

GT STRUDL® Version 2016 Release Guide



Release Guide

Release Date: October 2015



Notice

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

Copyright

Copyright © 2015 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- Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States.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Unpublished - rights reserved under the copyright laws of the United States.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
Huntsville, 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/complianceandenforcement/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	
2.1 GTMenu	2-1
2.2 CAD Modeler	2-9
2.3 DBX	2-23
2.4 Efficiency Improvements	2-24
2.5 Offshore	2-25
2.6 Steel Design	2-26
2.7 Export CIS/2 File to Interface to Other Intergraph Products	2-27
CHAPTER 3 ERROR CORRECTIONS	
3.1 Base Plate Wizard	3-1
3.2 CAD Modeler	3-1
3.3 DBX	3-3
3.4 Dynamic Analysis	3-3
3.5 Finite Elements	3-4
3.6 General	3-5
3.7 GTMenu	3-5
3.8 Nonlinear	3-6
3.9 Steel Design	3-6

CHAPTER 4 KNOWN DEFICIENCIES

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

CHAPTER 5 PRERELEASE FEATURES

5.1	Introduction	5.1-1
5.2	Design Prerelease Features	5.2-1
5.2.1	ACI Code 318-99	5.2-1
5.2.2	Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)	5.2-4
5.2.3	ASCE4805 Code for the Design of Steel Transmission Pole Structures	5.2-14
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

This page intentionally left blank.

Chapter 1

Introduction

Version 2016 covers GT STRUDL operating on PC's under the Windows 8 and Windows 7 operating systems. For users who are accustomed to our older version numbering system, the version is internally known as Version 35.0.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2015. In particular, Chapter 2 briefly describes new features including the following significant new features:

- New GTMenu user interface with significantly improved usability and graphical display and the ability to easily use the mouse for zooming and rotation of the model.
- Enhancements to the AutoCAD based CAD Modeler (first introduced in Version 2015) including the ability to specify area loads and query the database using member filters as well as improved result display.
- Seismic provisions from ANSI/AISC 341-10 have been added to the AISC14 steel design code.
- Offshore fatigue S-N curves have been enhanced and now may be assigned to brace/chord weld sides with a single fatigue analysis covering all applicable S-N curves.

Chapter 3 provides you with details regarding error corrections that have been made since the Version 2015 release. Chapter 4 describes known problems with Version 2016. 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 ACI Concrete Code 318-99. Rather than use the 318-99 code which is a prerelease feature, users should use the 318-05 code which is a released feature.

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

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

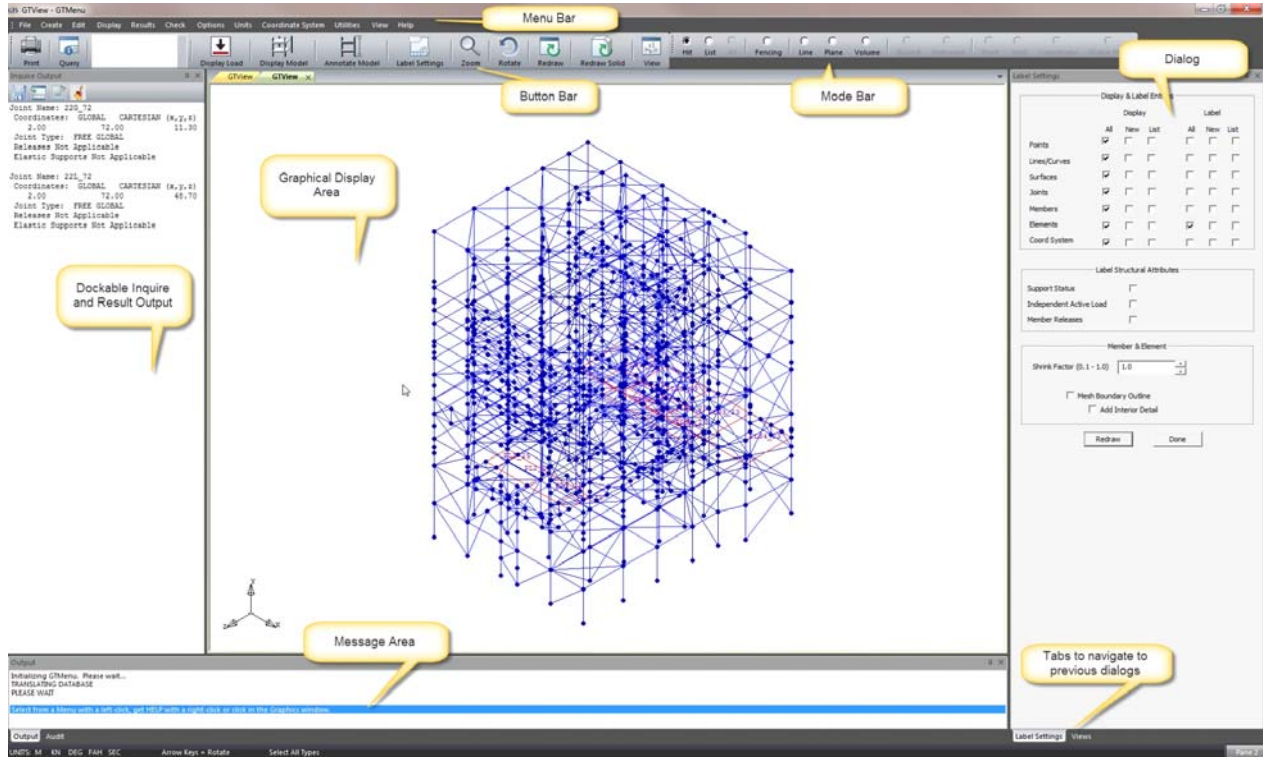
This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GT STRUDL in Version 2016. This release guide is also available online upon execution of GT STRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 GTMenu

A new GTMenu user interface is now available with the following significant new features:

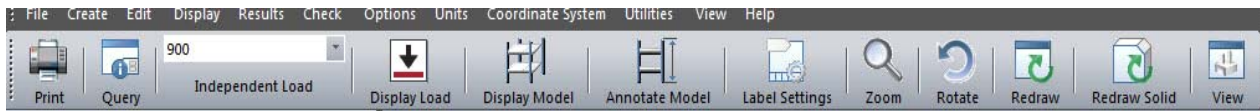
- New appearance
- Dockable dialogs
- Simplified work flow and procedures
- Significantly faster graphical display
- Greatly improved model manipulation with the mouse
- Two views of the model in the graphical display area

The new GTMenu user interface layout is shown below.



New Desktop Layout

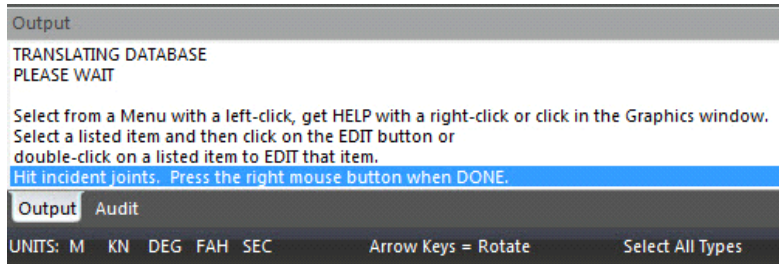
The components of the new interface are shown below:



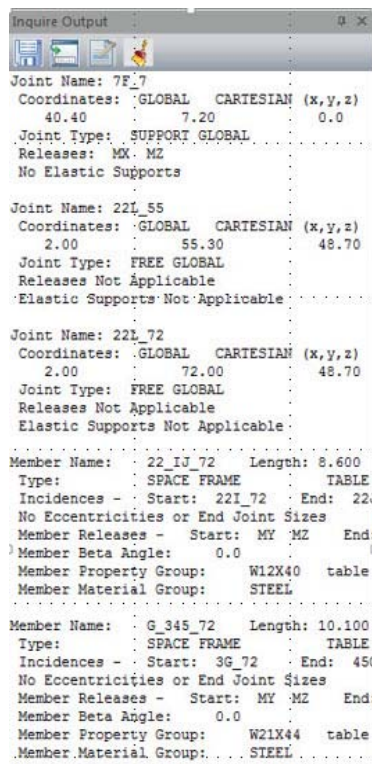
New Menu Bar and Button Bar which are toolbars that can be moved to another location or docked at the top of the Desktop.



The Mode (Selection) Bar is a toolbar which can be moved to other locations on the Desktop

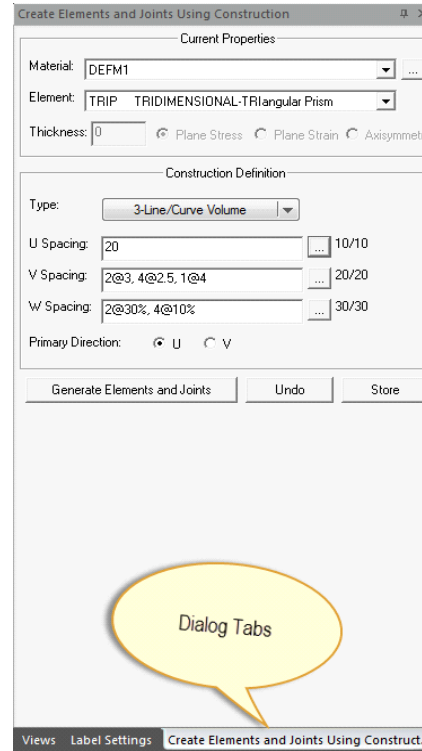
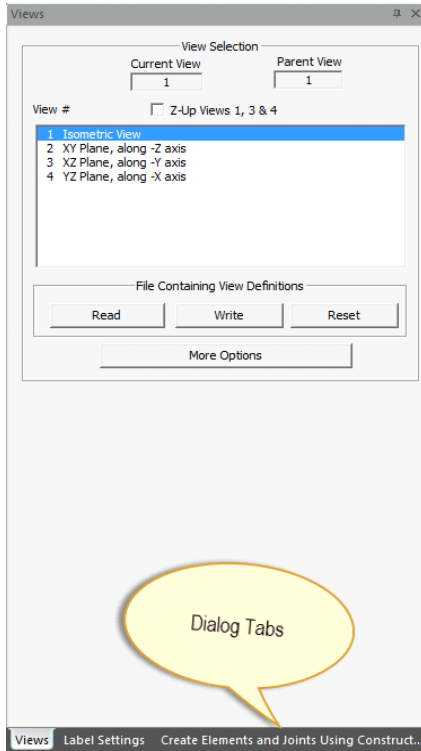


Dockable Message Output Area which is now located at the Bottom of the Screen by default



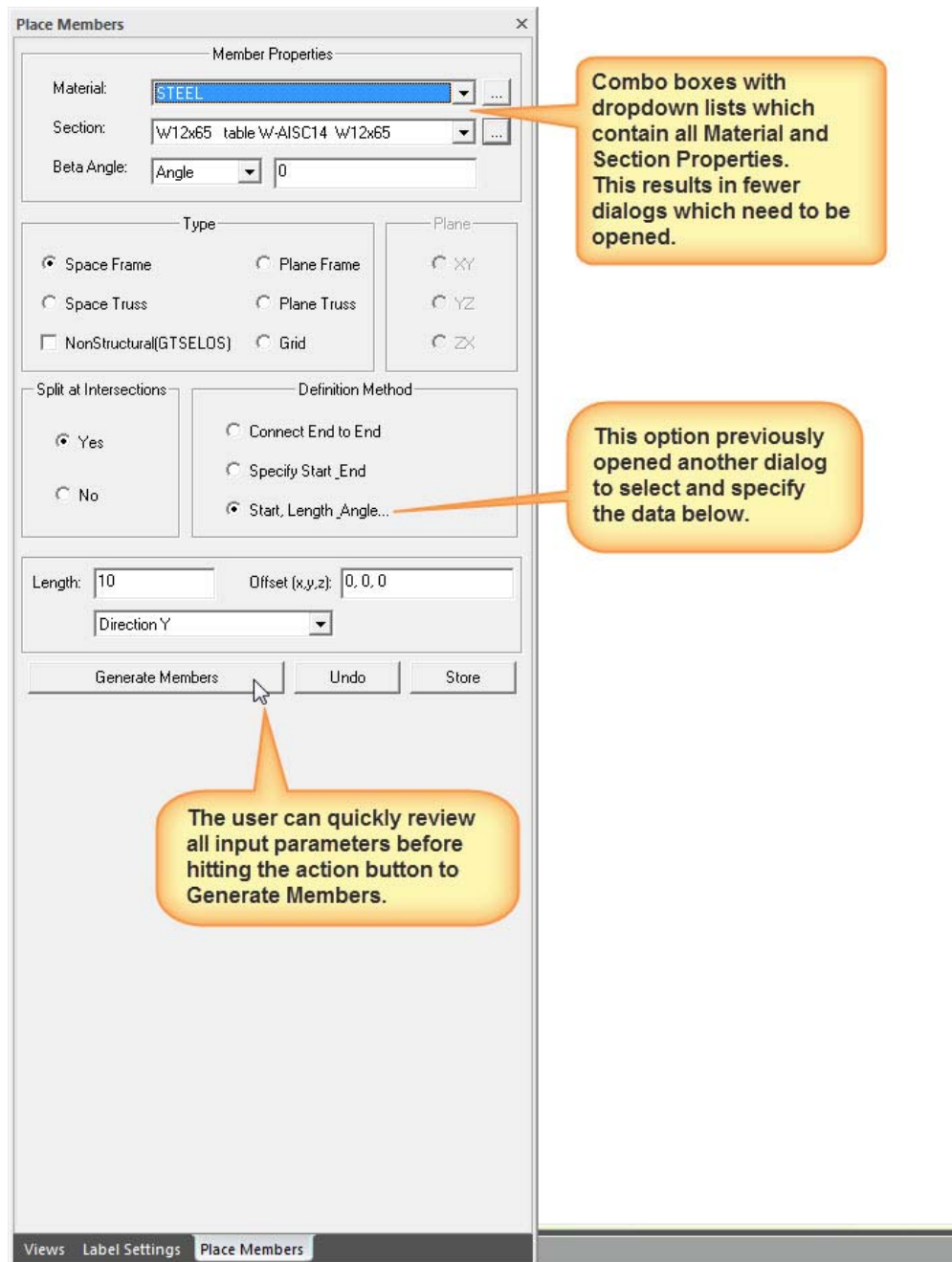
Dockable Inquire and Result Output Dialog no longer covers other open dialogs

Dialogs are placed in a Dockable Frame with Tabs which allow the user to more easily navigate to open dialogs as shown in the figures below:



The Mode Bar, Message Output Area, Inquire Output as well as the dialogs which are popped on the screen are all initially docked on the GTMenu Desktop. You can undock them by grabbing the top of the dialog and moving them to another location on the Desktop or to another monitor. To redock them, grab the dialog at the top and move it on the screen. You will see arrows appear on the top, left, right and bottom of the screen as well as a group of four arrows at the center of the screen. When you move the mouse over an arrow while still grabbing the dialog, the docked location will be shown in a different color. You can also redock them by simply double clicking on the top of the dialog and they will be docked in their last location.

Fewer dialogs are used to accomplish many of the same tasks. As a result, fewer dialogs are popped on top of each other making it easier to understand and specify the data needed. The new Place Members dialog is shown below:



GTMenu can now render large models (Redraw Solid) much faster than in previous versions.

Zoom, Pan and Rotation can be performed using the mouse directly without having to go through several menu picks:

- Zoom: The mouse wheel can be used to zoom in and out.

- Pan: The view can be panned by holding the mouse wheel down and moving the mouse to pan.

- Rotate Rotation of the model can be done by holding the right mouse button down and moving the mouse.

Zoom, Pan and Rotate are very fluid with little or no flickering between renders. This makes it much easier to rotate to a preferred view.

Result contour displays can now be rotated.

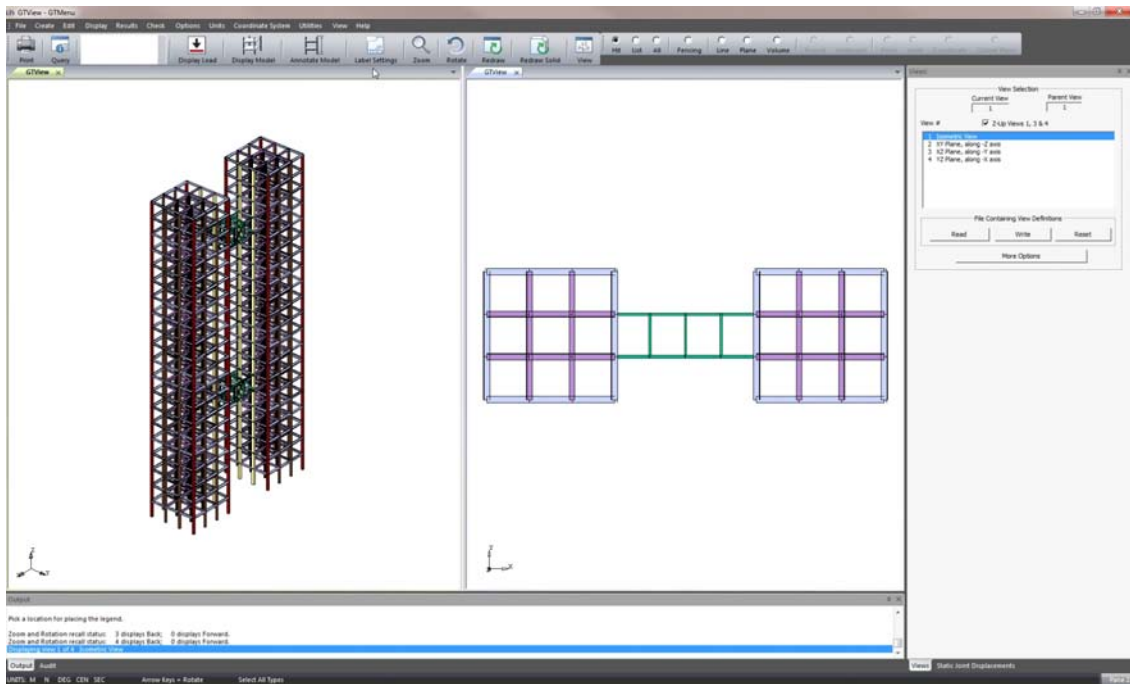
Other result displays such as diagrams and deformed shapes can also be rotated.

You can now display two views of the model side by side (two viewports). At the top of the Graphics Windows there are two tabs called GTView as shown below with the current display shown in the tab with GTView in bold:

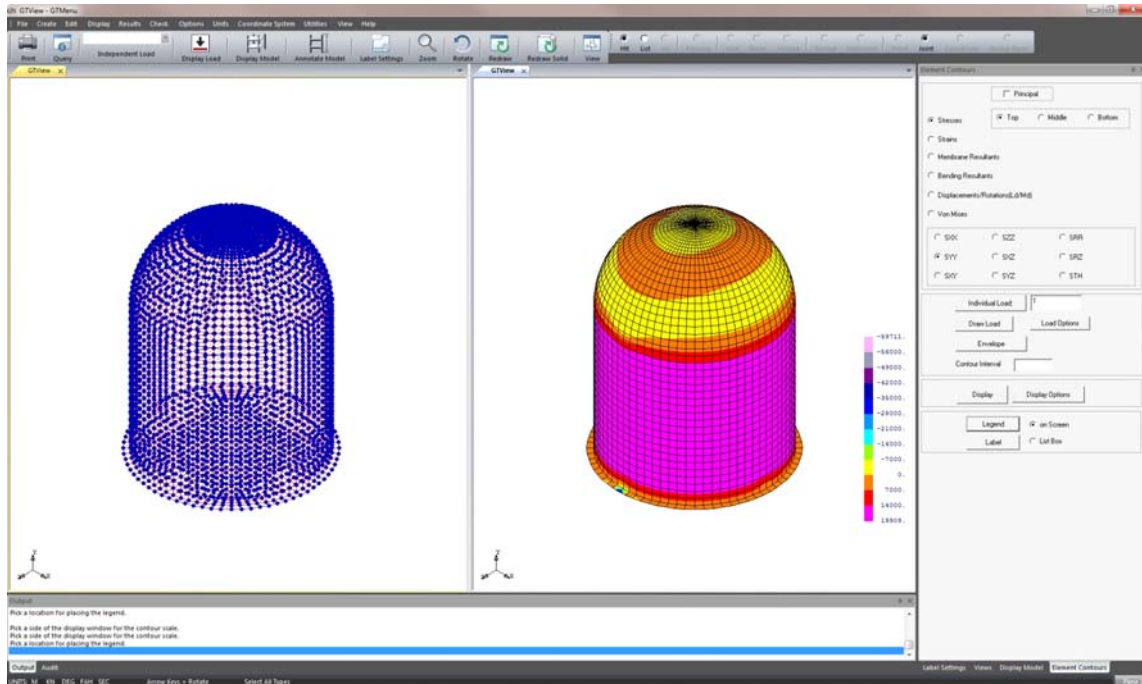


To display both viewports beside each other, select either GTView tab and while holding down the left mouse button, move the mouse to the left or right side of the Graphics Window and then release the mouse. You will then have the two viewports side by side with the same image of the structure. You can then switch between the two viewports by simply clicking in the Graphics Window of the viewport where the active viewport has **GTView** in bold.

Each viewport can display a different View or rotation of the structure, display different Display and Label Settings and also display different results as shown in the examples on the next page.



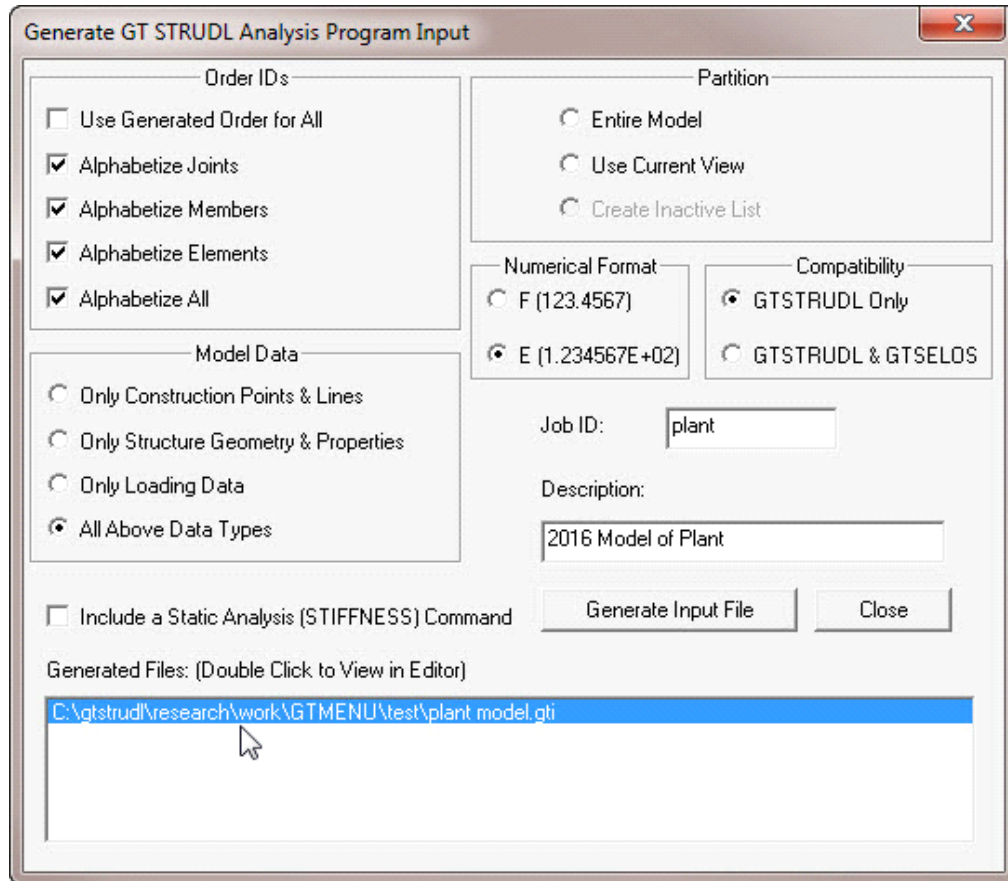
Isometric and plan view of the model



Wire frame stress contour of the model

To return to a single viewport, select either GTView tab with the mouse and move it to the side of the other tab. Currently, the two viewports are not retained after exiting GTMenu.

The dialog to create the text input file (.gti file) has been enhanced where you can now easily open the generated file in an editor by double clicking on it. The new dialog is shown below:



When generating an input file, the SELF WEIGHT command in the generated input file will now have INCLUDE FINITE ELEMENTS in the .gti file if this option was in the original input file.

Users are encouraged to go through the examples in the GTMenu Guide. For best results, it is recommended that you have a monitor with at least 1600 x 1050 resolution.

2.2 CAD Modeler

GT STRUDL Version 2015 was the first version that included the AutoCAD based graphical modeling add-on to AutoCAD. A number of new and enhanced features have been added in Version 2016 as summarized below:

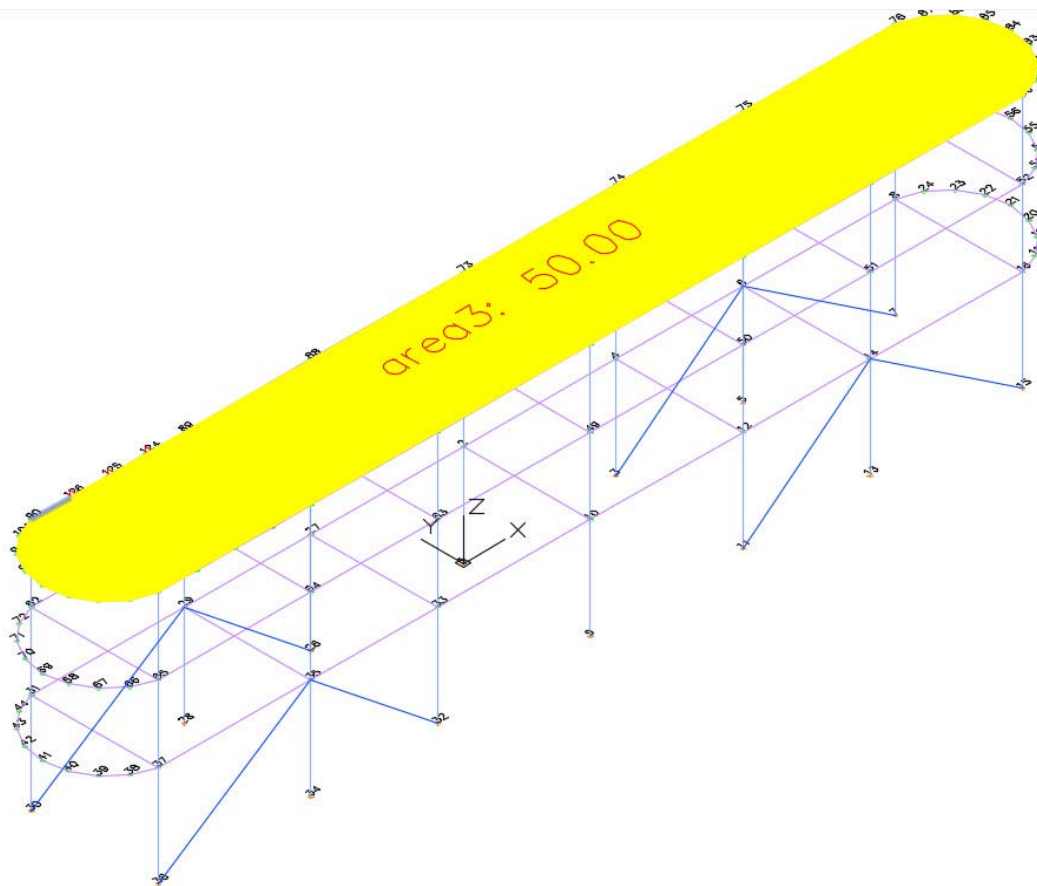
- Area loads can be created by graphically defining the outline of a region to be loaded (graphical selection of members forming a closed area) including the ability to exclude areas within the loaded region such as an opening or an area with a different area load. You may also select members to be ignored inside the area such as bracing members.
- The ability to filter members according to their properties, loading and member forces
- Multiple grids are now available to facilitate the placement of members and grids can now be displayed at any level.
- You can now append other gti files or macros to the input file created by CAD Modeler.
- A new Ribbon Tab called “GTS Display” has been introduced for all Display Functions.
- New dialogs for element results and member diagrams
- New Display Options which allow you to apply a shrink factor to members and elements and to disable displaying finite element thicknesses
- The Local Axes for members, the Planar Axes for elements, and Restrained Degrees of Freedom for Support Joints may now be displayed.
- AutoCAD 2016 is now supported.

The new Area Load dialog is shown below:

The screenshot shows the 'Area Load' dialog box with the following settings:

- Generate:** Name: area3, Description: top floor area load. Buttons: Create New, Save / Modify, Delete.
- Load - Direction:** Load Value: 50.0000. Global Direction Perpendicular to the Loading Plane: X (unselected), Y (unselected), Z (selected). Plane Tolerance: 0.1667.
- Elevation:** Plane Perpendicular at: Value (coordinate) (unselected) 0.0000, Joint (selected) 73.
- Distribution:** Two way (selected), X (unselected), One way (unselected), Y (unselected), Custom (unselected) X: 0.0000, Y: 0.0000.
- Advanced Features:** Define Outline Region >> (Reset), Exclude Area >> (Reset), Ignore Members >> (Reset).
- Bottom Buttons:** Display >>, Clear, OK, Cancel.

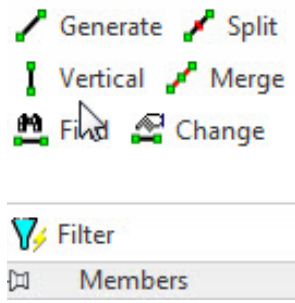
With this new dialog, you specify the name of the loading, the global direction of the load, either the coordinate of the global plane or a joint in the global plane and one-way or two-way distribution of the load. If no other data is specified, the entire plane is loaded. If you want to create a load on only part of the plane, you may define the outline of a region completely enclosed by members. You may define areas to exclude within the region to account for openings and you may also specify members to ignore such as bracing members. Once the data is specified, you may select the Display button at the bottom of the dialog to have the loaded area shown. An example illustrating applying the load to an entire floor is shown on the next page.



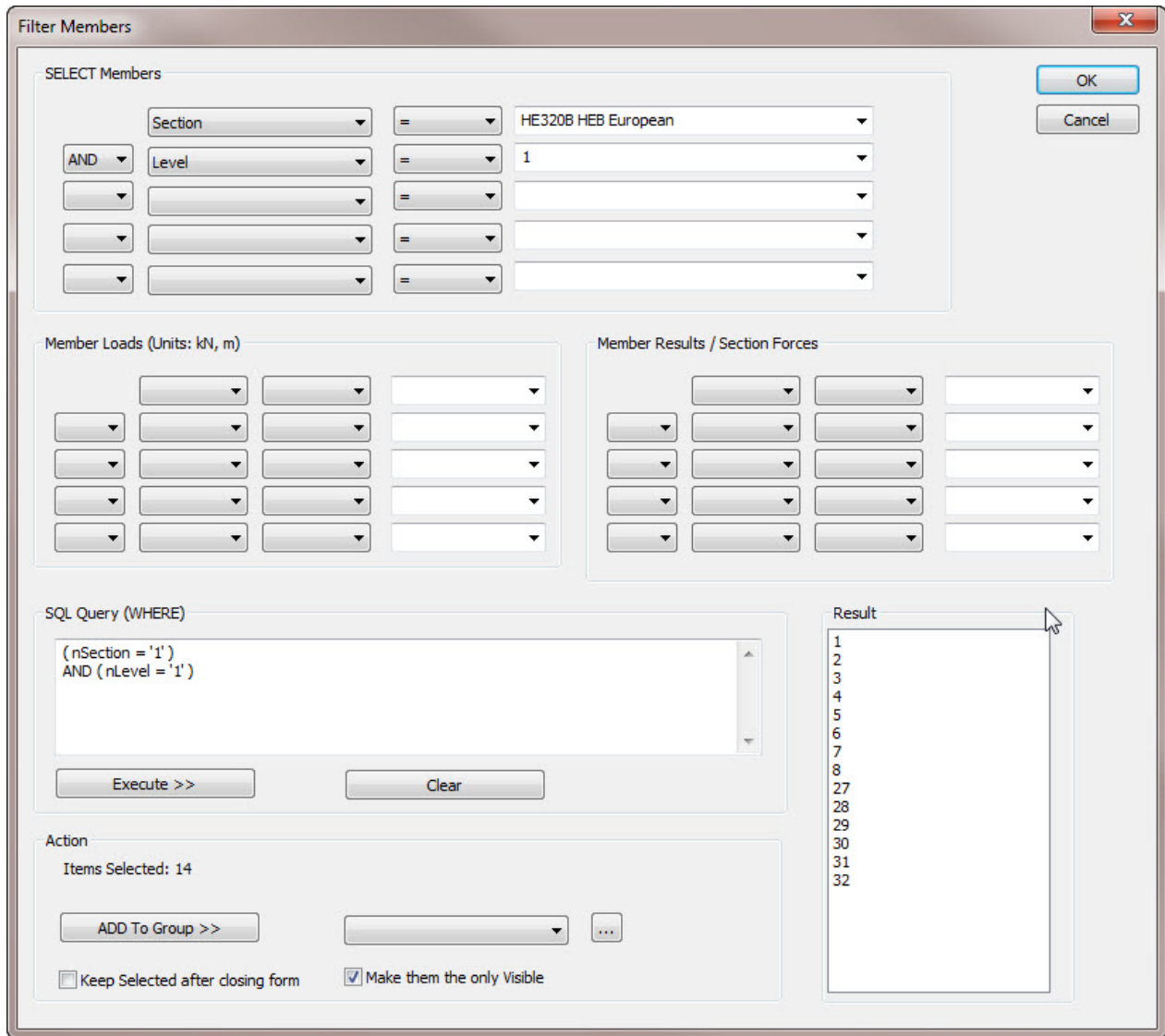
In the figure shown above, area3 is the name of the loading and 50 is the magnitude of the load. When the gti file is created in CAD Modeler, a GT STRUDL Area Loading command will be created.

A new Filter dialog has been implemented which allows you to filter member properties, loadings, and forces. In addition, you have the option to automatically select the filtered members, store them in a group, or make them the only visible members. Selected members can then be edited, moved, copied, etc.

The new Filter dialog is available in the Members panel as shown on the next page:

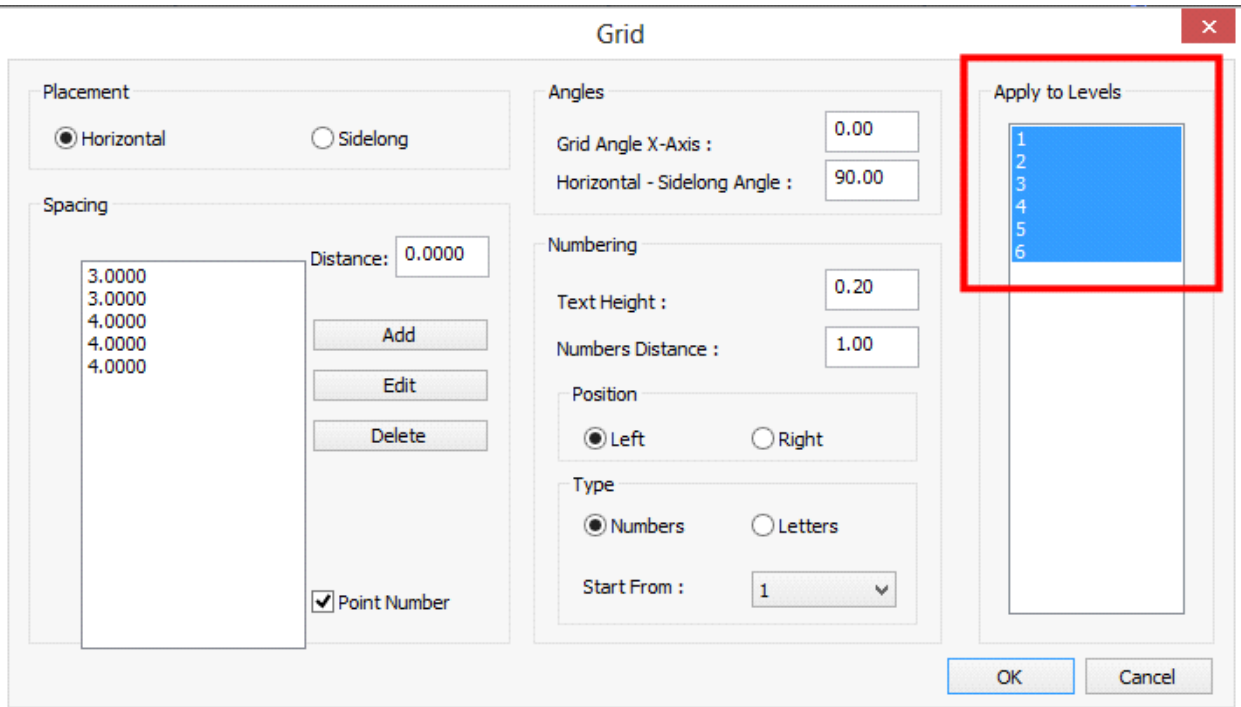


The new Filter Members dialog is shown below:

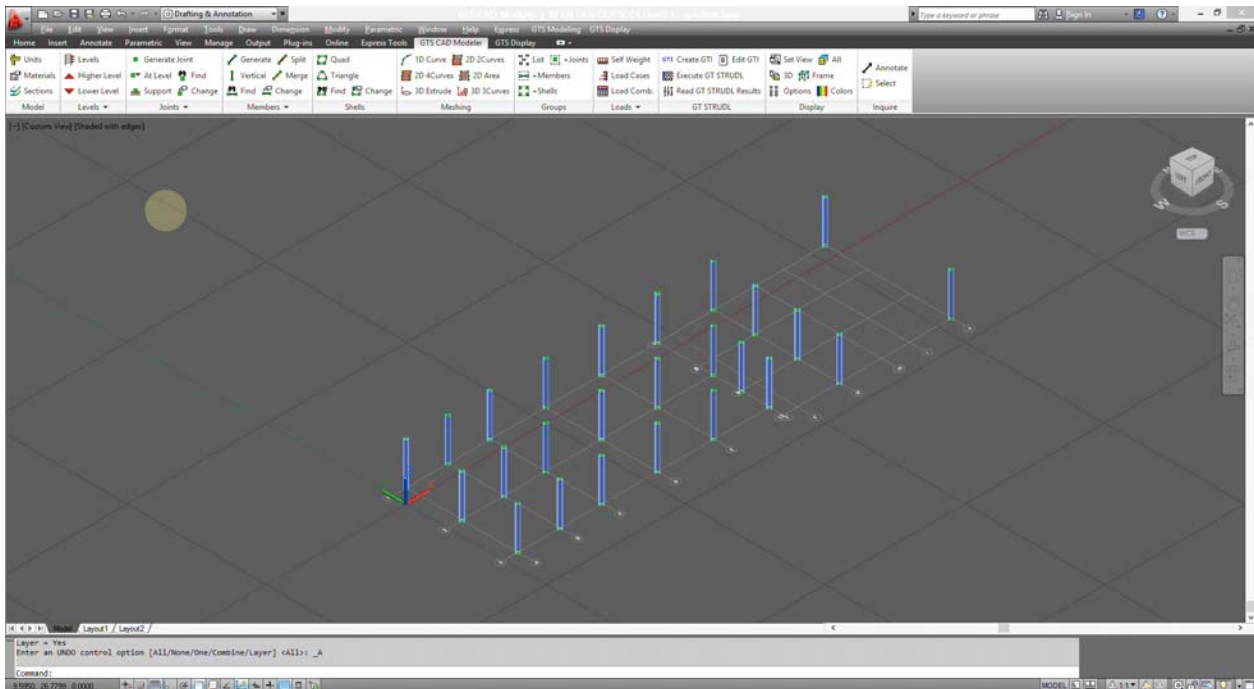


In the above dialog, the Filter options are set to find members which are HE-----320B and are located on Level 1. After clicking the Execute button, the Result box shows the members which were found to satisfy these criteria.

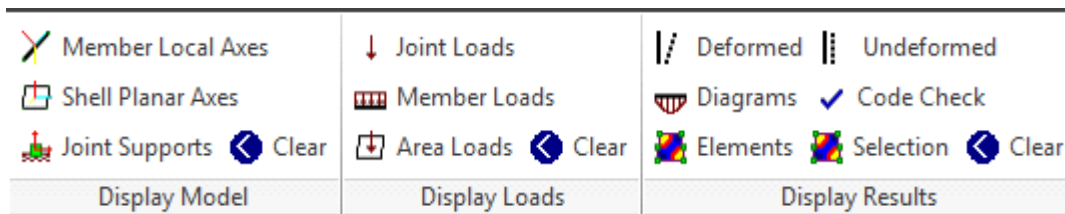
The Grid dialog has been enhanced to allow the creation of multiple grids and for the grids to be displayed at any level. The new Grid dialog is shown below:



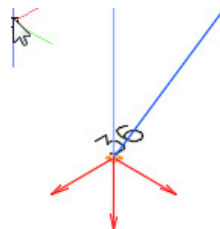
An example illustrating multiple grids on Level 1 is shown along with the placement of columns on the next page.



A new Ribbon Tab “GTS Display” has been created for all Display Functions as shown below:



Using the Display Model panel, you may now display the Local member coordinate system, the Planar coordinate system for Shell elements and the restrained degrees-of-freedom for supports. An example showing the display of the restrained degrees-of-freedom at a support are shown below:

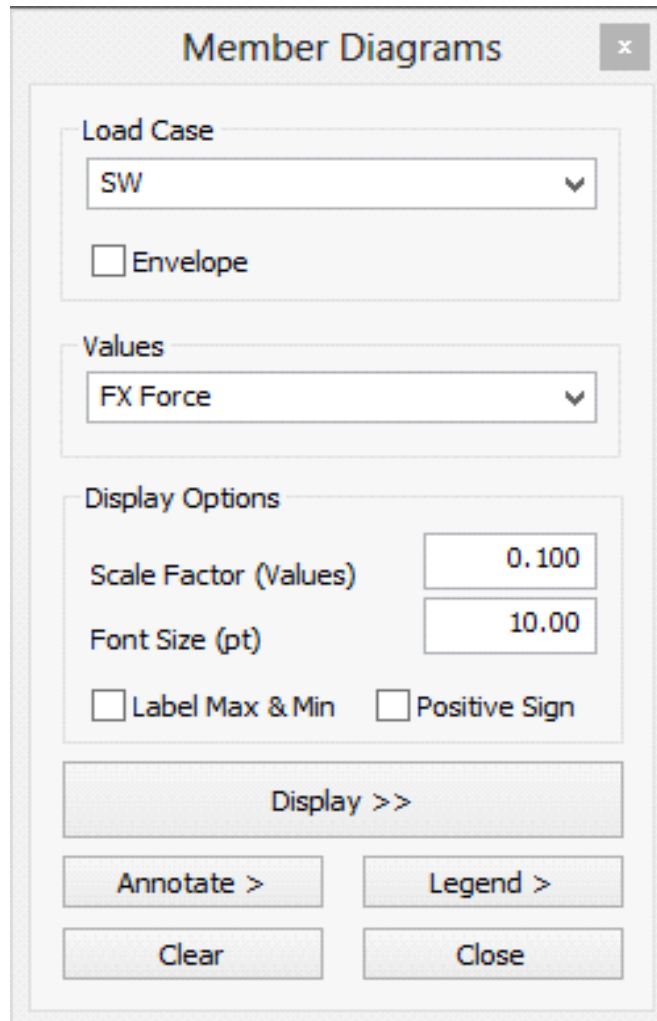


In the figure above, restrained forces are shown with a single arrowhead. Restrained moments would be shown with a double headed arrowhead.

The Display Loads panel allows you to display joint, member and area loads.

The Display Results panel allows you to display the deformed structure, member forces and moment diagrams and envelopes, finite element results and member code checks.

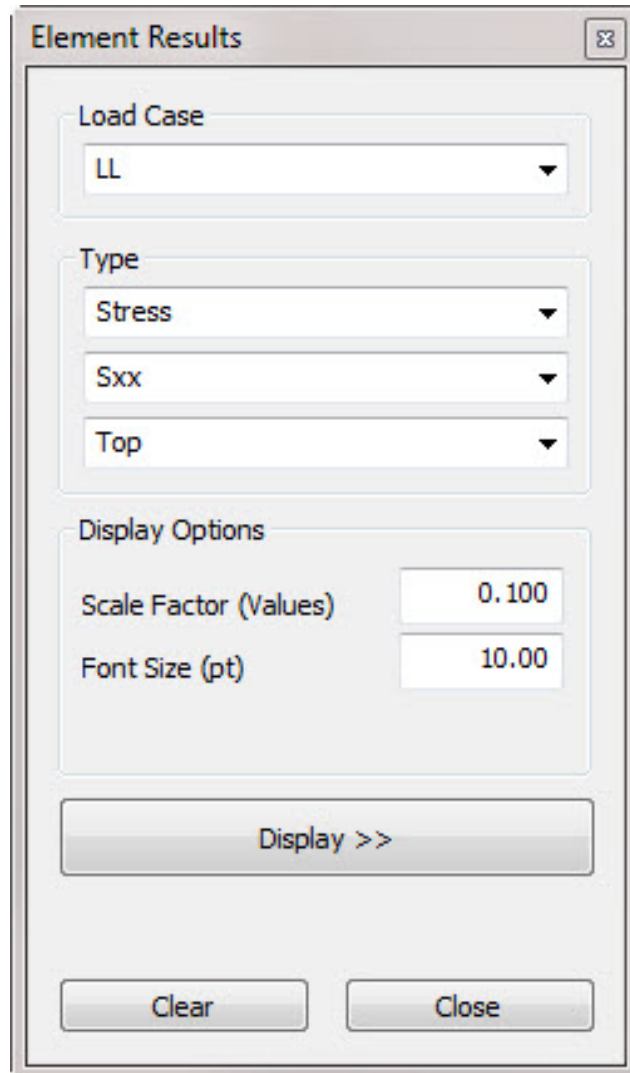
The new dialogs for member diagrams is shown below:



The image shows a software dialog box titled "Member Diagrams" with a close button (X) in the top right corner. The dialog is organized into several sections:

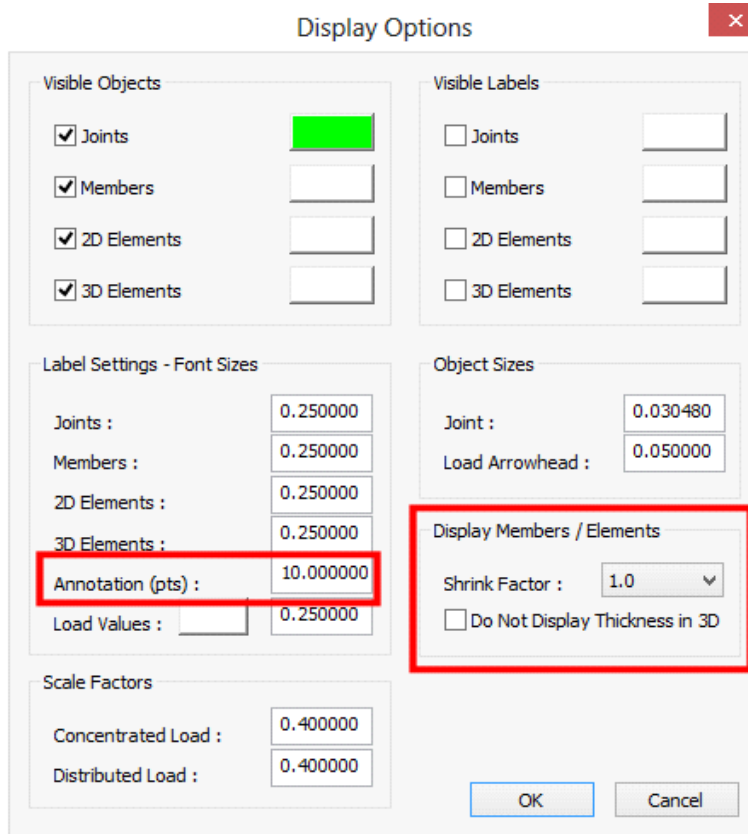
- Load Case:** A dropdown menu currently showing "SW".
- Envelope:** An unchecked checkbox.
- Values:** A dropdown menu currently showing "FX Force".
- Display Options:** A section containing:
 - Scale Factor (Values): A text input field with "0.100".
 - Font Size (pt): A text input field with "10.00".
 - Label Max & Min: An unchecked checkbox.
 - Positive Sign: An unchecked checkbox.
- Buttons:** A large "Display >>" button, and below it, four smaller buttons: "Annotate >", "Legend >", "Clear", and "Close".

The new dialog for Element Results is shown below:

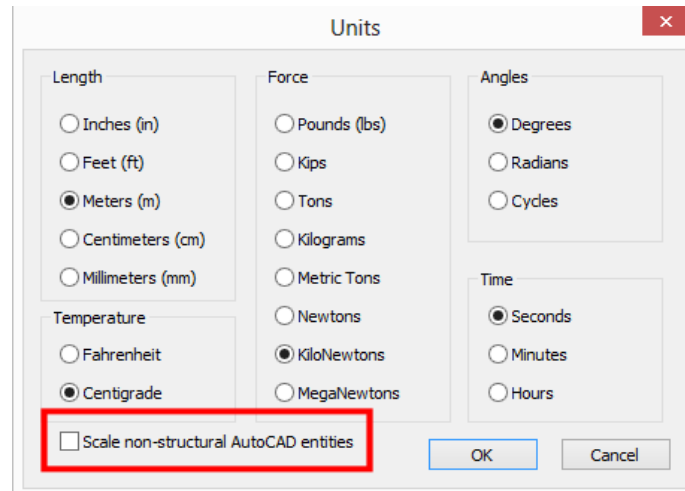


The Display Options dialog has been modified to allow you to now shrink the members and finite elements. In addition, the Display Options dialog has been modified to allow you to disable the display of finite element thicknesses which will speed up the display on large models. The modified Display Options dialog is shown on the next page. Also, the Annotation font is now entered in Points (pts) and not active units so that it is easier for the user to control the annotation size on the screen.

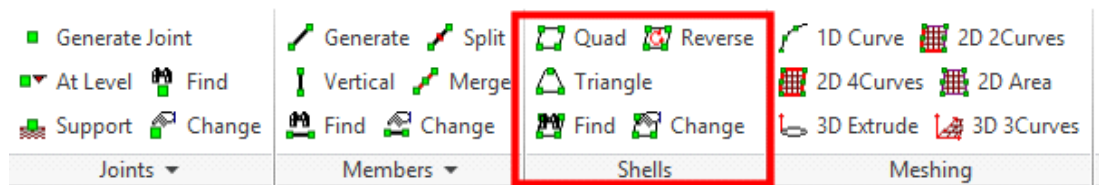
The modified Display Options dialog is shown below:



You now have the ability to scale AutoCAD non-structural entities using the modified Units dialog shown on the next page while changing the length units. If this option is checked, drawing entities such as lines and polylines are scaled together with CAD Modeler Objects.

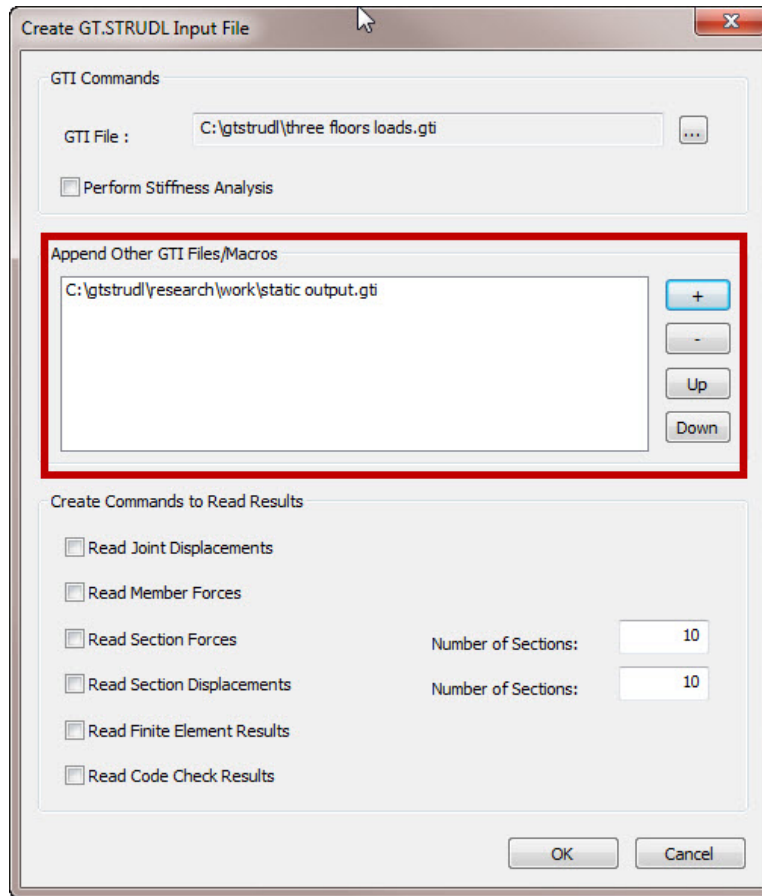


A new panel called Shells has been implemented under the GTS CAD Modeler tab. The new Shells panel is highlighted below:

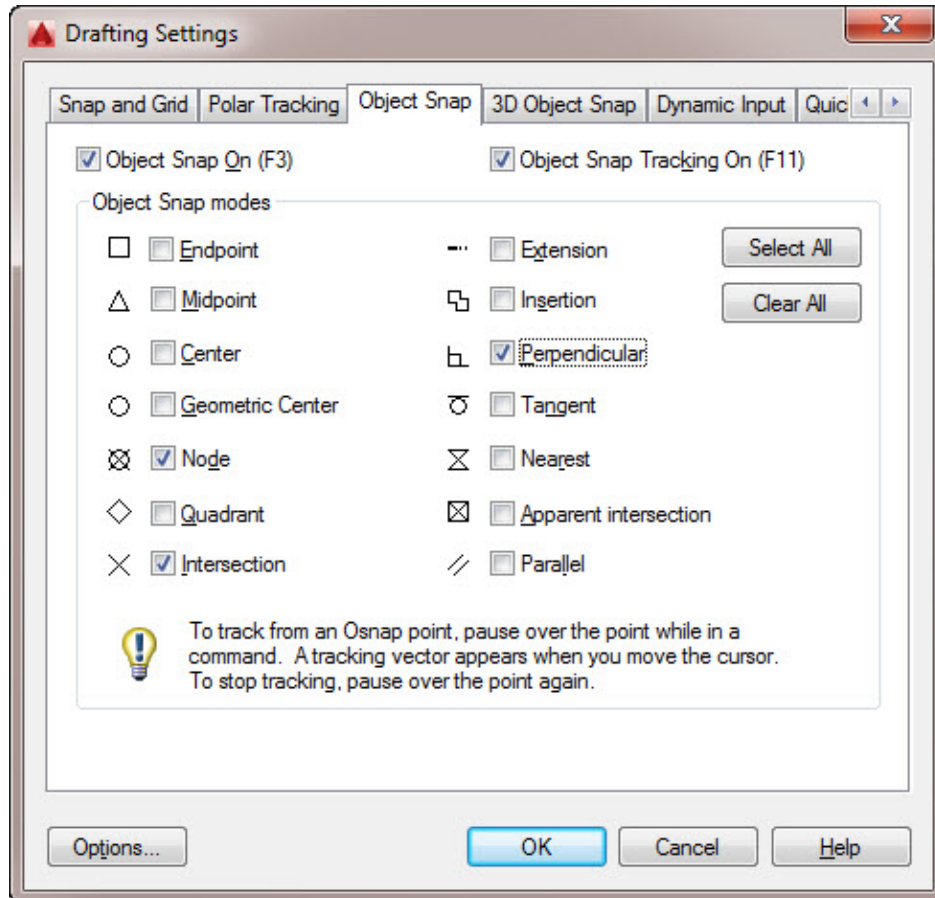


Previously, only commands were available to add Quad and Triangle elements or Find or Change them. In addition, you may now Reverse the element incidences in order to change the Planar coordinate system for shell elements.

The Create GT STRUDL Input File dialog has been enhanced to allow you to append other gti files or macros to the input file created by CAD Modeler. The enhanced dialog is shown below with the new Append Other GTI Files/Macros portion of the dialog highlighted:



An Object Snap in AutoCAD can now be applied when placing a GT STRUDL Member. Therefore, you can now snap on a member's Endpoint, Midpoint, Nearest point, Perpendicular to a member, etc. By entering the OSNAP command, these snap points are set in AutoCAD's Drafting Settings dialog which is shown below:



The following additional enhancements have been added to CAD Modeler:

- DBX files are now stored in the same directory as gti files having the same prefix.
- A new NO FLIP option is available when mirroring members and elements which will prevent the Planar system for elements being reversed and also keep the Local system for members the same.
- Clear Results is now much faster.
- Unicode file and directory names are now supported.

2.3 DBX

1. The REPLACE option has been added to the DBX FILE SPECS command. Use this option to overwrite existing DBX files.

Example

DBX FILE SPEC BINARY 'FileName' REPLACE

Documentation

Section 1, Volume 5, GT STRUDL Reference Manual.

2. The BINARY option has been added to the WRITE UNREGISTERED command. Use this option to write binary DBX files that do not require a DBX FILE SPEC command and are not added into the DBX directory data base.

Example

WRITE UNREGISTERED BINARY JOINT RESULTS 'JtRslt' ALL

Documentation

Section 2, Volume 5, GT STRUDL Reference Manual.

3. The character limit for file names has been changed from 80 to 256.

Documentation

Section 2, Volume 5, GT STRUDL Reference Manual.

2.4 Efficiency Improvements

1. In Version 2016, both the GTSES and GT64M static analysis solvers exhibit efficiency improvements when launched by the STIFFNESS ANALYSIS command because all available processors are used in the solution. In versions prior to and including 2015, the STIFFNESS ANALYSIS command launched these static analysis solvers as single thread processes. In Version 2016, these solvers have been found to perform the solution 2 to 8 times faster depending on the number of processors that are available on the computer.
2. The compilers used to create GT STRUDL were changed in Version 2016 which also resulted in efficiency improvements that weren't related to the improvements related for the GTSES and GT64M solvers described above. In some models, this may result in significant overall performance improvements when compared with previous versions.

2.5 Offshore

1. Several new features have been added to the definition, storage, and utilization of fatigue analysis S-N data as follows:
 - A. The STORE S-N DATA command has been enhanced with an option that allows for the specification of either AMPLITUDE or RANGE S-N stress data. Only AMPLITUDE could be specified previously.
 - B. The STORE S-N DATA command now includes an optional thickness size factor that is applied to the S-N stress when cycles to failure is computed from the defined curve.
 - C. The new STORE S-N CURVE DATA command has been added. This command allows for the specification of bi-linear log-log S-N curves by a set of parametric curve data.
 - D. Both the STORE S-N DATA and STORE S-N CURVE DATA commands can now store the specified S-N data on a user data set file.
 - E. The MEMBER S-N DATA command has been added, which is used to assign S-N curve data that were previously stored on a user data set file to specified members for which a fatigue analysis is performed.

Documentation:

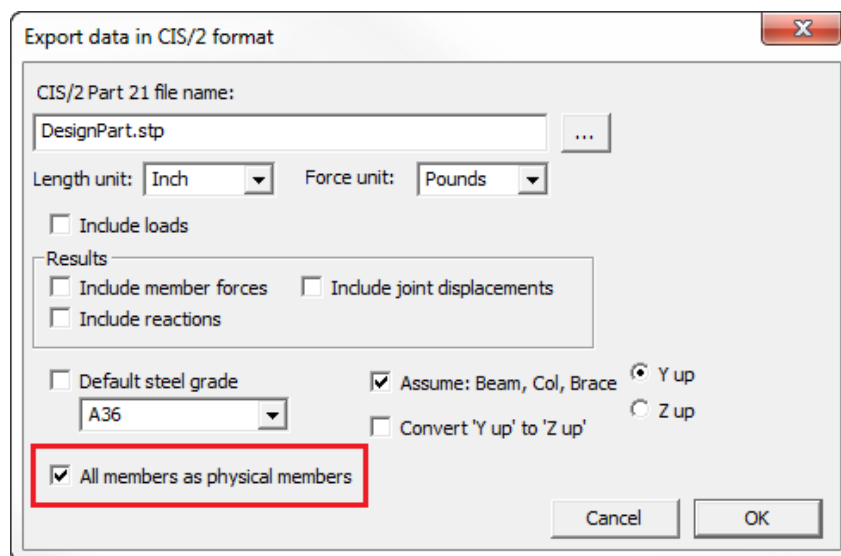
Fatigue Analysis Data, Section 5.4, Volume 8.

2.6 Steel Design

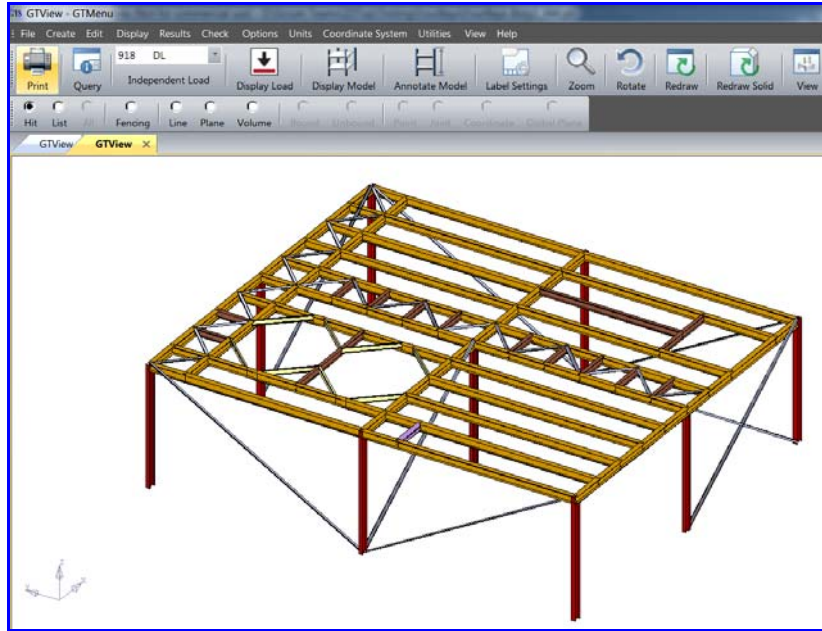
1. Seismic provisions from ANSI/AISC 341-10 have been added to AISC14 code. The seismic provisions are applicable to all rolled sections documented in the steel design AISC14 User Reference Manual. Steel design parameters are documented in the Table 1.6-1, Section 1.6 of the AISC14 User Reference Manual.
2. A new parameter 'N690-LD' has been added to N690-12 code check which allows user to specify impactive or impulsive force effect load names that are used to perform ANSI/AISC N690-12 code check. Parameter 'N690-LD' is documented in the Table 1.4-1, Section 1.4 of the AISC14 User Reference Manual.
3. The Parameter command has been modified to accept values without a member list. When a member list is not specified for a parameter, the ALL option is assumed and a warning message is issued. Note that when a parameter value is specified with the ALL option, all previous values for that parameter are removed. To remove the warning message, specify the member list.
4. The parameter 'WarnMess', which may be used to prevent printing of a specific warning message, is now documented in Section 7.2 of the Volume 2A User Reference Manual.

2.7 Export CIS/2 File to Interface to Other Intergraph Products

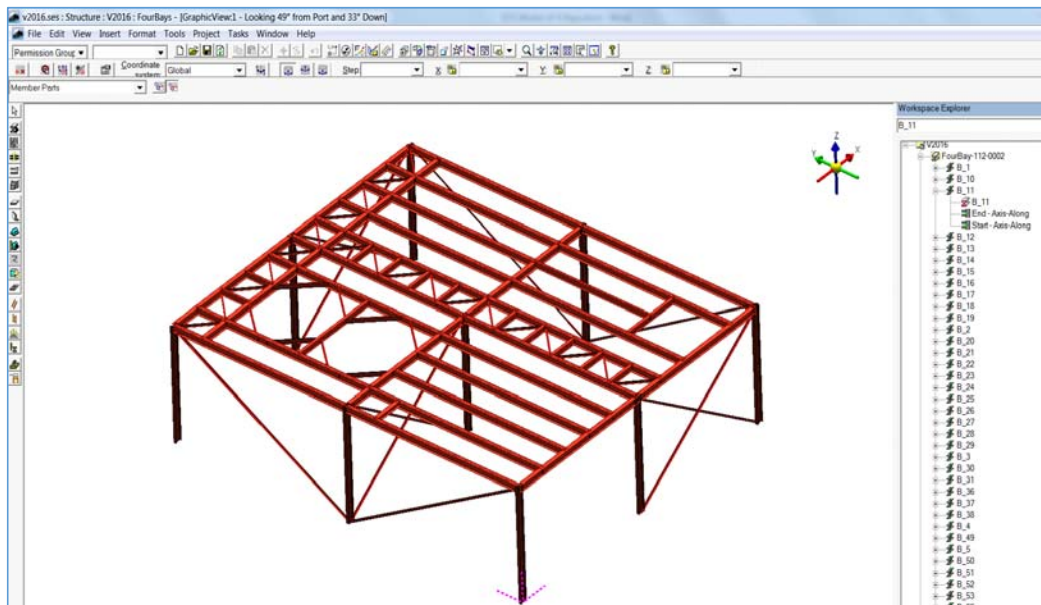
1. The generated .stp file from “File → Export CIS/2” will now include DESIGN_PART if the “All members as physical members” box is checked. This will allow viewing programs that understand a CIS/2 design model to view the GTSTRUDL structural model. Intergraph’s SmartPlant Interop Publisher (SPIO) can use the new format to create a file that can be referenced in Smart 3D, SmartPlant Review, or CAESAR II. This model can be updated any number of times from SmartPlant Interop Publisher allowing a user to continuously view the latest changes from the GTSTRUDL model.



Shown below is an example of taking a model created in GT STRUDL and bringing the model into Smart3D using the new CIS/2 option:



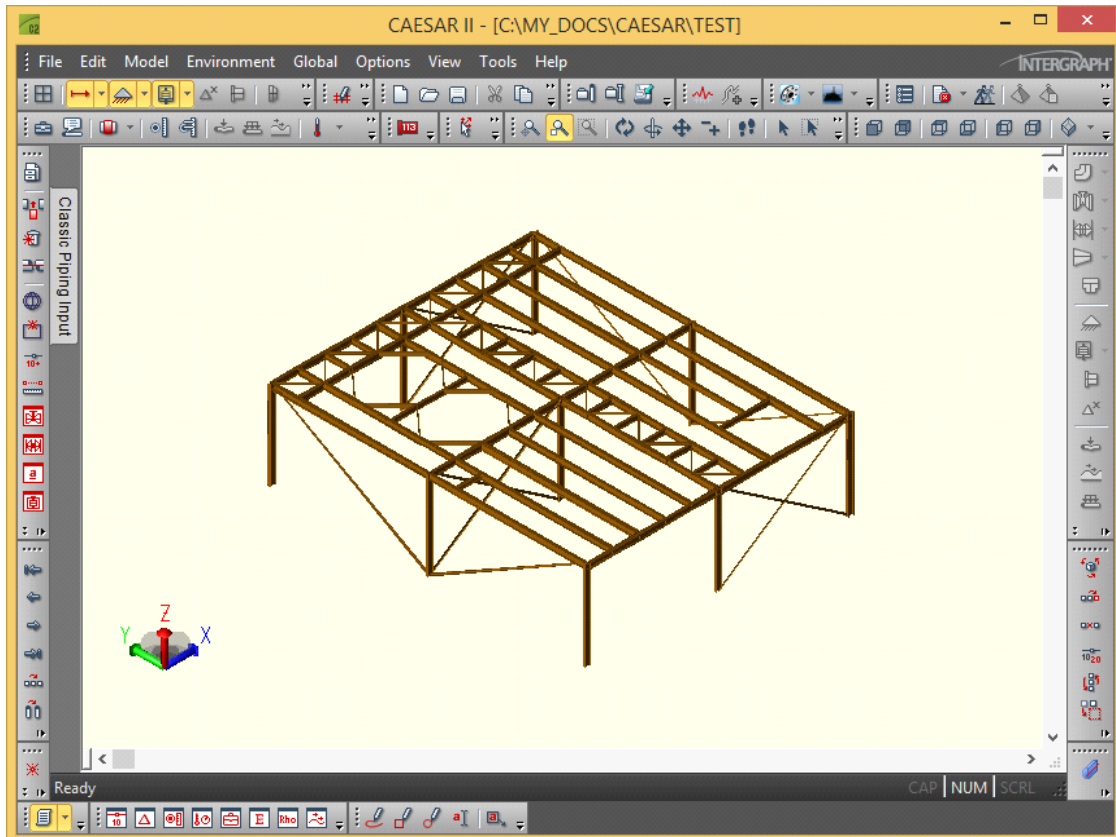
GT STRUDL model in GTMenu



Model in Smart3D using the CIS/2 Design File

You can also bring the model into Smart Plant Review, Smart Plant Construction and Smart Plant Foundation.

SPIOP will also provide you with the capability to create a VUE file which can be displayed as a reference model in Smart3D or in CAESAR II as shown below:



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 Base Plate Wizard

1. The possible incorrect "Passed" status for Anchors if two different interaction limits are set within the same base plate model has been fixed.
(GPRF 2015.03)
2. A deleted association of a stiffener with an attachment will no longer be persisted in the .gtbp file.
(No GPRF issued)
3. Changing the input units in the "Add a new load case" will no longer delete any data already entered.
(No GPRF issued)
4. The Pipe shape for the "Create from standard shapes" option in the Create Attachment dialog (Base Plate Wizard Users Guide, Section 3.2) now works correctly and allows the 'Cutout' option.
(No GPRF issued)

3.2 CAD Modeler

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

1. The 2D Area Mesh capability no longer aborts when creating large models.
2. The "Clear Results" no longer takes an excessive amount of time to finish and no longer aborts. In version 2016, there is a speed improvement and more stable behavior.

3. CAD Modeler will no longer create the .gti file if the Cancel Button is pressed in the "Create GTI" form. Also, there is now a warning to save the .dwg file prior to creating a .gti file.
4. Unicode directories (directories having Chinese, Spanish, Italian, Greek, etc characters) will now work.
5. Previously, there was a potential inconsistency between DBX files that already existed in the current directory which may contain results from another structure since the files names were always the same. Now, DBX files are created using the same file name and directory as the .dwg file.
6. In version 2015 there was a problem with the database when saving a structure using the default "Drawing1.dwg" filename. In Version 2016, there is now a check for this filename.
7. In version 2015 there was a limitation of 80 characters in the .gti path name. This limit has been increased to 256 characters.
8. The AutoCAD OSNAP functions (midpoint, endpoint, perpendicular, etc) now work when placing members.
9. An abort no longer occurs when importing a .gti file which contains Physical Member definitions.
10. Previously when selecting Exit in the Self Weight Form without previously selecting Create New or Save/Modify, the data was not stored. The data in the form is now stored.
11. Load cases without a description no longer produce incorrect .gti files.
12. An abort no longer occurs when trying to create a grid without first creating levels. Now, a level is added automatically, if needed, prior to creating a grid.
13. The UNDO function now works properly for data entered using forms. Previously, the UNDO function only worked correctly for data entered graphically.
14. The Legend and value annotation on the screen are now correct when displaying member or element results after changing the Length or Force units.

15. GTI files having force units of TONS or MTONS are now imported correctly. Previously, there was an error in the units conversion.
16. The error in the sequential ID numbering for joints, members, elements (starting from 1 and increasing by 1) has been corrected.
17. Errors in the Mirror and Copy commands have been corrected. Previously, an error occurred when using the Mirror command and members along the mirror line were selected to be mirrored to themselves. For the Copy command, an error occurred when using a zero displacement vector for the copy which resulted in members being copied onto themselves. In version 2016, these errors have been corrected although duplicate members/elements are generated.

3.2 DBX

1. Previously, if an incomplete DBX FILE SPEC command was issued and no valid DBX FILE SPEC command followed, no error message was issued when a named file was requested to be written. This error has been corrected and an error message is generated as shown in the example below.

```
{ 101} > DBX FILE SPECS BINARY $ 'FileName' omitted
{ 102} > WRITE REPLACE JOINT RESULTS 'Example02Step22.10' JOINTS 1

**** ERROR_GADBFA -- The specified file was not found in the DBX file
                    directory. Check the spelling of the file name:
                    Example02Step22.10
                    The requested file operation will not be performed.
```

(No GPRF issued)

3.3 Dynamic Analysis

1. The dynamic analysis COMPUTE command has been modified to include the computation of reactions at joint constraint master joints that also are support joints. Earlier versions do not perform this function.

Documentation

Computation of Backsubstitution Transient Results, Section 2.4.5.7,
Volume 3, GT STRUDL Reference Manual

Computation of Backsubstitution Steady State Results, Section 2.4.5.8,
Volume 3, GT STRUDL Reference Manual

Computation of Backsubstitution Response Spectrum Results and Modal
Combinations, Section 2.4.5.9, Volume 3, GT STRUDL Reference
Manual

Computation of Backsubstitution Harmonic Loading Results, Section
2.4.5.10, Volume 3, GT STRUDL Reference Manual

(GPRF 2015.02)

2. An abort will no longer occur for some finite elements if the density is specified as zero for the element and INERTIA OF JOINTS LUMPED has been specified. The GT STRUDL element types that were affected by this problem are shown below:

plane stress - IPQL, IPQLQ4 series Q6CDRL
plate bending: PBMITC
tridimensional - IPSL, IPSQ, TRANS3D, WEDGE15
plate : SBMITC

(GPRF 2015.06)

3.4 Finite Elements

1. The unit conversion for Element Edge Forces for axisymmetric elements such as the IPQL and IPQLQ4 is now correct. Edge forces for axisymmetric elements are now assumed to be in units of F/L^2 while the edge forces for plane stress and tridimensional elements are assumed to be in units of F/L . Volume 3 of the GT STRUDL Reference Manual has been updated to reflect this. Previously, edge forces on axisymmetric elements were correct only if the length unit was inches.

(GPRFs 2014.03 and 2015.04)

2. The abort and incorrect results in the command CALCULATE ERROR ESTIMATE ENERGY NORM for the IPSLIM and others elements has been corrected. This abort and other errors occurred in Versions 32, 33 and 2015.

Other more commonly used CALCULATE ERROR ESTIMATE options were not affected by this error such as the command shown below:

CALCULATE ERROR ESTIMATE DIFFERENCE FROM AVERAGE.

(No GPRF issued as the CALCULATE ERROR ESTIMATE command was and still is a prerelease feature)

3.5 General

1. In prior versions, the ROTATE LOAD command may fail to store all computed member load components when two or more individual member load entries are rotated. This error has been corrected.

Documentation:

The ROTATE LOAD Command, Prerelease Features, Section 5.4.1.

(GPRF's are not issued for prelease features)

3.6 GTMenu

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

1. Multiple property groups are no longer created in the input file for members which have the same properties.
2. Redraw Solid will now work correctly if a 2D finite element is attached to a 3D solid's element face. Previously, neither element would be shown in Redraw Solid if the shrink factor was 1.0.
3. An abort or incorrect geometry will no longer occur when Placing Members with the Split at Intersections option being checked.
4. The Pass/Fail Steel Code Check status display in GTMenu is now working when a Member Constraints command has been specified before the CHECK command.

3.7 Nonlinear

1. Nonlinear static and dynamic analyses now compute member end forces for nonlinear members that correctly reflect the presence of member eccentricities that are specified as local. In previous versions, the member end forces computed for nonlinear members having local eccentricities incorrectly assumed that all eccentricities were specified as global.

(GPRF 2015.05)

Documentation:

Nonlinear Static Analysis, Section 2.5, Volume 3.

Nonlinear Dynamic Analysis, Section 2.4.10, Volume 3

3.8 Steel Design

1. The lateral-torsional buckling (LTB) provisions in the AISC13 code for I-shape and Solid Rectangular Bar cross-sections has been corrected. This error occurred when a member was in combined axial tension and bending (major Z axis bending).

(GPRF 2015.01)

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 2016. 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.
3. User defined cross sections which have been stored in userdat.ds files can't be imported or used in the model creation.

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

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 Load Summations option available under CHECK MODEL will produce incorrect load summations for line, edge, and body loads on all finite elements. The Load Summations are also incorrect for projected loads on finite elements. The load summations for line and edge loadings should be divided by the thickness of the loaded elements. The body force summations should be multiplied by the thickness of the loaded elements for two-dimensional elements.

Workaround: You can check the load summation by specifying the LIST SUM REACTIONS command after STIFFNESS ANALYSIS.

(No GPRF issued)

4. Projected element loads will be displayed incorrectly when they are created or when they are displayed using Display Model → Loads.

Workaround: Verify that the loads are correct in the 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)

8. An abort will occur if more than one session of GTMenu is opened in the same GT STRUDL Working Directory.

(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 ACI Concrete Code 318-99. Rather than use the 318-99 code which is a prerelease feature, users should use the 318-05 code which is now a released feature. Consequently, this will be the last release with the 318-99 code available.
- 5.2.2 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)
- 5.2.3 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

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 ACI Code 318-99

Design of beams and columns by the 1999 ACI code has been added. Only members designated as TYPE BEAM or TYPE COLUMN in a DESIGN DATA command can be PROPORTIONed when the METHOD is set to ACI318-99. When you specify ACI318-99, you will be reminded that it is a pre-release feature by a message (see the Example below).

Note that CHECK is not available for codes after ACI318-77, including ACI318-99.

$$\text{METHOD} \left(\left\{ \begin{array}{l} \rightarrow \text{ULTIMATE (STRENGTH)} \\ \text{WORKING (STRESS)} \end{array} \right\} \right) \left\{ \begin{array}{l} \text{ACI318-99} \\ \text{ACI318-89} \\ \text{ACI318-83} \\ \text{ACI318-77} \\ \text{ACI318-63} \\ \text{(BSI) CP110-72} \\ \text{(BSI) BS8110} \end{array} \right\} - \\
 \\
 \left(\text{BARS} \left\{ \begin{array}{l} \rightarrow \text{ASTM} \\ \text{CANADIAN (STANDARD)} \\ \text{UNESCO} \\ \text{KOREAN (STANDARD)} \end{array} \right\} \right) \left(\left\{ \begin{array}{l} \rightarrow \text{NONSEISMIC} \\ \text{SEISMIC} \\ \text{MODERATE SEISMIC} \end{array} \right\} \right)$$

Example:

```
METHOD ACI318-99
```

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

```
DESIGN DATA FOR MEMBER 1
```

```
TYPE BEAM RECT
```

```
PROPORTION MEMBER 1
```

```
....
```

```
ACTIVE CODE = ACI 318-99
```

```
....
```

```
(the rest of the output is the same format as previous codes)
```

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

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

CONSTANT	Explanation	ACI 318-99	Assumed Value
FCP	Compressive strength of concrete, f'_c		4000 psi
FY	Yield strength of reinforcement, f_y		60000 psi
WC	Unit weight of plain concrete		145 pcf
DENSITY	Unit weight of reinforced concrete ⁽¹⁾		150 pcf
FC	Allow compr. stress in concrete, F_c	A.3.1	0.45(FCP)
VU	Ult. shear stress in beam with web reinf. ⁽²⁾	11.5.6.9	$(8\sqrt{FCP} + v_c)$ ⁽⁵⁾
V	Allow. shear stress in beam with web reinf.	A.3.1(b)	$(5.5\sqrt{FCP})$
RFSP	Splitting ratio, $f_{ct}/(\sqrt{f'_c})$ ⁽³⁾	9.5.2.3	6.7
FYST	Yield strength of stirrups		60000 psi
FYSP	Yield strength of spiral		60000 psi
FS	Allowable tension stress in primary reinf.		20000 psi for
FSC	Allowable compressive stress in column reinf. ⁽⁴⁾	A.3.2	Grades 40, 50
FV	Allowable tension stress in stirrups ⁽⁵⁾		24000 psi for Grade 60
PHIFL	Flexure capacity reduction factor	9.3.2	0.9
PHISH	Shear capacity reduction factor	9.3.2	0.85
PHIBO	Bond capacity reduction factor	9.3.2	0.85
PHITO	Torsion capacity reduction factor	9.3.2	0.85
PHISP	Spiral column capacity reduction factor	9.3.2	0.75
PHITI	Tied column capacity reduction factor	9.3.2	0.7
BLFR	Ratio of max p, (p - p') or (p _w - p _f) to p _{bal}	10.3.3	0.75
PMAXCO	Maximum allowable reinforced ratio in columns	10.9.1	0.08
PMINCO	Minimum allowable reinforced ratio in columns	10.9.1	0.01
PMINFL	Minimum allowable reinforced ratio in flexural members	10.5.1	200/FY
ES	Modulus of elasticity for reinf. steel	8.5.2	29x10 ⁶ psi
EC	Modulus of elasticity for concrete	8.5.1	$33(WC)^{1.5} \sqrt{FCP}$
EU	Ult. strain in concrete at extreme comp. fiber	10.2.3	0.003

Notes:

1. The constant 'DENSITY' is the GT STRUDL constant of the same name which has been set to a value of 150 pcf for reinforced concrete.
2. VU is multiplied by PHISH internally.
3. Calculations for V_c and T_c are modified by replacing $\sqrt{f'_c}$ with $(RFSP/6.70)\sqrt{f'_c}$ as per Section 11.2.1.1.
4. The assumed value of FSC is also limited to 30,000 psi maximum.
5. This value is defined only at the time of stirrup design.

5.2.2 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) -
 $\left. \begin{array}{l} \left\{ \begin{array}{l} \text{WOOD (AND) (ARMER)} \left\{ \begin{array}{l} \rightarrow \text{AVERAGE} \\ \text{MAXIMUM} \end{array} \right\} \\ \text{CALCULATE (RESULTANT) (ELEMENT) (FORCES)} \end{array} \right\} \text{ (ALONG) -} \\ \text{(CUT } \left\{ \begin{array}{l} \text{'a'} \\ i_1 \end{array} \right\} \left\{ \begin{array}{l} \text{JOINTS} \\ \text{NODES} \end{array} \right\} \text{ list}_1 \text{ ELEMENT list}_2 \text{ (TABLE } \left\{ \begin{array}{l} \rightarrow \text{ASTM} \\ \text{UNESCO} \end{array} \right\} \text{) -} \\ * \left\{ \begin{array}{l} \text{TOP (FACE) (BARS } i_2 \text{) (SPACING } v_1 \text{)} \\ \text{BOTTOM (FACE) (BARS } i_3 \text{) (SPACING } v_2 \text{)} \\ \text{BOTH (FACES) (BARS } i_4 \text{) (SPACING } v_3 \text{)} \end{array} \right\} - \\ \left\{ \begin{array}{l} \rightarrow \text{INNER (LAYER)} \\ \text{OUTER (LAYER)} \end{array} \right\} \text{ (COVER } v_4 \text{) (LINEAR (TOLERANCE) } v_5 \text{) -} \\ \text{(TORSIONAL (MOMENT) (WARNING) } v_6 \text{)}$

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.

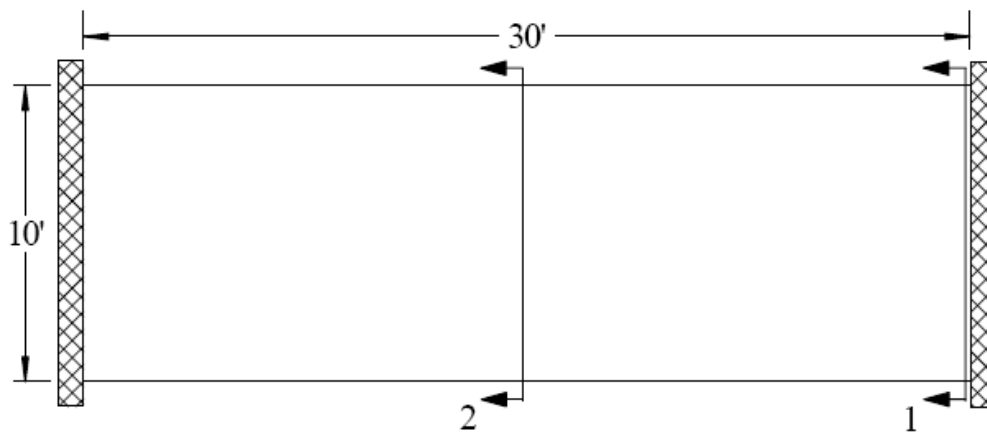


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

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.

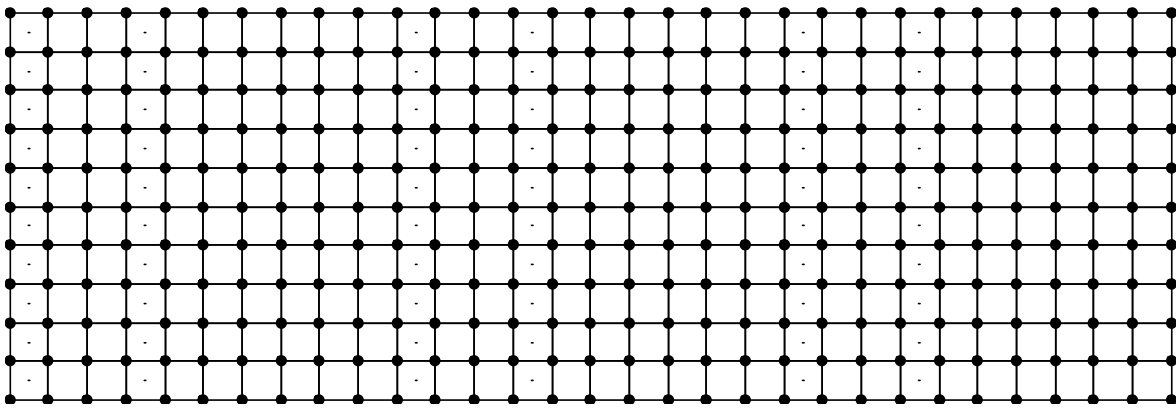


Figure 5.2.3-2 Example Finite Element Model

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

```

5.2.3 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 in the output vector. All error estimates are either zero or positive, since all use the absolute value of the various factors.

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

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

1. Enter GTMenu.
2. Select Results, Finite Element Contours, and then Energy & Stress Error Estimates.
3. Select the Estimate Method including Value, Surface, and Stress Component.
4. Select the Loading.
5. Select Display (solid colors or lines) to produce a contour of the error estimate.
6. Select Legend to place a legend on the screen indicating the type of error estimate, loading, and surface.

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

General form:

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

Explanation:

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

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

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

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

PROBLEM - None

ACTIVE UNITS  FEET KIP  RAD  DEGF  SEC
```

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

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

CONSTANTS/BETA Commands for Adjusted Beta Angles

=====

UNITS RAD

CONSTANTS

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

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

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

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

Elements:

$\underline{i}_R / \text{'a}_R \text{'}$ = 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:

$$\text{REFERENCE (COORDINATE) (SYSTEM) } \left\{ \begin{array}{l} i_1 \\ 'a_1' \end{array} \right\} -$$

$$\left\{ \begin{array}{l} (\text{ORIGIN } [\underline{X}] v_x [\underline{Y}] v_y [\underline{Z}] v_z) (\text{ROTATION } [\underline{R1}] v_1 [\underline{R2}] v_2 [\underline{R3}] v_3) \\ \left\{ \begin{array}{l} \text{JOINT } \left\{ \begin{array}{l} i_2 \\ 'a_2' \end{array} \right\} \\ \left[\begin{array}{l} \underline{X} v_4 \quad \underline{Y} v_5 \quad \underline{Z} v_6 \end{array} \right] \end{array} \right\} \left\{ \begin{array}{l} \text{JOINT } \left\{ \begin{array}{l} i_2 \\ 'a_2' \end{array} \right\} \\ \left[\begin{array}{l} \underline{X} v_4 \quad \underline{Y} v_5 \quad \underline{Z} v_6 \end{array} \right] \end{array} \right\} \left\{ \begin{array}{l} \text{JOINT } \left\{ \begin{array}{l} i_2 \\ 'a_2' \end{array} \right\} \\ \left[\begin{array}{l} \underline{X} v_4 \quad \underline{Y} v_5 \quad \underline{Z} v_6 \end{array} \right] \end{array} \right\} \end{array} \right\}$$

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 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

{ 'a₁' } coordinate-specs₁

.
.
.

{ 'a_n' } coordinate-specs_n

Elements:

coordinate-specs = [X] v₁ [Y] v₂ [Z] v₃

where,

'a₁', 'a₂', ..., 'a_n' = 1 to 8 character alphanumeric Point identifiers beginning with P (i.e. P1 P2 ...)

v₁, v₂, v₃ = Cartesian Point coordinates (integer or real).

(2) **GTMenu LINE INCIDENCES**

General Form:

GTMenu LINE INCIDENCES

{ 'a₁' } type₁ incidence-specs₁

·
·
·

{ 'a_n' } type_n incidence-specs_n

Elements:

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

incidence-specs = { point₁' } { 'point₂' } . . . { '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

End of Document