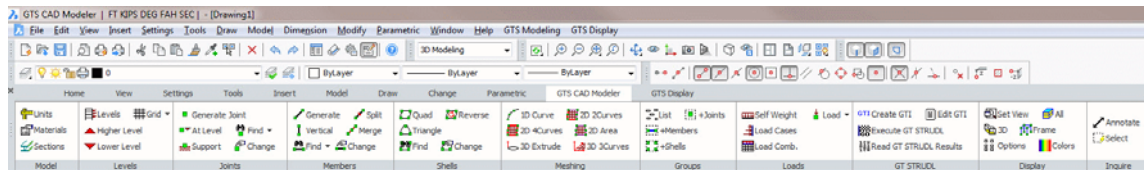
CAD Modeler now runs on BricsCAD Pro and Platinum editions (Version 16.2.09) in addition to AutoCAD. CAD Modeler was first released in Version 2015 as an add-on to AutoCAD which allows you to create GT STRUDL Input Files (.gti files) graphically using the CAD tools and graphical display capabilities. All of the CAD Modeler features available in AutoCAD are now available in BricsCAD.

For those not familiar with CAD Modeler, a summary of the important features is shown below:

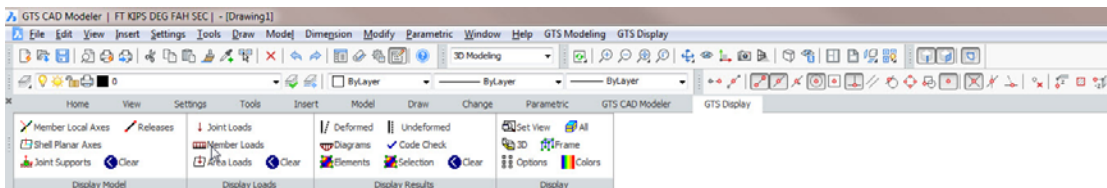
- Create GT STRUDL models using CAD functionality
- Import, View and Modify existing GT STRUDL models
- Switch between different floors or levels and also use a column grid to facilitate model creation and modification
- Generate GT STRUDL joints and control their properties such as constraints and spring values
- Choose any of the steel shapes available in GT STRUDL standard tables
- Generate GT STRUDL members and be able to define their cross-section, material, releases, elastic connections and member eccentricities
- Generate member and finite element meshes along a curve, between lines and extruded polyline surfaces
- Create 2D area meshes of triangular finite element meshes for regions defined using closed polylines with internal boundary and point constraints
- Use CAD functions such as move, copy, delete, rotate and mirror in the creation and modification of your models
- Switch between 3D Solid and wireframe views of your model
- Load your model using self-weight loadings, joints loads, members loads, finite element loads and area loads
- Create Load Combinations
- Define Groups and control the display of members and finite elements with Groups and include the Group definitions in the GT STRUDL input file created by CAD Modeler

- Locate potential errors in your model by checking and eliminating duplicate and floating joints
- Create GT STRUDL input files (.gti files) identical to the ones created by GTMenu
- View results such as the deformed shape, member force/moment diagrams, finite element contours and code check pass-fail results
- Use GT STRUDL units at any time

CAD Modeler adds two entries to the Ribbon Bar of BricsCAD and AutoCAD. The GTS CAD Modeler and GTS Display additions to the BricsCAD Ribbon Bar are shown below:

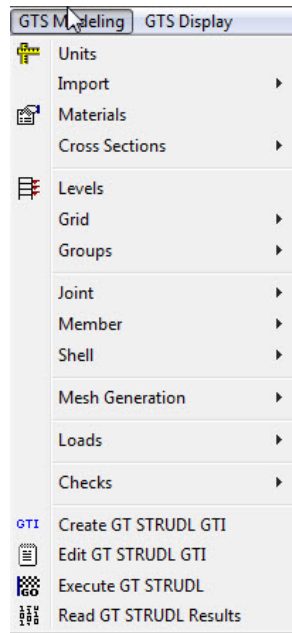


BricsCAD GTS CAD Modeler Ribbon Bar

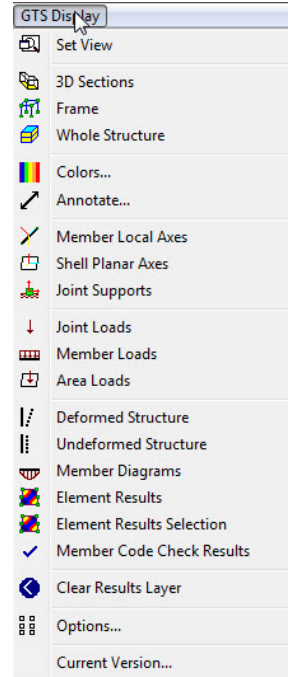


BricsCAD GTS Display Ribbon Bar

In addition, GTS Modeling and GTS Display pulldowns are added to the BricsCAD and AutoCAD Menu Bar. The BricsCAD pulldowns are shown on the next page:

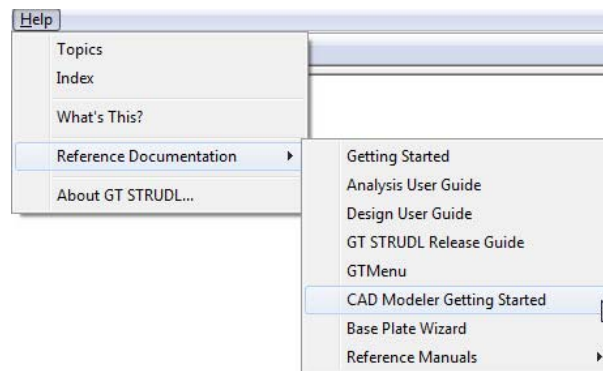


GTS Modeling pulldown



GTS Display pulldown

The CAD Modeler Getting Started Guide has been revised to include changes for BricsCAD. The Getting Started Guide is available under Help in the GT STRUDL Shell as shown below:



Additional new features and enhancements have been added to CAD Modeler and are summarized below:

- AutoCAD 2017 is now supported.
- Loads are displayed as they are being applied.
- Member releases can now be displayed on members using the Display Model panel on the GTS Display Ribbon Bar. Member releases are displayed similar to the method used in GTMenu.
- The magnification for loads has been increased so loads are more easily seen without the user having to adjust the scale factor.
- Large models can be imported significantly faster. In addition, the property pages for members are brought up much faster on models which contain a large number of member groups.
- The colors used in CAD Modeler have been adjusted to be more visible when a white background is used.
- When selecting members from a large table such as W-AISC14, you can now scroll through the table quicker using the mouse thumbwheel.
- For uniform member loads, FR (fractional) is now the default and the start and end are set to 0.0 and 1.0 respectively so a uniform load is applied over the entire member's length by default.
- The default factor for self weight loads has been set to 1.0.
- Labels for members, elements, joints and loads and result annotations now appear in the screen plane rather than only the global x-y plane.

2.2 General

1. A new WIND LOAD command is now available to create a GT STRUDL independent loading condition comprised of applied member and joint loads computed according to ASCE Standards 7-05 and 7-10, Minimum Design Loads for Buildings and Other Structures, and the ASCE publication Wind Loads for Petrochemical and Other Industrial Facilities, prepared by the ASCE Task Committee on Wind-Induced Forces. The WIND LOAD command and the WIND LOAD main shell dialog are shown below:

```

WIND LOAD {iWL  
           {aWL} ('descrWL')  

(UNITS...)  

general data  

(UNITS...)  

list0 member force / area data  

⋮  

(UNITS...)  

listn member force / area data  

END (OF) (WIND) (LOAD) (DATA)

```

where

general data =

$$\left\{ \begin{array}{l}
 \left(\text{STANDARD} \left\{ \begin{array}{l} \rightarrow \text{ASCE 7-05} \\ \text{ASCE 7-10} \end{array} \right\} \right) \\
 \left(\text{ELEVATION (AXIS)} \left\{ \begin{array}{l} \rightarrow \text{Y} \\ \text{Z} \end{array} \right\} \right) \\
 \left(\text{DIRECTION ANGLE} \left\{ \begin{array}{l} \rightarrow 0.0 \\ v_{\text{DIR}} \end{array} \right\} \right) \\
 \text{SPEED } v_s \left\{ \begin{array}{l} \rightarrow \text{MPH} \\ \text{M/S} \end{array} \right\} \\
 \text{EXPOSURE (CATEGORY)} \left\{ \begin{array}{l} \text{B} \\ \text{C} \\ \text{D} \end{array} \right\} \\
 \text{TOPOGRAPHIC (FACTOR)} \left\{ \begin{array}{l} [\text{KZT}] v_{\text{KZT}} \\ [\text{K1}] v_{\text{K1}} \quad [\text{K2}] v_{\text{K2}} \quad [\text{K3}] v_{\text{K3}} \end{array} \right\} \\
 \left(\text{DIRECTIONALITY FACTOR (KD)} \left\{ \begin{array}{l} \rightarrow 1.0 \\ v_{\text{KD}} \end{array} \right\} \right) \\
 \left(\text{IMPORTANCE (FACTOR)} \left\{ \begin{array}{l} \rightarrow 1.0 \\ v_{\text{I}} \end{array} \right\} \right) \\
 \text{GUST (FACTOR) (G)} v_{\text{G}} \\
 \text{GROSS (AREA) (AG)} v_{\text{AG}} \\
 \text{ADDED (FORCE) (AREA) (AFADD)} v_{\text{ASADD}}
 \end{array} \right\}$$

member force / area data =

$$\begin{array}{c}
 \left\{ \begin{array}{l}
 \text{INDUSTRIAL (FRAME) (MEMBERS)} \\
 \text{TOWER (MEMBERS) -} \\
 \left\{ \begin{array}{l} \rightarrow \text{SQUARE} \\ \text{SQUARE DIAGONAL} \\ \text{TRIANGLE} \end{array} \right\} \\
 \text{LATTICE (FRAMEWORK) (MEMBERS)} \\
 \left\{ \begin{array}{l} \text{SQUARE} \\ \text{SQUARE DIAGONAL} \\ \text{HEXAGONAL} \\ \text{OCTAGONAL} \end{array} \right\} \\
 \text{C-T-R-S} \left\{ \begin{array}{l} \text{MODERATELY (SMOOTH)} \\ \text{ROUND (ROUGH)} \\ \text{VERY (ROUGH)} \end{array} \right\}
 \end{array} \right\} - \\
 \\
 \text{FORCE (AREA)} \left\{ \begin{array}{l} \text{industrial} \\ \text{tower} \\ \text{lattice} \\ \text{c-t-r-s} \end{array} \right\} \text{ (miscellaneous / override data)}
 \end{array}$$

and where

industrial =

$$\left\{ \begin{array}{l} \rightarrow \text{PROPERTIES} \\ \text{RECTANGLE [B] } v_B \text{ [H] } v_H \\ \text{ROUND [D] } v_D \\ \text{AF } v_{AS} \end{array} \right\} (\text{THICE } v_{THICE}) (\text{THINS } v_{THINS}) \text{ CDG } v_{CDG}$$

tower = lattice =

$$\left\{ \begin{array}{l} \rightarrow \text{PROPERTIES} \\ \text{RECTANGLE [B] } v_B \text{ [H] } v_H \\ \text{ROUND [D] } v_D \\ \text{AF } v_{AS} \end{array} \right\} (\text{THICE } v_{THICE}) (\text{THINS } v_{THINS})$$

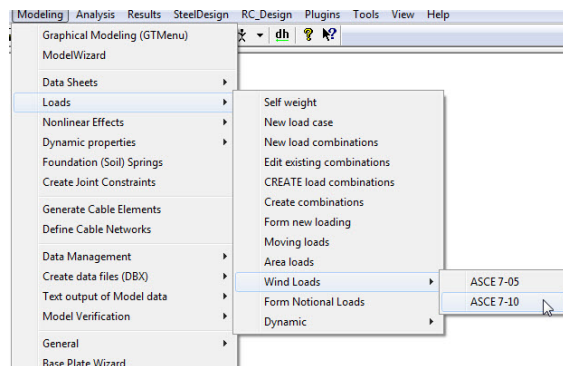
c-t-r-s =

$$\left\{ \begin{array}{l} \text{HCTRS } v_{HCTRS} \text{ DCTRS } v_{DCTRS} \\ \text{DCTRS } v_{DCTRS} \text{ HCTRS } v_{HCTRS} \\ \text{AF } v_{AS} \end{array} \right\} (\text{THICE } v_{THICE}) (\text{THINS } v_{THINS}) - \\
 \text{COMPONENT (ID) 'a}_{CID}$$

miscellaneous/override data =

$$[\underline{KZ}] v_{KZ} [\underline{CF}] v_{CF} [\underline{QZ}] v_{QZ} [\underline{FLD}] v_{FLD}$$

Dialogs are available in the Shell to create the Wind Loads as shown below:



The resulting ASCE 7-05 and 7-10 dialogs have two tabs for General Wind Load Data and Member Wind Load Data as shown below:

A screenshot of the 'Wind Load Generation ASCE 7-10' dialog box. The dialog has a title bar with the text 'Wind Load Generation ASCE 7-10'. Inside, there are two tabs: 'General Wind Load Data' (selected) and 'Member Wind Load Data'. The 'General Wind Load Data' tab contains several input fields and buttons. At the top, there is a 'Wind Load ID:' field and a 'Delete Wind Load' button. Below that is a 'Description:' field. The 'Active Units' section shows 'IN LBS RAD SEC StdMASS' and a 'Change Units' button. The 'Design Wind Speed' section has a 'V:' field with a value of '0' and radio buttons for 'mph' (selected) and 'm/s'. The 'Gust Factor (G)' field has a value of '0.85'. The 'Elevation Axis' section has radio buttons for 'Y' (selected) and 'Z'. The 'Wind Direction Angle' field has a value of '0'. The 'Exposure Category' section has radio buttons for 'B' (selected), 'C', and 'D'. The 'Directionality Factor (Kd)' field has a value of '1'. The 'Topographic Factor' section has a 'Kzt' field with a value of '0' and three fields for 'K1', 'K2', and 'K3', all with values of '0'. The 'Gross Area (Ag)' field has a value of '0'. The 'Added Force Area (AFadd)' field has a value of '0'. At the bottom, there are five buttons: 'Review Data', 'Report Wind Load', 'Save Commands Only', 'Save Load', and 'Exit'. There is also a 'Help' button.

General Wind Load Data Tab

Wind Load ID:

Description:

General Wind Load Data | Member Wind Load Data

Active Units: IN LBS RAD SEC StdMASS

Selected Members:

Wind Load Component Type:

Af and Cf Parameters

☐ Properties ☐ Round D:

☐ Rectangle B: H:

☐ Af

Hctrs: Dctrs: C-T-R-S ID:

THice: THins:

CDg:

Misc. Parameters/Overrides

Kz: Cf:

WP: FLD:

Member Wind Load Data Tab

The new WIND LOAD command is documented in Section 2.1.11.3.9, Volume 1 of the GT STRUDL Reference Manuals.

2. The cable element WIND LOAD and the PRINT WIND LOAD DATA commands from GT STRUDL versions prior to and including 2016 R1 (Reference Manual Volume 3 Revisions prior to and including Revision Z) have been replaced by the CABLE WIND LOAD DATA and PRINT CABLE WIND LOAD DATA commands respectively. The WIND LOAD and PRINT WIND LOAD DATA commands are now used for the new wind load commands previously described.

The revised CABLE WIND LOAD command is documented in Section 2.6.4, Volume 3, of the GT STRUDL Reference Manuals.

- For self-weight loadings which include finite elements, the PRINT LOAD DATA Loading title will now indicate if finite elements are included in the loading as shown below:

```
LOADING - 3          sw3                                STATUS - ACTIVE
Self Weight 3      is in the -Y direction with a load factor of 1.200 including finite elements for ALL ACTIVE MEMBERS
```

- The QUERY command will now output the number of Groups that have been defined in the model as shown in the figure below:

```
-----
**** CURRENT GTSTRUDL PROBLEM STATISTICS ****

ACTIVE UNITS:      FEET      KIP      DEG      DEGF      SEC

INPUT MODE: ADDITIONS          SCAN MODE INITIATED: NO
ACTIVE SOLVER: STANDARD

CURRENT STRUCTURAL TYPE: SPACE FRAME

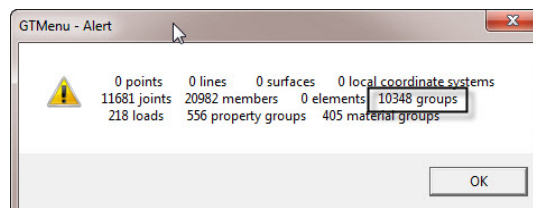
      JOINTS      MEMBERS      ELEMENTS      SUPERELEMENTS
ACTIVE  11681      20982          0              0
INACTIVE  0          0          0              0

GROUPS          10348
RIGID BODIES          0
JOINT TIES          0

      LOADS:      INDEPENDENT      DEPENDENT
ACTIVE          0              0
INACTIVE        0              0
-----
```

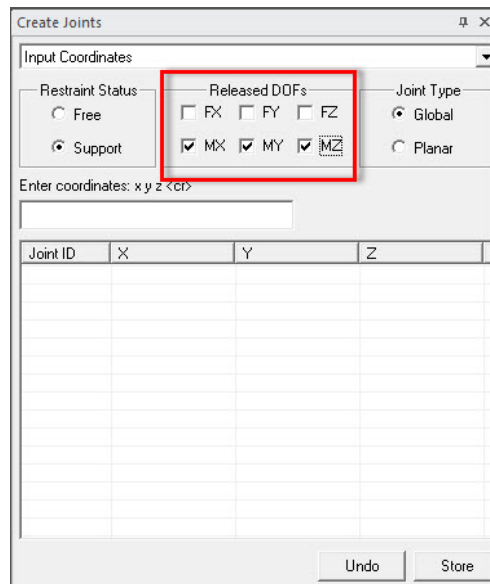
The results of the QUERY command are also automatically output when a model is restored or when returning from GTMenu.

In GTMenu, the Model Statistics output which is available under the Query button and the Check and Help pulldowns will now also show the number of Groups in the model as shown in the figure below:

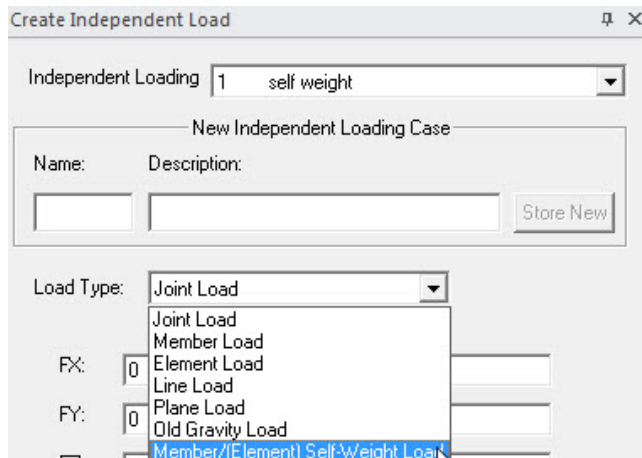


2.3 GTMenu

1. You can now specify Joint Releases when creating Joints. Previously, joints would have to be edited to specify releases. The modified Create Joints dialog is shown below:



2. You are now able to create a self-weight loading which includes finite elements as well as members. Previously, you would have to create a self-weight loading for members and then another self-weight loading for elements. When creating loads, the Load Type pulldown options have changed as shown in the figure below:



After selecting Member/Element Self Weight Load, the dialog shown on the next page appears where you can enter the direction and factor for the self-weight load. The dialog now has a check box to Include Elements which is checked by default. If you want only members to be included in the self-weight loading, remove the check

mark in the check box. If your model doesn't contain any elements, you can leave the box checked.

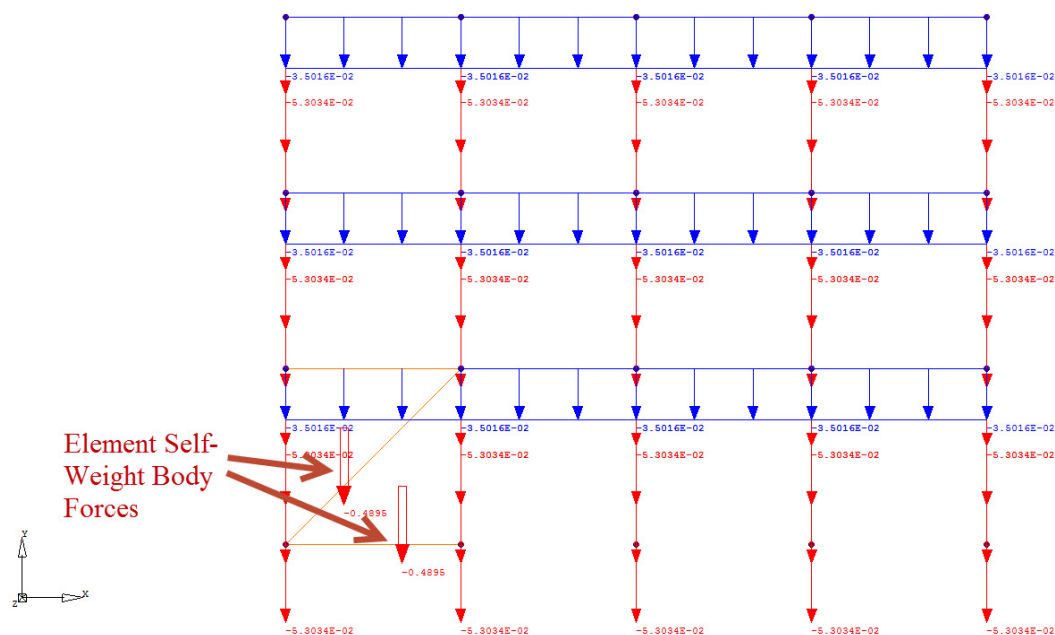
Load Type: Member/(Element) Self-Weight L

Direction: Negative Y [-Y] Factor: 1

☒ Include Elements

Generate Loads

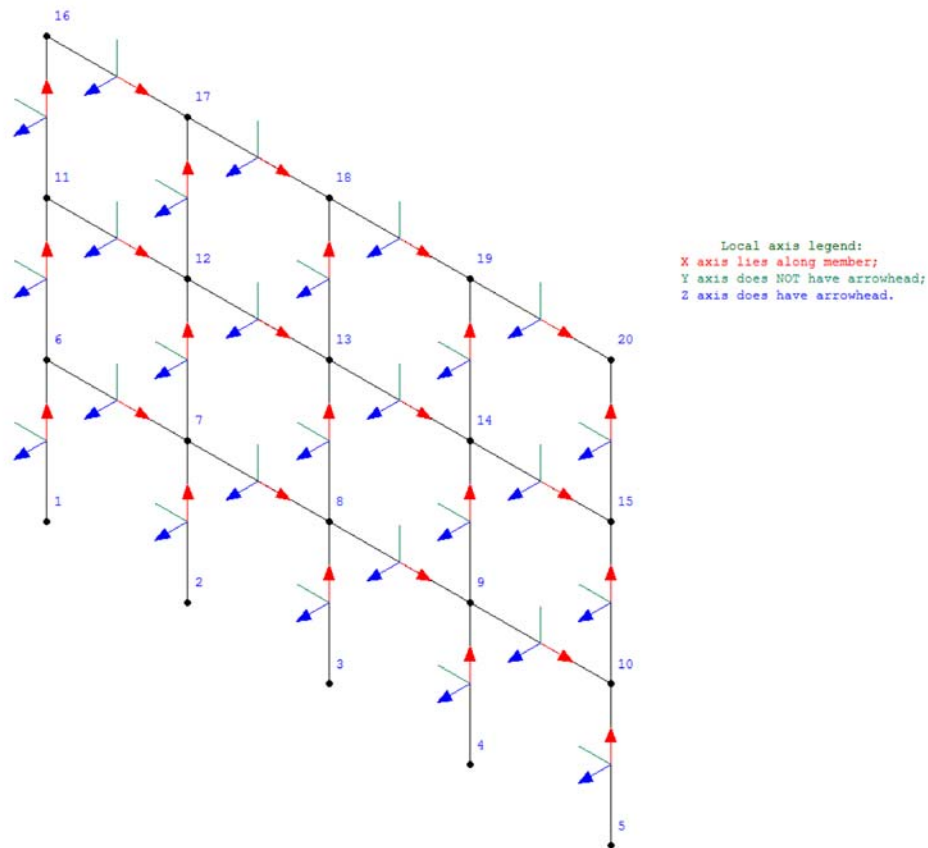
When asked if you would like to display the self weight load, the self-weight load for finite elements will be displayed as a body force as illustrated in the following figure:



- The Display Load button on the Button Bar will also display the self-weight load just as shown above. In addition, a title of the loading indicating that the loading is a self-weight loading will now be output at the top left corner of the display area with the self weight factor. An example of the new loading title and factor output is shown on the line below:

SELF WEIGHT LOAD 2 FACTOR 1.000

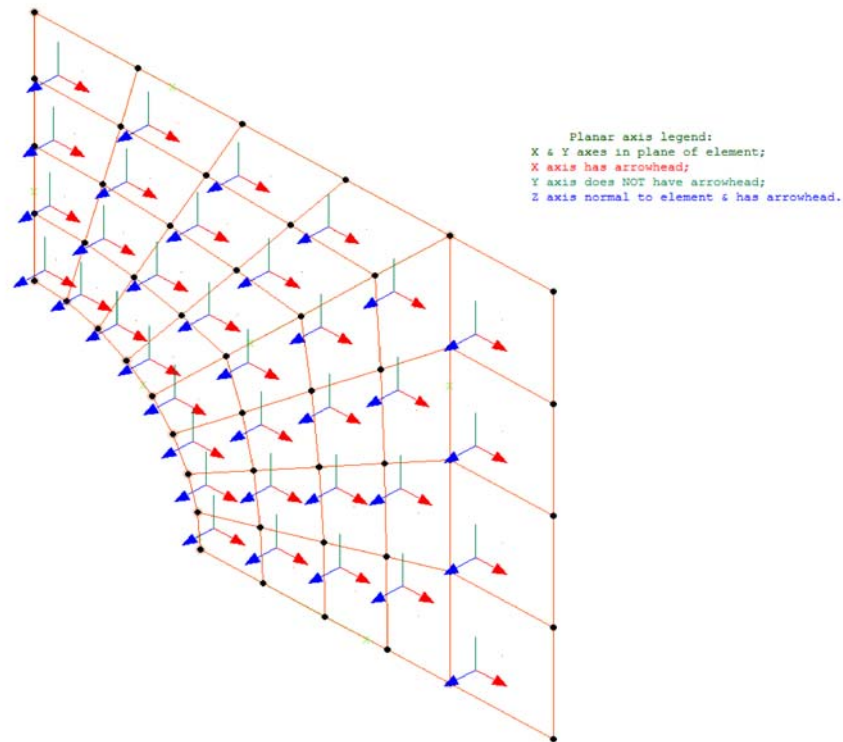
4. The Display Model function to display the local axes of members will now be drawn using highlight colors 1-3 for the local x,y and z axes respectively. By default, highlight colors 1-3 are red, green and blue and can be changed by the user. In addition, the legend will also use these same colors. An example is shown in the figure below:



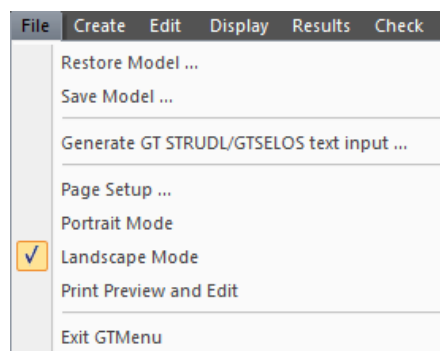
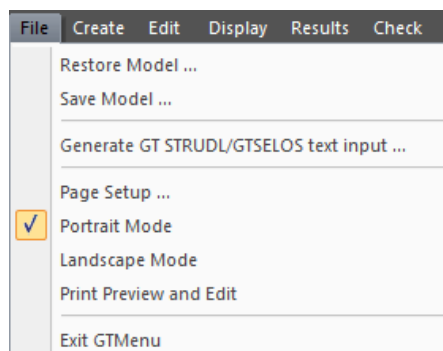
The size of the arrowhead has also been made thinner for many display functions.

The Display Model function for the display of the Beta Profiles will also use the same colors for the display of the local y and z axes of members.

The Display Model function for the display of the planar axes for two-dimensional finite elements will also use these same colors as shown in the figure below:

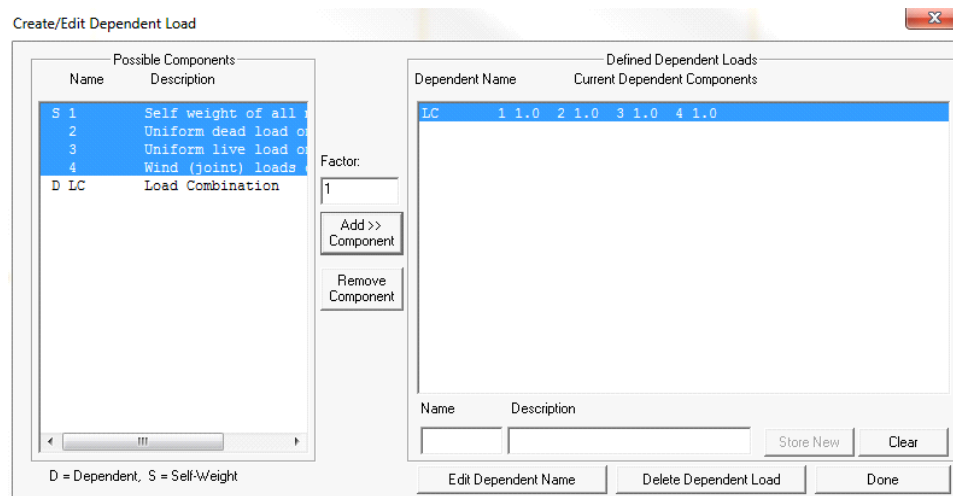


5. The File drop-down menu has been changed so that setting Landscape or Portrait printing orientation mode does not automatically send a print request as in previous versions. Selecting 'Portrait Mode' or 'Landscape Mode' sets the orientation for subsequent printing through the 'Print Preview and Edit' menu selection or the Print icon on the button bar.



6. The Create/Edit Dependent load dialog has been modified to allow multiple selection of components. The multiple components can be selected with the Shift key or the left Ctrl key as defined below:
- Shift - select a group of components that are contiguous by one clicking.
 - left Ctrl - randomly select multiple components that are anywhere on your component list.

An example illustrating the multi-selection of loadings to define a dependent load is shown below.



7. When using the mouse wheel to zoom in on your model, the zoom will occur at the current mouse location. This will allow you to zoom into a particular region of your model much faster without having to zoom and pan multiple times.
8. The Color by Section selection is now retained when rotating, panning or zooming.

9. The Extrude Model Dialog has been improved to allow the extrusion of the model in the negative direction of the global axes as shown in the revised Extrude Model dialog below:

The screenshot shows the 'Extrude Model' dialog box with the following sections and controls:

- RESET Mem/Ele to Extrude** and **Select Mem/Ele to Extrude** buttons at the top.
- ID Assignment** section:
 - ☒ **Default**
 - ☐ **Increment**
 - Joint IDs:
 - Mem/Ele IDs:
- Extrude Control** section:
 - Repetitions:
 - Direction: A group of eight radio buttons for **+X**, **+Y**, **+Z**, **-X**, **-Y**, and **-Z**. The **-X**, **-Y**, and **-Z** options are circled in red.
 - Spacing:
- Additional Options** section:
 - ☐ **Add Members in Opposite Extension**
 - Extension Length:
 - Material:
 - Section:
 - Beta Angle:
 - ☐ **Extrude Selected Joints to Connecting Members (Columns)**
 - Material:
 - Section:
 - Beta Angle:
 - ☒ **All Joints** | **Select Joints** | **Exclude Joints**
 - ☐ **Extrude Selected Members by Adding 2D FE's (Shear Wall)**
 - Material:
 - Element:
 - Thickness:
 - Select Members** button
 - ☐ **Extrude All 2D FE's into 3D FE's**
- Buttons at the bottom: **Generate Extrusion**, **Undo**, and **Store**.
- Footer tabs: **Views**, **Label Settings**, and **Extrude Model**.

10. You can now contour results that are above a certain value or within a certain range. For instance, you might want to contour results that are only above the yield strength of the steel. The revised Element Contours dialog is shown below with the new panel to set the Limits for the contours:

Element Contours

☐ Principal

☐ Stresses

☐ Strains

☐ Membrane Resultants

☐ Bending Resultants

☐ Displacements/Rotations[Load/Mode]

☒ Von Mises

☐ Top ☒ Middle ☐ Bottom

Compare with ☒ Yield ☐ Yield/Sqrt3

☒ Single Load ☐ Envelope

Select Load: 1

☒ Solid Colors ☐ Auto Display

☒ with Outline

☐ Lines Only Contour Interval 2.5

Limits

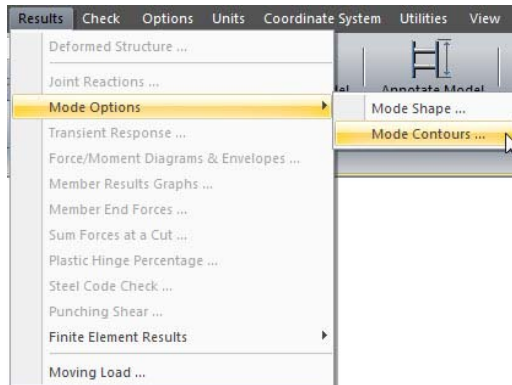
Minimum 32 ☐ Draw Contour Outside the Limits

Maximum 66

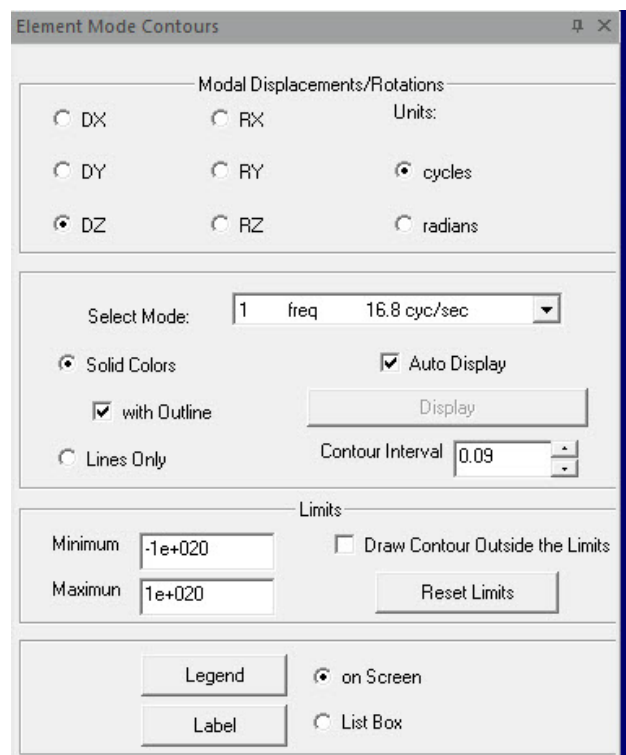
☒ on Screen ☐ List Box

Using the Limits options, you can draw contours within the limits or outside the limits.

11. A new Dialog has been added to contour dynamic mode shapes for models which contain finite elements by selecting Results - Mode Options - Mode Contours as shown below:



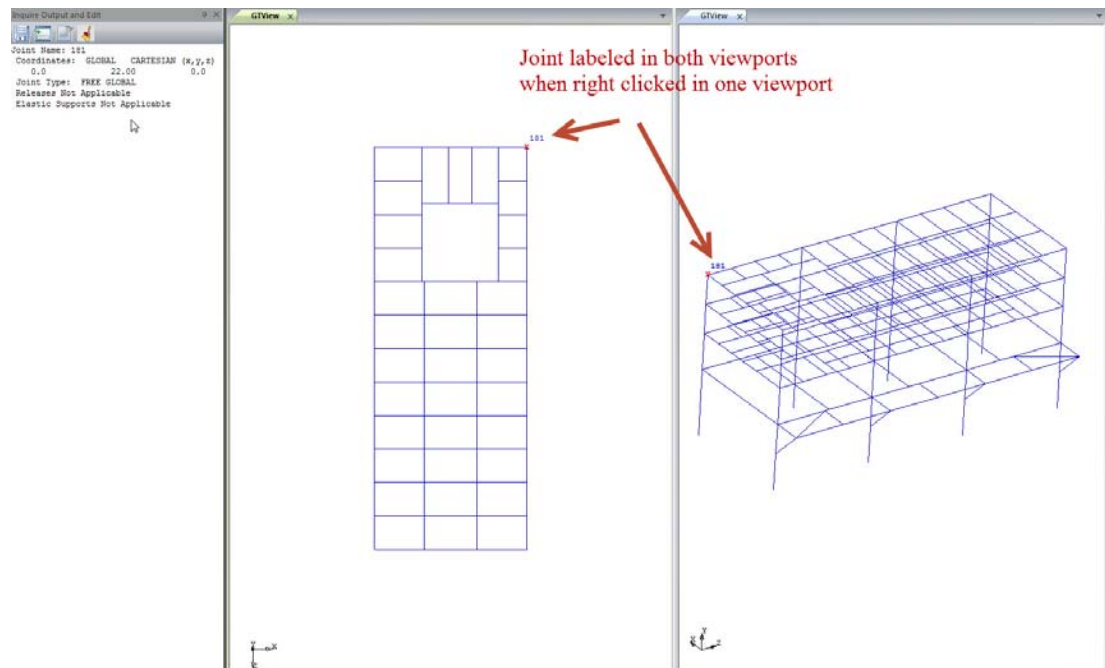
The new Element Mode Contours dialog is shown below:



If static element stresses exist, you may also contour the mode shapes using the Element Contours dialog.

12. Now, when you have two viewports displayed and you right click on a joint, member or element in one of the viewports, the joint, member or element label will be displayed in the other viewport if it is visible. This is particularly useful when you right click in a view which contains only a portion of the model such as a floor and you want to see where that joint, member or element is located in the other viewport.

An example of this is shown below where you have right clicked on a joint in the left viewport which contains only the top floor and you can see the joint also labeled in the right viewport which displays the entire structure.



13. Surface Definition data is now written to the input file created by GTMenu. A new command has been implemented to read the Surface Definition data. The new GTMenu Surface Definition command is described in Section 5.4.4 of this Release Guide.
14. A new dialog has been created that you may use when using the Global Plane option on the Mode Bar. The new dialog is shown below:



- To use the Global Plane option on the Mode Bar, select Plane - Unbounded - Global Plane. The Global Plane option is useful when creating supports or editing joint or member data.
15. The Inquire Output Area has been renamed to Inquire Output and Edit to emphasize the ability to Edit items in the Area.

2.4 Finite Elements

1. The CALCULATE ERROR ESTIMATE command using the MAX DIFFERENCE, DIFFERENCE FROM AVERAGE, PERCENT MAX DIFFERENCE, PERCENT DIFFERENCE FROM AVERAGE, NORMALIZED PERCENT MAX DIFFERENCE or NORMALIZED PERCENT DIFFERENCE FROM AVERAGE options as well as the contouring of error estimates using these options is now significantly faster. An example of the improved computational speed is shown below:

Model Description:

15,000 SBHQ6 elements

1 loading

The time for the CALCULATE ERROR ESTIMATE command with all of the options shown above:

Version 2016 R1 118 seconds

Version 2016 R2 5 seconds

The CALCULATE ERROR ESTIMATE command as well as the contouring of error estimates remains a prerelease feature as described in Section 5.3.1 of this Release Guide.

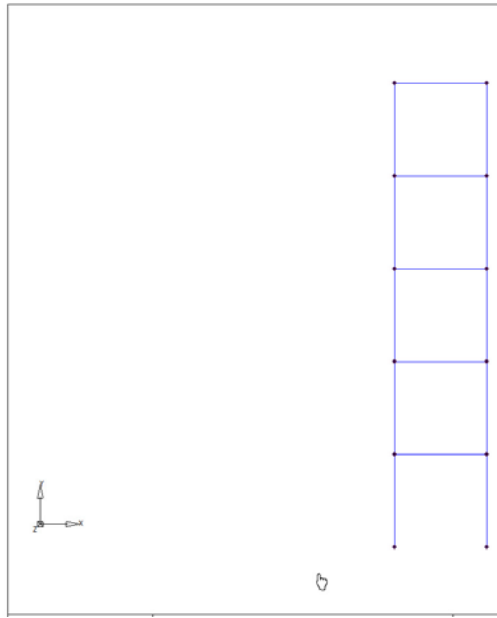
2.5 Base Plate

1. A new “**Include elements in GROUP definition**” option has been added to Constraints. If this option is checked, the base plate finite elements incident to the constraint and on one side of the constraint are added into the GROUP definition for the constraint. If this option is selected, you can choose to add LIST SUM FORCES to the generated results.

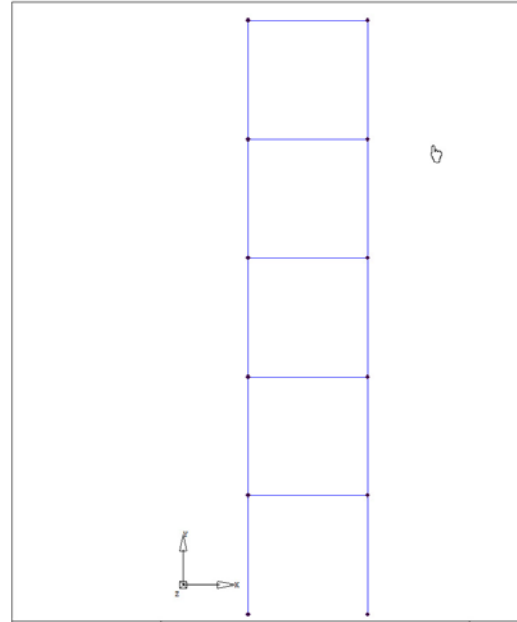
The purpose of this new feature is to allow you to create a “free body” cut across the base plate to facilitate an external check of plate stresses when attachment endpoints or plate notches cause artificially high “hot spot” stresses.

2.6 Scope Editor

1. When printing or previewing images from GTMenu, the Scope Editor will now move the global axes drawn in the lower, left corner (the default position) next to the structure image before scaling and centering the image. This will result in images that fit on the page better as in the image on the right below. No change is made when the global axes are drawn on the origin.

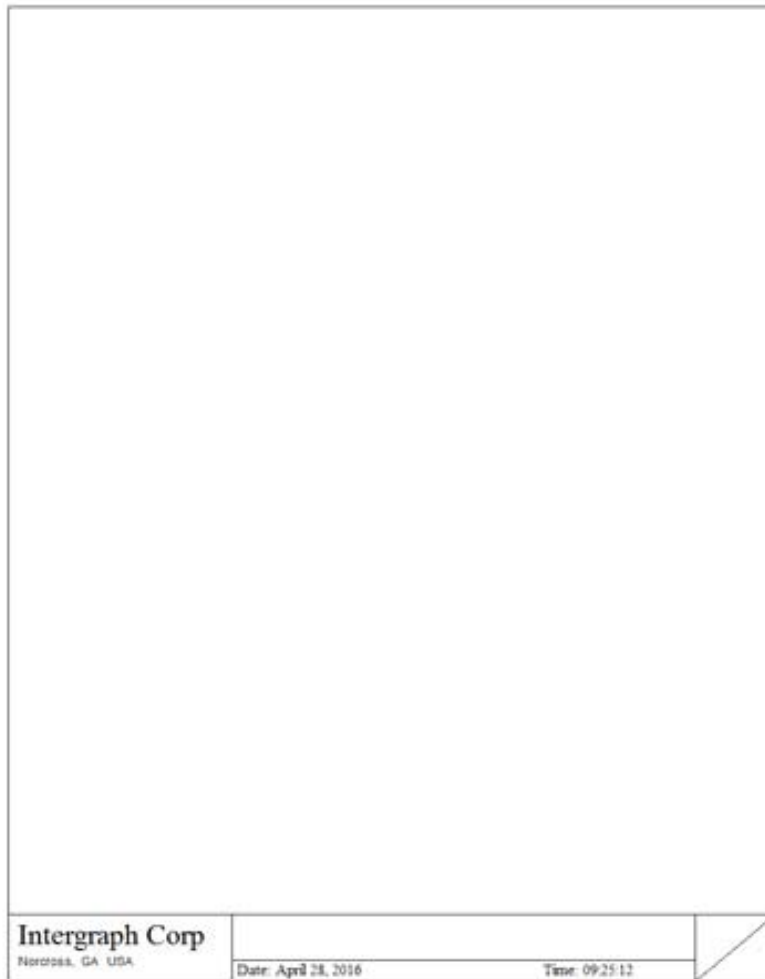


Previous Versions



Version 2016 R2

2. The Scope Editor now has a default template as shown below. This template is automatically applied when Print Preview and Edit is selected in GTMenu.



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. The original database is no longer overwritten when you make modifications to the DWG prior to using the Save As command.

3.2 DBX

1. Using a specified file name without the UNREGISTERED option no longer causes an abort if the DBX FILE SPECS command has not been given.
(No GPRF issued)

3.3 File → Export → CIS/2

1. The File → Export CIS/2 feature now correctly exports MEMBER ECCENTRICITIES.
(No GPRF issued)

3.4 Finite Elements

1. The CALCULATE ERROR ESTIMATE command will no longer abort for large models.
(No GPRF issued as this was and still is a Pre-Release feature as described in Chapter 5 of this Release Guide)

3.5 GTMenu

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

1. The Modes Bar default options have been corrected for a number of dialogs.
2. Materials and Properties Lists selections are now updated when a new Material or Property has been added to the list.
3. An abort will no longer occur when there are Variable Section properties in the Model and Color By Section is used.
4. The edges of finite elements are now displayed when the Contour Line option is used to contour stresses.
5. Newly created objects such as construction points and lines, joints, members and elements are now automatically stored correctly when dialogs are closed.
6. When generating members and elements using surfaces, the U and V direction labels are now displayed.
7. Previously when performing split member and then storing, the member selection on the Mode Bar had to be reset to function properly. For instance, if All was selected on the Model Bar and all the members split and the split stored, a subsequent split on All members did not split all members. This problem has been corrected.
8. Aborts will no longer occur when Material or Property dialogs are invoked while performing selections in another dialog such as the Place Members dialog.
9. Self-weight loads are no longer allowed to be edited in the Edit Loads dialog except that you can change the description of the self-weight load or delete it.
10. The legends for Color by Section, Rigid Bodies and Superelements no longer truncates at the bottom when the number of labels exceed the vertical space selected by the position box. The popup dialog asking to place the next legend column has also been removed.
11. Fonts are now saved so that Scope Editor displays the same size, position and font name as used in GTMenu.

12. When saving forces to a file using the Sum Forces at Cut dialog, the file name is no longer truncated to 32 characters.
13. A spurious line will no longer appear when displaying the deformed structure when construction lines are visible in the model.
14. The position and symbol size for legends for sections, materials and supports have been moved closer to the lower right corner of the display area to reduce the possibility that the symbols are written on top of the structure.
15. In versions 2016 and 2016 R1, the directions of the self-weight load were reversed for the plus and minus (+/-) Z directions. This has been corrected for the enhanced self-weight feature described in Chapter 2.
16. If one or more elements of the element types shown below is deleted in GTMenu, an abort will no longer occur during the stress backsubstitution phase of STIFFNESS ANALYSIS. This abort would not occur if the elements were deleted using commands.

Element types: IPQL,IPQLQ1,IPQLQ2,IPQLQ3, IPQLQ4, IPSL, IPSQ, TRANS3D

17. In versions 2016 and 2016 R1, the load name was not displayed when using Display Load. The load name was being written in the graphics background color and was only visible when printing or when you did Print Preview and Edit using the Scope Editor. This problem has been corrected so the load name is now visible when using Display Load.
18. The input boxes in all of the dialogs now accept strings larger than the width of the input box.
19. An abort no longer occurs in the Edit IDs Dialog using the Global Plane option.
20. A memory conflict no longer occurs when switching between the Display Load dialog and other dialogs such as Create Joints or Points.
21. Previously if no Groups existed in the model and a new group was to be created, the Members button was grayed out and members could not be added to the group. This problem has been corrected.

22. You can now have a value of 0.0 for the starting load when using the Line Load option of the Create Loads Dialog.

3.6 GTShell (GT STRUDL Output Window)

1. Changing the font in the Joint Displacements datasheet no longer causes an abort.
(No GPRF issued)

3.7 SPLM Licensing

1. SPLM licensing now works for the Offshore commands in GTSTRUDL.
(No GPRF issued)

3.8 Steel Design

1. GT STRUDL will no longer print multiple error messages and subsequently abort during a code check using the 69AISC and 78AISC steel design codes for Pipe and Solid Round Bar cross-sections if the axial compression stress is larger than the Euler stress.

(GPRF 2016.03)

Documentation:

The 69AISC and 78AISC codes are in the Volume 2B of the Reference Manual.

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 2016 R2. 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, MYY, and MXY are resolved on a per node basis along the cut, and either the average effect or the maximum effect on the cut is applied to the entire cross-section.

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

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

The user may also specify which layer of bars to be designed, using the modifier INNER or OUTER. These refer to the location of reinforcing bars on each surface. At most slab locations, reinforcement is placed in two perpendicular directions

on both surfaces of the slab. Since each layer of reinforcement cannot occupy the same space, one layer must be placed on top of the other. OUTER refers to the layer closest to the surface, while INNER refers to the layer nearest the center of the slab.

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

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

Requirements:

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

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

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

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

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

Usage:

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

Usage Example: Description of Example Structure

The example structure is a rectangular slab system, shown in Figure 5.2.3-1. The clear span of the structure is thirty feet, and the slab strip has a width of ten feet. The two ends of the slab are fully fixed, while the thirty foot sides are free, resembling a fixed-fixed beam. The slab is one foot thick and constructed of normal strength concrete with FCP = 4000 psi. The example structure can be idealized as a subset of a larger slab system, perhaps the design strip running between two column faces in an interior region. The structure is loaded with a distributed surface pressure of 150 psf over the entire surface of the slab.

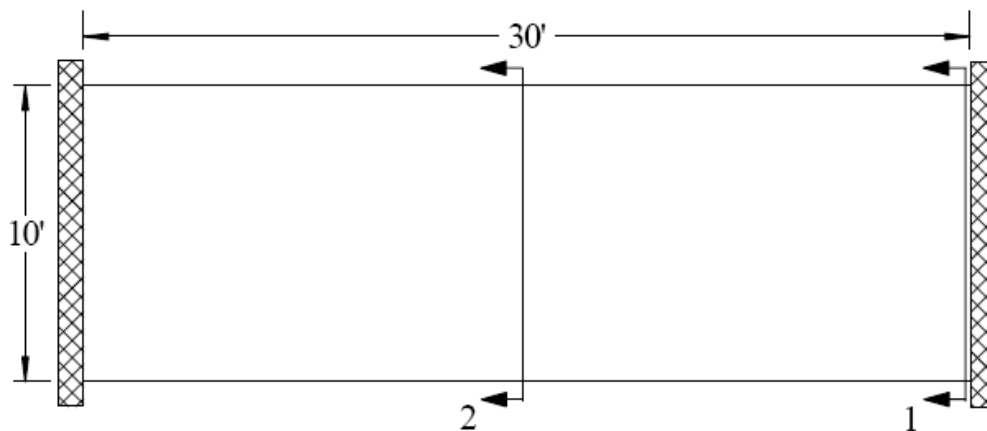


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

GT STRUDL Finite Element Model

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

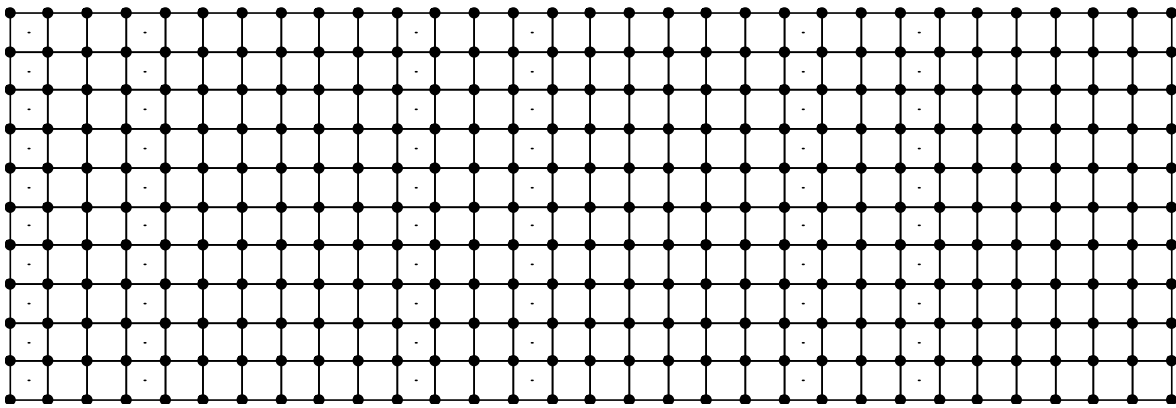


Figure 5.2.3-2 Example Finite Element Model

Definition of Cut Cross-Sections

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

Cut 1-1:

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

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

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

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

The GTSTRUDL output for this example is as follows:

```

** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **
  PROBLEM - VFE103      TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS

  RELEVANT ACTIVE UNITS:  INCH LB

  NUMBER OF ACTIVE LOADINGS:      1

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

** ELEMENT FORCE IMPLEMENTATION **

** DESIGN MOMENT ENVELOPE **

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

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

** SLAB CROSS-SECTION **

  Width      Depth      FCP      FY      Cover      Layer
  -----
  120.00      12.00      4000.00    60000.00    0.750      Inner

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

  Face      Bar      Spacing  AS PROV'D  MOMENT STRENGTH  MOMENT REQ'D  STATUS
  -----
  TOP      # 5      13.000    2.862      1561006.4280    1354381.4844  PASSES
  BOTTOM                                ( Reinforcement Not Required )

```

Cut 2-2:

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

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

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

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

The 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:

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

Elements:

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

r_1, r_2, r_3 = values in current angle units of the load component rotation angles $\theta_1, \theta_2, \theta_3$ 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