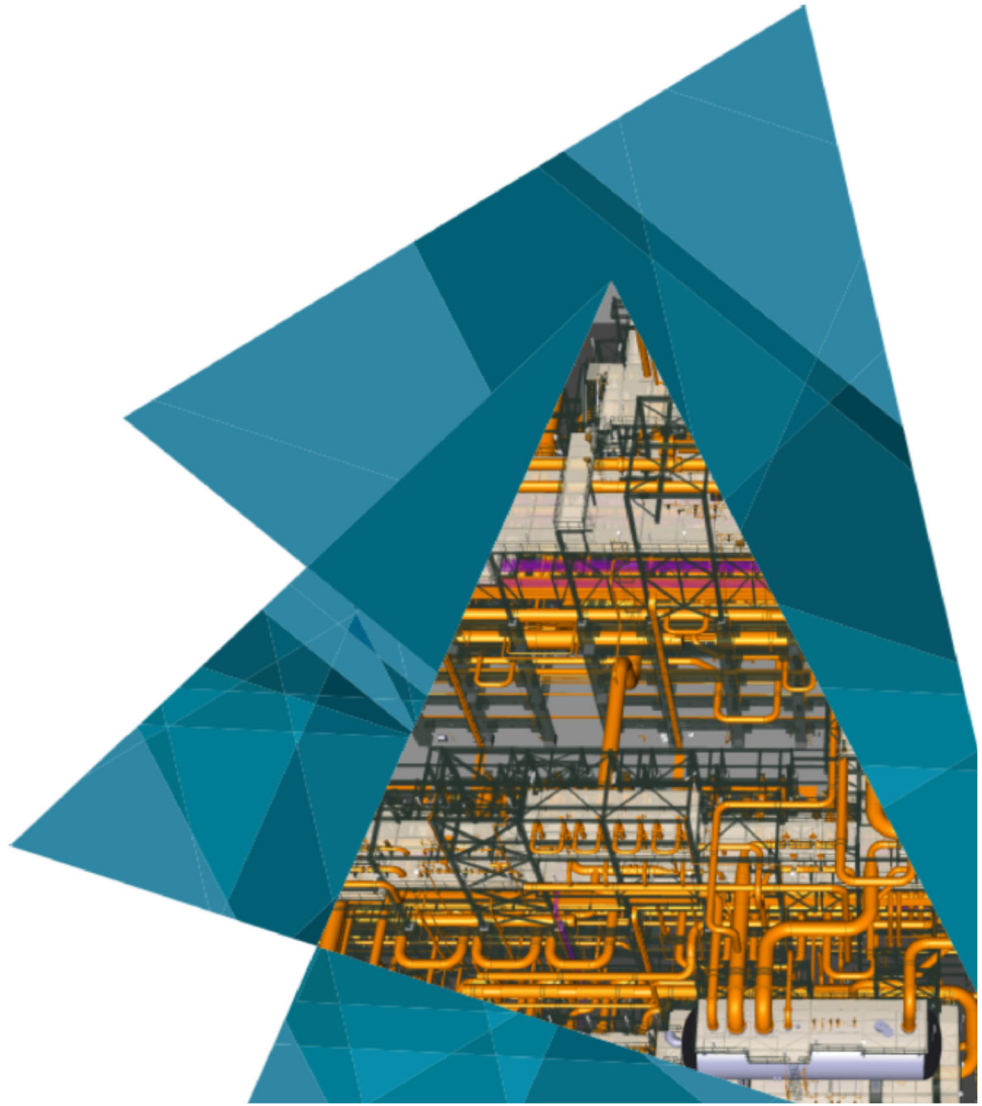


GT STRUDL® Version 44

Release Guide



Release Date: November 2025



Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 44 for use on PCs under the Microsoft Windows operating systems.

Copyright

Copyright © 2025 Hexagon AB and/or its subsidiaries and affiliates. All rights reserved.

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

U.S. Government Restricted Rights Legend

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- Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States.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

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference 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 Intergraph Smart Support, 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 Software License Agreement ("SLA") 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 SLA 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 use. 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 SLA 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

The Software Products and any software products obtained from Intergraph Corporation, its subsidiaries, or distributors, including any technical data related to these products ("Technical Data") are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. To the extent prohibited by United States or other applicable laws, these Intergraph Corporation software products and any software products obtained from Intergraph Corporation, its subsidiaries or distributors, Technical Data and any derivatives of either, shall not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

- a. to Cuba, Iran, North Korea, the Crimean region of Ukraine, or Syria, or any national of these countries or territories.
- b. to any person or entity listed on any United States government denial list, including, but not limited to, the United States Department of Commerce Denied Persons, Entities, and Unverified Lists, the United States Department of Treasury Specially Designated Nationals List, and the United States Department of State Debarred List. Visit www.export.gov for more information or follow this link for the screening tool: <https://legacy.export.gov/csl-search>.
- c. to any entity if Customer 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 Customer knows, or has reason to know, that an illegal reshipment will take place.

Customer shall hold harmless and indemnify PPM for any causes of action, claims, costs, expenses and/or damages resulting to PPM from a breach by Customer or any user of the export compliance restrictions set forth in this Agreement.

Any questions regarding export or re-export of these software products should be addressed to Hexagon PPM, Export Compliance Department, 305 Intergraph Way, Madison, Alabama 35758, USA or at exportcompliance@intergraph.com.

Trademarks

Intergraph®, the Intergraph logo®, Intergraph Smart®, SmartPlant®, SmartMarine®, SmartSketch®, SmartPlant Cloud®, PDS®, FrameWorks®, I-Route, I-Export, Isogen®, SPOOLGEN, SupportManager®, SupportModeler®, SAPPHIRE®, TANK, PV Elite®, CADWorx®, CADWorx DraftPro®, GTSTRUDL®, and CAESAR II® are trademarks or registered trademarks of Intergraph Corporation or its affiliates, parents, subsidiaries. Hexagon and the Hexagon logo are registered trademarks of Hexagon AB or its subsidiaries. Microsoft and Windows are registered trademarks of Microsoft Corporation. ACIS is a registered trademark of SPATIAL TECHNOLOGY, INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, ProtoViewCtrl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Incorporates portions of 2D DCM, 3D DCM, and HLM by Siemens Product Lifecycle Management Software III (GB) Ltd. All rights reserved. Gigasoft is a registered trademark, and ProEssentials a trademark of Gigasoft, Inc. VideoSoft and VXFlexGrid are either registered trademarks or trademarks of ComponentOne LLC 1991-2017, All rights reserved. Oracle, JD Edwards, PeopleSoft, and Retek are registered trademarks of Oracle Corporation and/or its affiliates. Tribon is a trademark of AVEVA Group plc. Alma and act/cut are trademarks of the Alma company. Other brands and product names are trademarks of their respective owners.

Table of Contents

Chapter	Page
NOTICES.....	ii
Table of Contents.....	iv
Chapter 1	
Introduction.....	1-1
Chapter 2 New Features in Version 44	
2.1 General.....	2-1
2.2 Steel Design.....	2-4
2.3 GTMenu.....	2-5
2.4 GTSelos	2-14
2.5 Base Plate Wizard	2-14
Chapter 3 Error Corrections	
3.1 GTMenu	3-1
3.2 General	3-1
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 A new national annex parameter for EC3-2005	

Steel design code.....	5.2-1
5.2.2 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)	5.2-5
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 The 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.....	5.4-4
5.4.2-1 Printing Reference Coordinate System Command	5.4-7
5.4.3 GTMenu SURFACE DEFINITION Command	5.4-8
5.4.4 Export to CAESAR II	5.4-10
5.4.5 Import CAESAR II Pipe Load	5.4-12

Chapter 1

Introduction and Installation

Version 44 covers GT STRUDL operating on PCs under the Windows 11 x64 based operating systems. Installation requires approximately 2.5 GB of storage for GT STRUDL 44. To interactively install GT STRUDL 44, run GTS_Setup.exe, as explained on the next page.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 43. Chapter 2 briefly describes the new features, including reference manual section numbers for more information when appropriate.

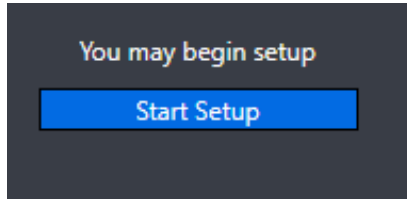
Chapter 3 provides you with details regarding error corrections that have been made since the release of Version 43.

Chapter 4 describes known problems with Version 44.

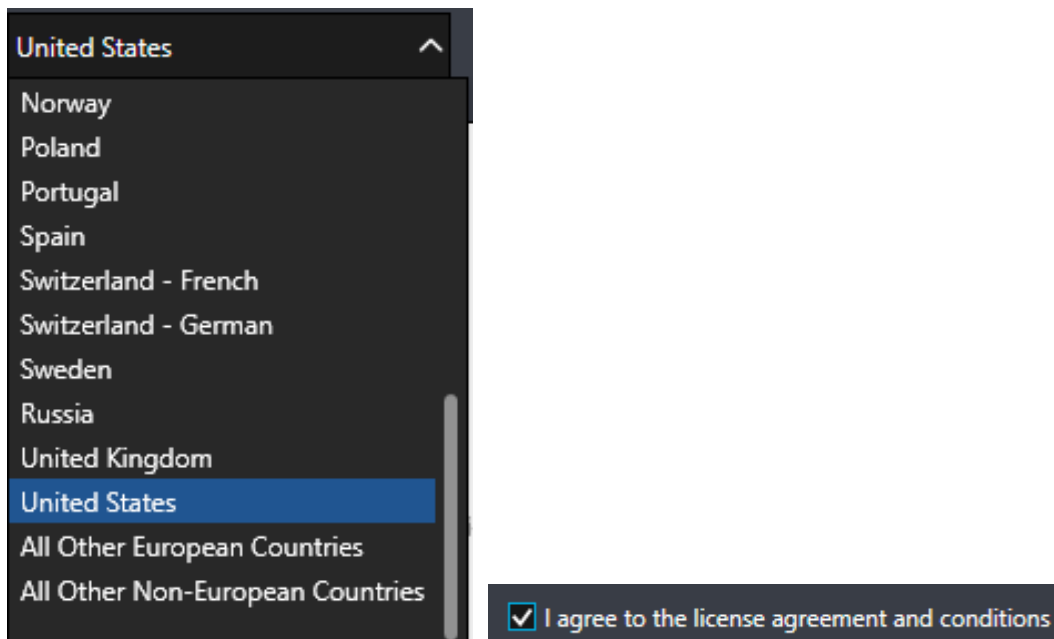
Chapter 5 describes prerelease features -- new features which have been developed and subjected to limited testing or features for which the user documentation has not been added to the GT STRUDL User Reference Manual. The command formats and functionality of the prerelease features may change before they become supported features based on additional testing and feedback from users. The Prerelease features are subdivided into Design, Analysis, and General categories.

Installation

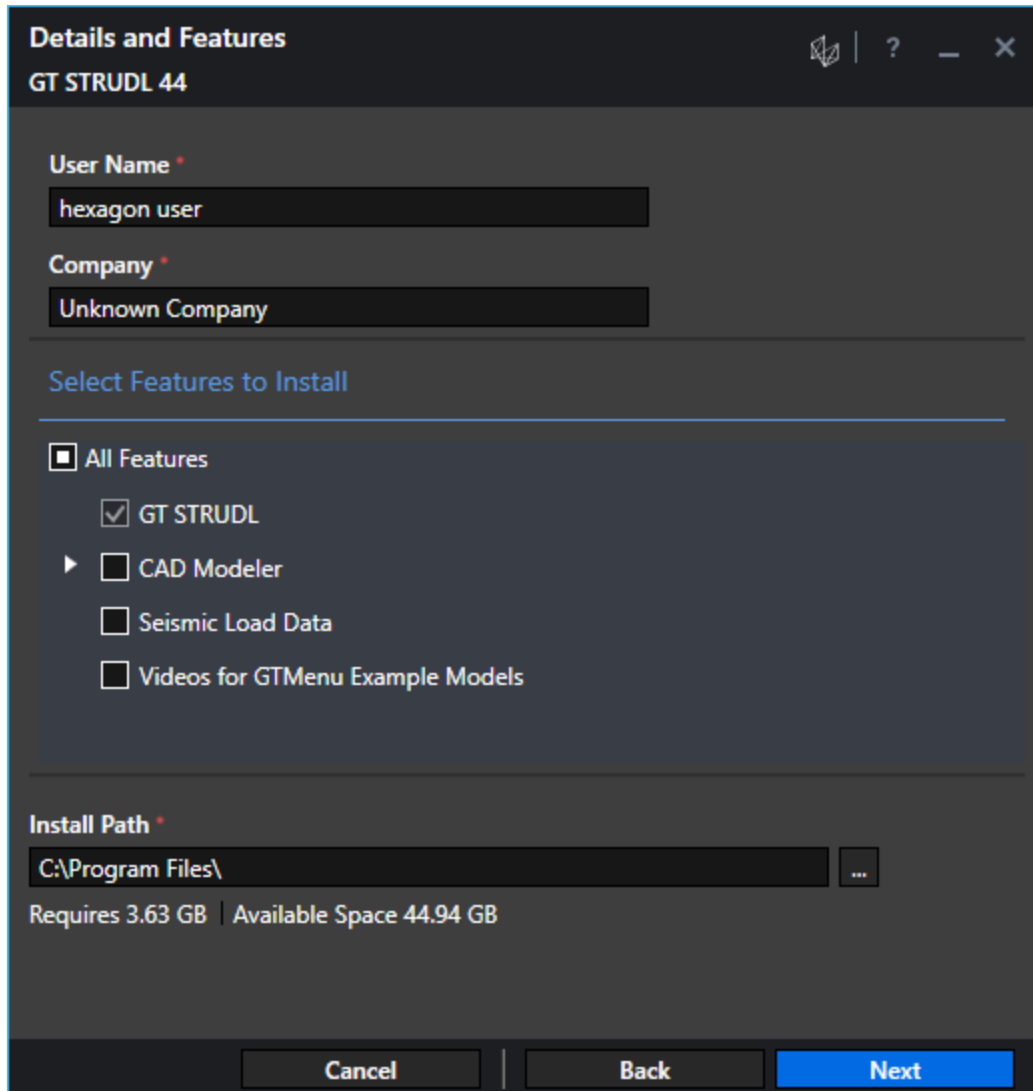
Initialize installation by double-clicking “GTS_Setup.exe” in the installation folder and then clicking the Start Setup button.



Select the desired language of the License Agreement and then click “I agree...”



On the subsequent “Details and Features” page enter your User Name and Company name (optional), choose the features you wish to install, and set the Install Path. The default feature selection is GT STRUDL only. The other features are described on the next page. The standard Install Path is “C:\Program Files”, but you may change to any path that has sufficient room to install.



CAD Modeler

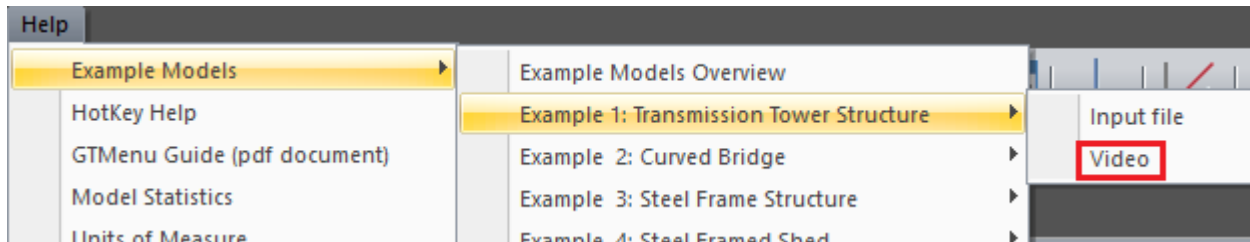
CAD Modeler is the CAD based structural modeler that gives you the power of AutoCAD® or BricsCAD® to create structural models that can then be passed to GT STRUDL for analysis. If you decide not to install CAD Modeler at this time, you can install it later by running GTS_Setup and selecting the Modify option. AutoCAD® or BricsCAD® must be installed on your computer to be able to use CAD Modeler. Supported versions are AutoCAD® 2020 through 2025 and BricsCAD® (Platinum or Pro) 21 through 25.

Seismic Load Data

This data used for determining S_s and S_1 (spectral response acceleration values) based on latitude and longitude. This data is about 1 GB in size. It is used for the MAP option of the SEISMIC LOAD command (Section 2.1.11.3.10.1, Volume 1 of the GT STRUDL Reference

Manual). If you decide not to install the Seismic Load Data at this time, you can install it later by running GTS_Setup and selecting the Modify option.

Videos for GTMenu Example Models



These are mp4 videos of using the 12 example files included with GTMenu Help. The example input files are always included in the installation and are available for input. The videos total about 215 MB and if you decide not to install the Videos for GTMenu Example Models at this time, you can install them later by running GTS_Setup and selecting the Modify option.

Silent Installation:

Below are the instructions for silent (no GUI) installation and uninstallation:

Navigate to the folder where GTS_Setup.exe exists.

Install:

To install just GT STRUDL without the optional features:

```
GTS_Setup.exe /install /silent SLAACCEPT=YES
```

To install GT STRUDL with the optional features "SESIMIC LOAD DATA" and "Videos for GTMenu Example Models", provide a list of features with comma separation to "ADDLOCAL="

```
GTS_Setup.exe /install /silent  
ADDLOCAL="GTSTRUDL,SeismicLoadData,VideosforGTMenuExampleModels" SLAACCEPT=YES
```

To include CAD Modeler in silent installation, include feature CADMODELER in the ADDLOCAL instruction followed by the CAD versions to be supported:

```
ADDLOCAL="GTSTRUDL,CADMODELER,AUTOCAD2025,BRICSCAD2023"
```

Modify:

Use the /modify option and provide a list of the required software with comma separation to "ADDLOCAL=" and include a list of features which are to be removed following "REMOVE= "

```
GTS_Setup.exe /modify /silent ADDLOCAL="GTSTRUDL,CADMODELER,AUTOCAD2025,BRICSCAD2023"  
REMOVE ="AUTOCAD2019" SLAACCEPT=YES
```

Repair:

```
GTS_Setup.exe /repair /silent
```

Uninstall:

```
GTS_Setup.exe /uninstall /silent
```

This page intentionally left blank.

Chapter 2

New Features in Version 44

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 44. This release guide is also available online upon execution of GT STRUDL under menu “Help → Reference Documentation → GT STRUDL Release Guide”.

2.1 General

1. The GENERATE TAPERED PROPERTIES command has been added to facilitate the modeling of tapered poles with pipe or polygon properties. The new command is documented in Volume 1 of the Reference Manual, Section 2.1.9.4.

Syntax:

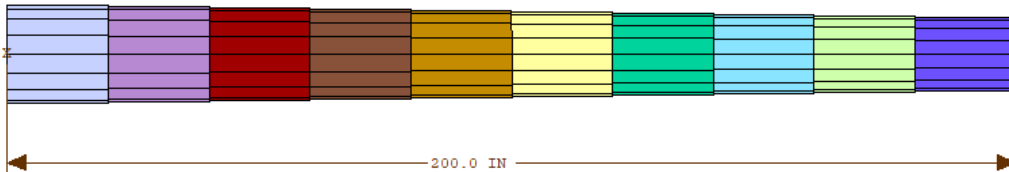
```
GENERATE TAPERED (MEMBER) (PROPERTIES) list { PIPE pipe-values
                                               POLYGON polygon-values }
```

where pipe-value and polygon-values are the same as documented in Section 2.1.9.2.

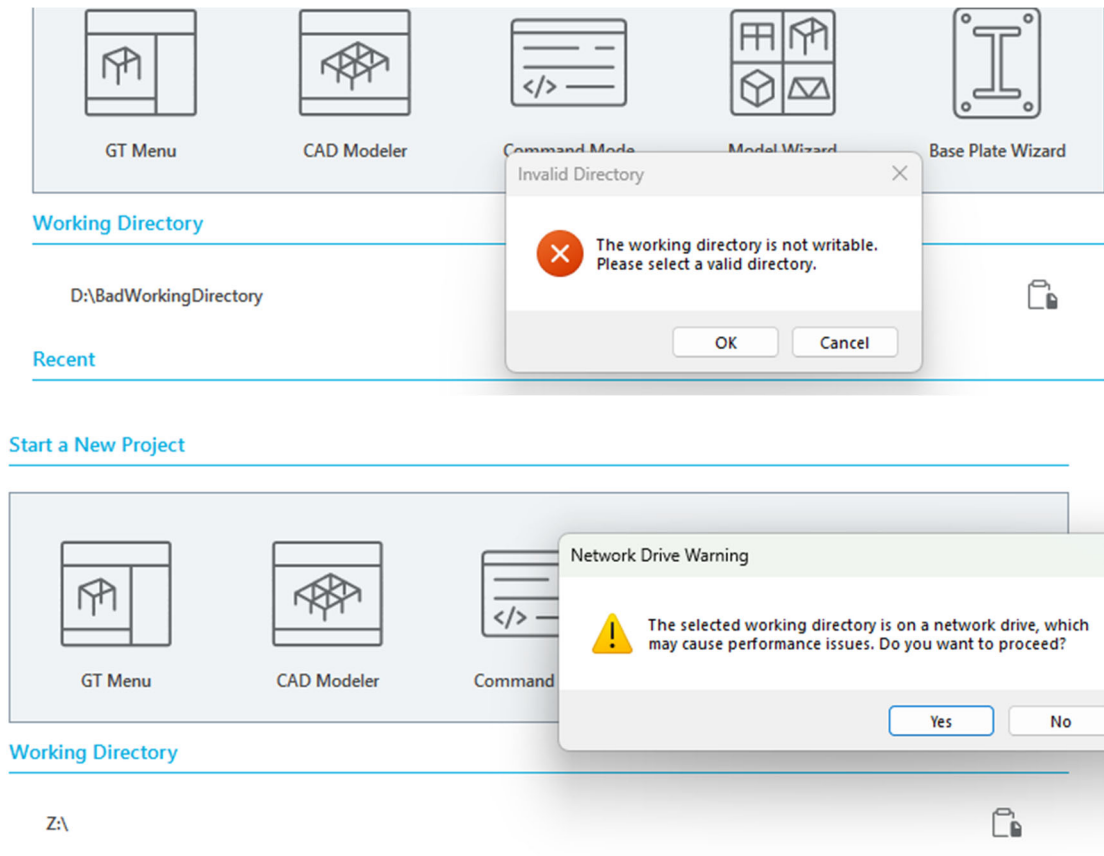
Example:

For 10 members, 20.0 inches long:

```
GENERATE TAPER 1 TO 10 PIPE OD 19.7 TO 14.30 THICK 0.1875
```



2. Working directory validation is implemented into GTShell. When selecting a working directory, the program will automatically flag if the working directory is not writable and prompts a change. If the user does not change the invalid working directory, it will automatically be set to a temp folder. The program will now also warn about diminished performance if the working directory is a network drive.



- When a FORM LOAD is added to in CHANGES mode, the updated load and factor list will be included when PRINT LOAD output is generated. Previously the list would not be included after changes. Also, the NOREFORM option is not required for analysis after changes to a FORM LOAD.

Documentation: 2.1.11.3.2 Independent FORM LOADING Command, Volume 1

- All standard and GT64M nonlinear analysis functions, including static analysis, cable prestress analysis, pushover analysis, and nonlinear dynamic analysis, have been updated to support the NLSGAP element, which was released in Version 43. The NLSGAP element is a specialized compression-only nonlinear spring element designed for the purpose of analyzing the conditions of contact between two closely spaced points within a structural model.

Documentation: The Gap Finite Element, Section 2.5.3.4, Volume 3,

- Version 44 includes enhancements to the standard and GT64M nonlinear dynamic analysis algorithms to improve numerical time history integration performance and accuracy. In earlier versions, the instantaneous stiffness matrices for nonlinear members and elements, and the effective global stiffness matrix, were updated only during the initialization of each time step and remained constant through the equilibrium iterations. In Version 44, these

matrices are updated at every equilibrium iteration inside each time step. This change generally lowers the number of equilibrium iterations required for convergence and provides greater stability during intervals of rapidly changing stiffness.

6. The following example of the DAMPER ELEMENT DATA command sequence illustrates the case wherein only one joint name, rather than two, is given for the INCidences option in the properties line for viscous damper element 'DAMP2':

```
DAMPER ELEMENT DATA
'DAMP2' INC 2 LOCAL CTZ 1.E7
END DAMPER DATA
```

In Version 43 and earlier, when element 'DAMP2' was created, a joint named 'DEDAMP2' was automatically generated as a fixed starting joint with the single specified joint 2 as the ending joint, and having coordinates calculated by offsetting the coordinates of joint 2 by a value of 0.001 inches, thus always creating a finite-length element. In version 44, this convention has been changed such that the unspecified support joint is created as coincident with the specified end joint. This always creates a zero-length damper element, which results in structure damping property changes revealed by differences in the analysis results when compared to those of Version 43.

Documentation:

The Damper Element Data Command, Section 2.4.3.7.1, Volume 3, GT STRUDL User Reference Manual.

2.2 Steel Design

1. The new steel design code ‘CSA-2024’ has been added. The code is based on the Canadian Standards Association, *S16-24, Design of Steel Structures*, Published in April 2024 by CSA Group, Toronto, Ontario, Canada. The CSA-2024 code is based on the limit state design technique. See the full documentation of the implementation in “Steel Design Code CSA-2024” available through Help → Reference Documentation → Reference Manuals → Steel design (Volumes 2A, 2B plus codes) → CSA-2024: Canadian Standards Association S16-24.

The following shapes are included in the implementation:

I Shapes (W, S, M, and HP)

Channels

Single Angles

Tees

Double Angles

Solid Round Bars

Solid Square Bars

Rectangular Bars

Circular Hollow Sections (Pipes)

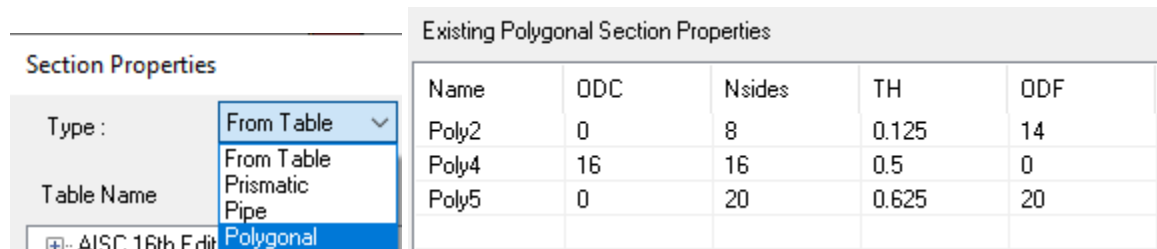
Rectangular Hollow Sections (Tubes)


Access to the new design code is through the parameter “CSA-2024”.

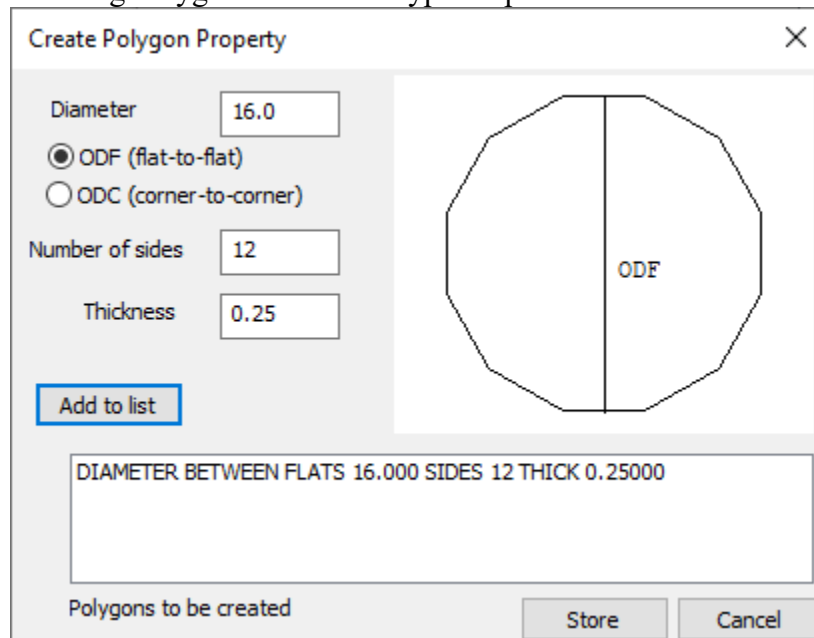
2.3 GTMenu

- The ability to create polygonal properties has been added to the Section Properties dialog. Additionally, prismatic properties created with the POLYGON option for MEMBER PROPERTIES in command mode are recognized as such in GTMenu and have a property name starting with “Poly”.

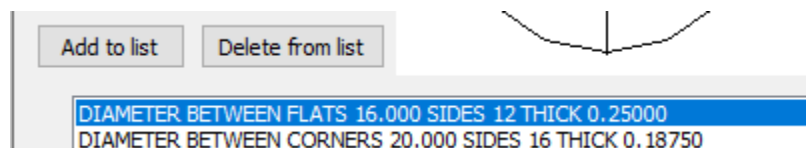
Polygonal properties are accessed through the Type dropdown



Use the  button to open the Create Polygon Property dialog after choosing Polygonal from the Type dropdown menu.



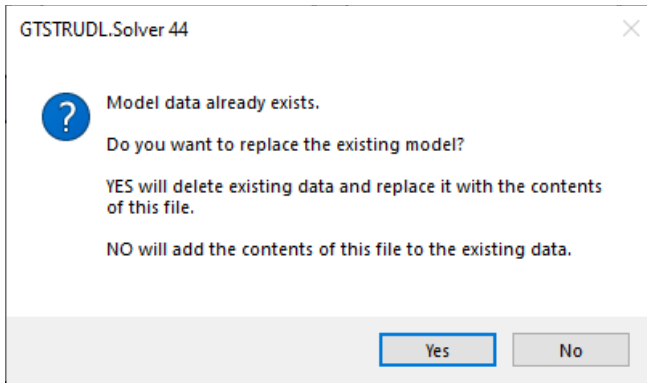
Enter the geometrical data, specifying whether the Diameter is flat-to-flat or corner-to-corner, then click the “Add to list” button. You can create more than one polygonal property. If you add an incorrect property to the list, it can be removed by selecting it and clicking the “Delete from list” button.



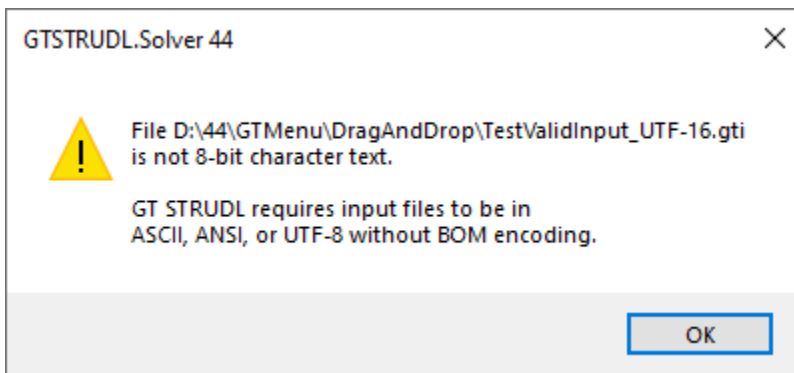
Click the Store button to add the new properties in the list to the GTMenu database.

Polygonal properties can also be created with the Generate Taper option in the Edit Member Data dialog, documented below.

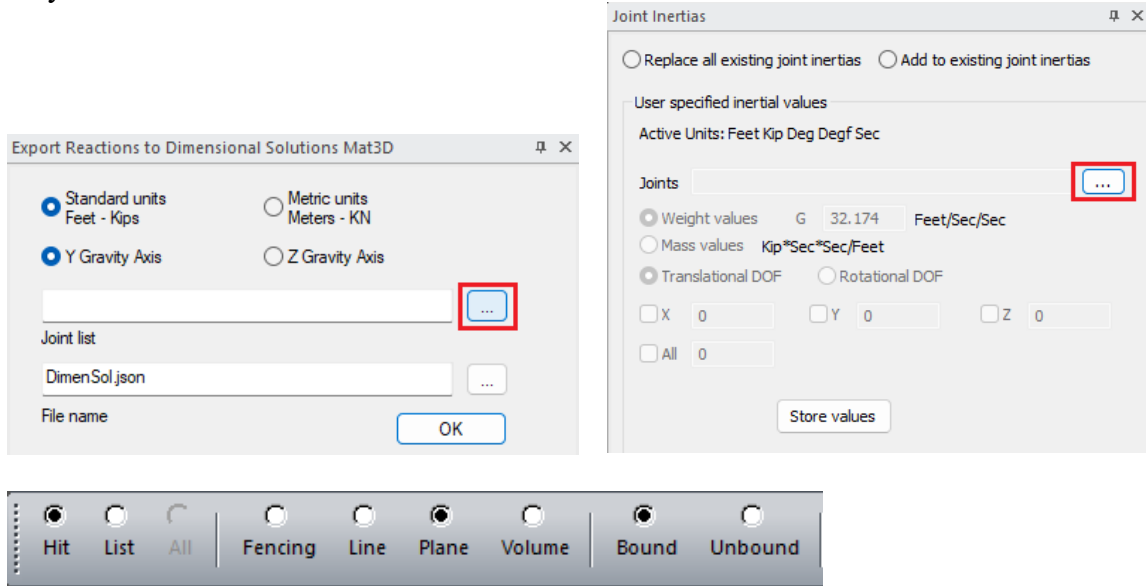
2. Drag-and-drop for model files and input files has been added to the GTMenu Graphics area. This is equivalent to using the File → Restore Model... or File → Read Input File... menu selection. If model data already exists, you will be queried if you want to add the data in the file to the existing model, for example, to add a loading file after verifying geometry, or if you want to delete the current model data and replace it with the contents of the file. This query, which has also been added to the File → Read Input File... operation, looks like this:



Input files are also checked to see if they are compatible with GT STRUDL processing. Input files must be plain text in ASCII, ANSI, or UTF-8 encoding. 16- or 32-bit characters are not allowed nor is formatted text, as from Word. UTF-8 files must be without BOM, which is the 'byte order marker' preceding the text content. If any invalid input file is detected, either because it violated the text restrictions or is not a text file, this message will appear, and the file will be ignored.

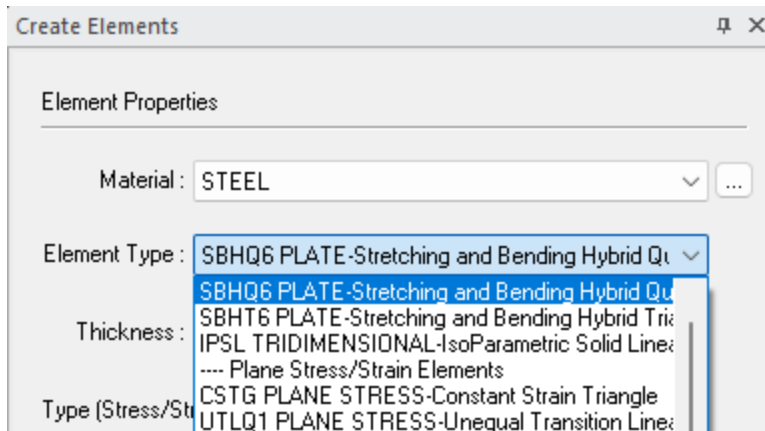


- Joint selection for Export to Mat 3D and Joint Inertias now allow Hit mode. Previously only List mode was enabled.

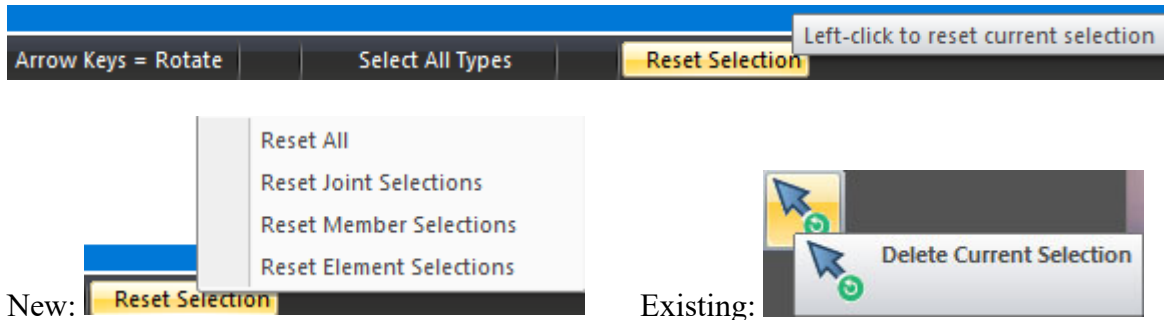


Line, Plane and Volume selections are now possible.

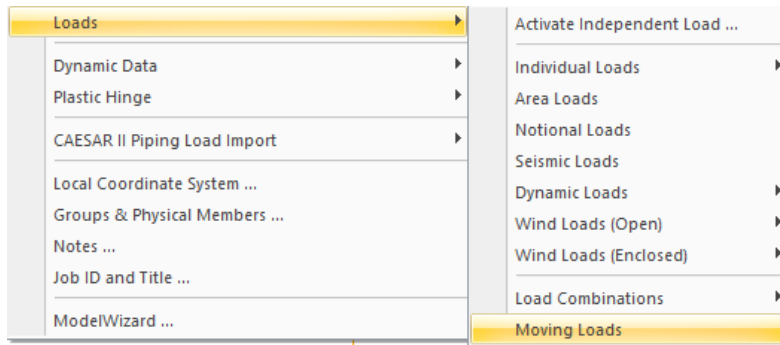
- The Element Type dropdown selection list for Create Elements has been reordered to put the most often used types at the top and the other available types organized into categories.



- Selection control equivalent to the 'Delete Current Selection' toolbar button has been added to the Status bar.

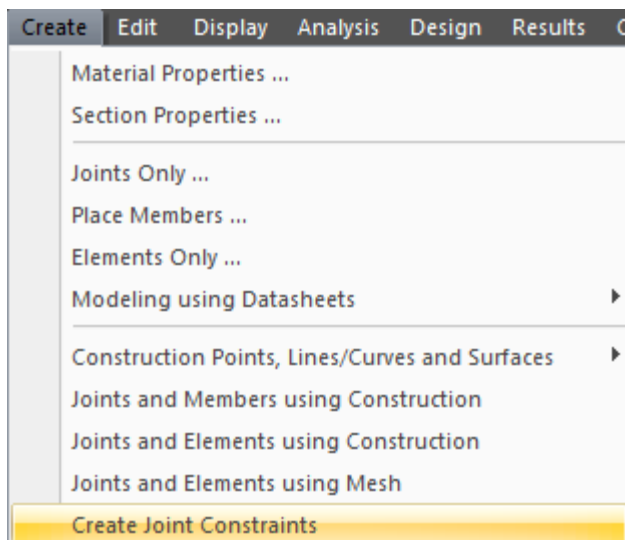


- The Moving Loads dialog has been added to the Create menu.

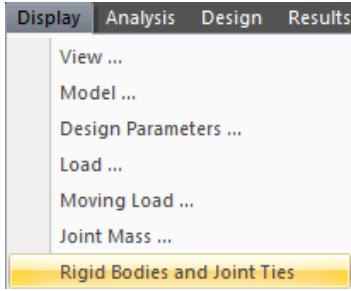


Use the Help button in the dialog for an explanation of how the dialog works.

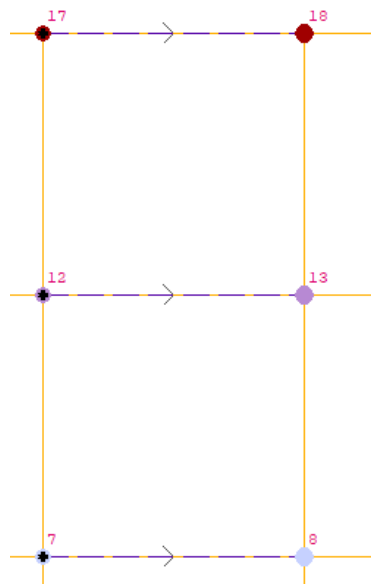
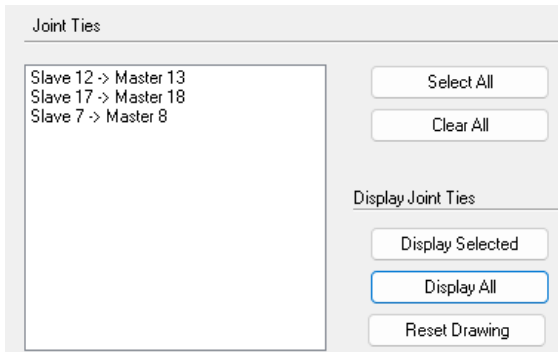
- The Joint Constraints dialog has been added to the Create menu to facilitate creating Joint Ties and Rigid Bodies (documented in Section 2.6.5, Volume 3 of the Reference Manual). Use the Help button to learn how the dialog works.



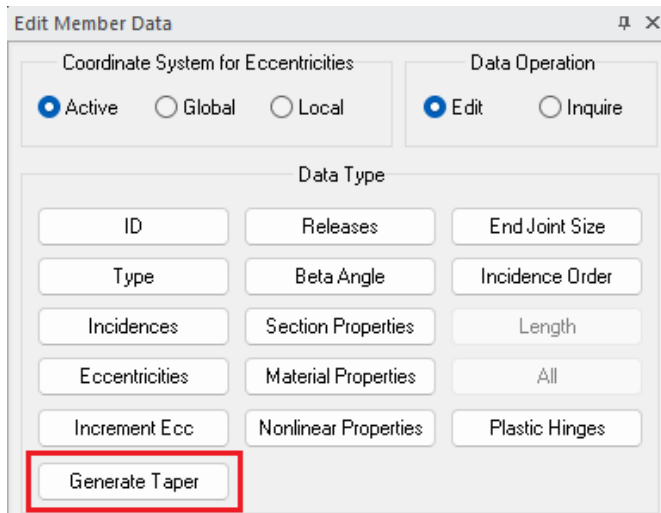
- 8. The ability to visualize Joint Ties has been added to the Display menu.



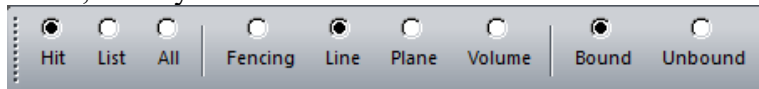
Displaying a Joint Tie will draw an arrow from the Slave joint to the Master joint.



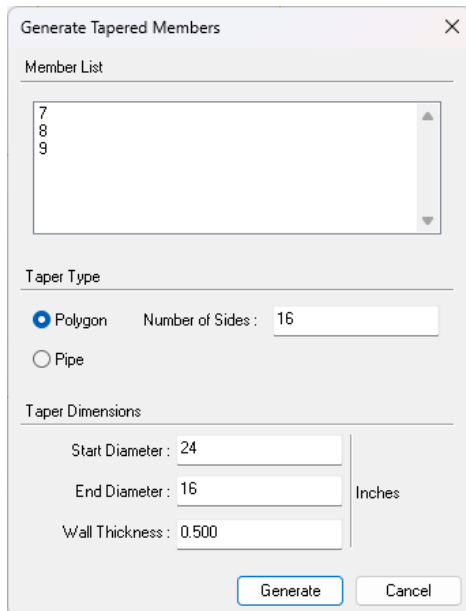
- The Generate Taper option has been added to the Edit Member Data dialog. This option can be used to assign polygonal or pipe properties to a line of members, tapering in a stepwise manner from the start to the end of the line of members. This option is the GTMenu equivalent of the new GENERATE TAPERED PROPERTIES command documented above. The line of members must conform to the requirements in Section 2.1.9.4 of Volume 1 of the Reference Manual.



Set the appropriate Length units for diameter and thickness plus the desired selection Mode, usually Bounded Line.



And then click the Generate Taper button to bring up the Generate Tapered Members dialog



Enter your data and click the Generate button and a GENERATE TAPERED PROPERTIES command will be processed. Use the Redraw Solid button if in wire-frame display to verify the new properties are correct.

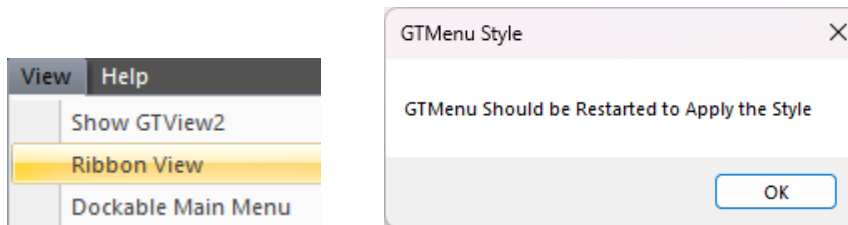


10. GTMENU POINTS, GTMENU LINES and GTMENU SURFACES commands are now allowed in input files processed while in GTMenu. Previously these commands would not be processed when reading input from GTMenu and would generate a message:

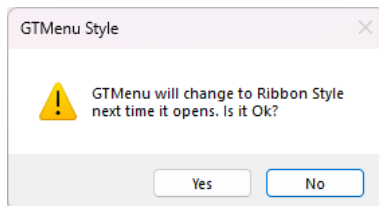
```
****INFO_CINPUT - GTMENU commands ignored when reading input files from GTMenu.
```

Note that the GTMENU SURFACE command is still a Prelease feature and will generate a Warning message.

11. A new Ribbon View has been added to GTMenu. When in the default Menu View, Ribbon View can be activated through the View menu, which will bring up the GTMenu Style prompt.

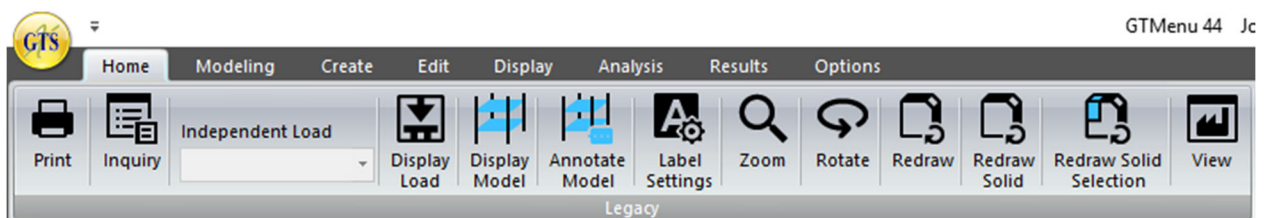


Exit GTMenu to Command Mode through File → Exit GTMenu or the frame exit X in the upper right corner. A confirmation prompt will appear.

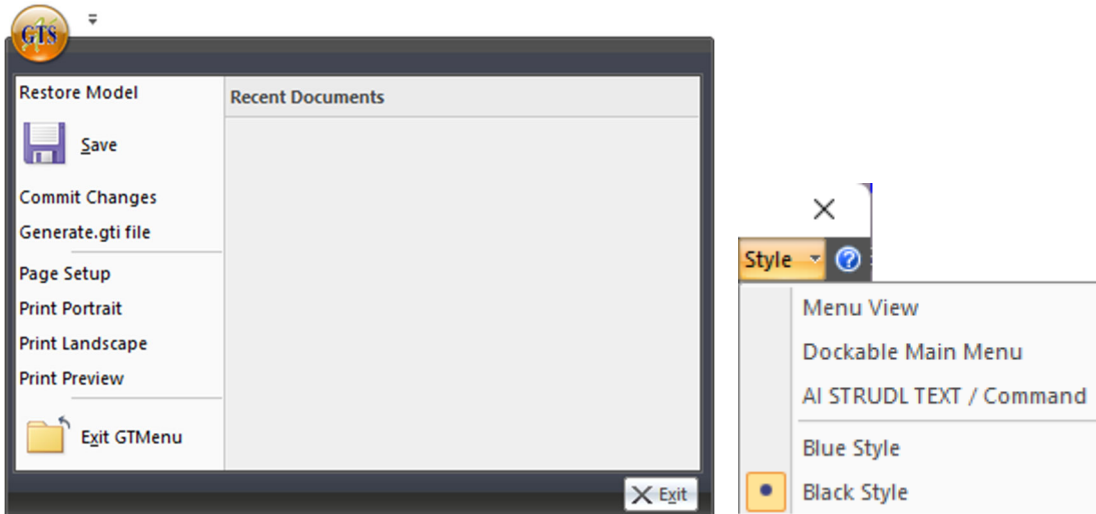


Answer Yes to complete the exit from GTMenu.

Return to GTMenu and Ribbon View will be activated and remain the GTMenu Style until changed. The Home ribbon is shown below.



A new GTS icon has been added to the upper-left corner of the application window, replacing the traditional "File" menu dropdown. Clicking on this icon provides access to all items previously found in the "File" menu. Also, a Style dropdown menu has been added in the upper, right corner.

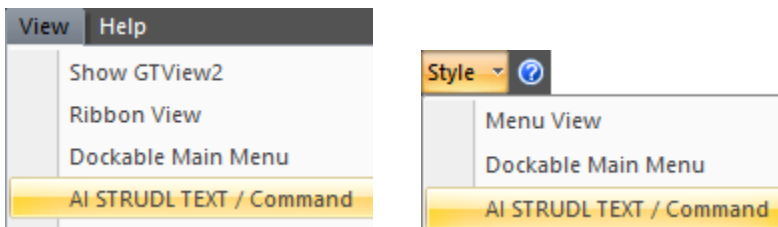


You can return to Menu View by choosing the Menu View option. Again, close and then re-open GTMenu and the default Menu View will be activated.

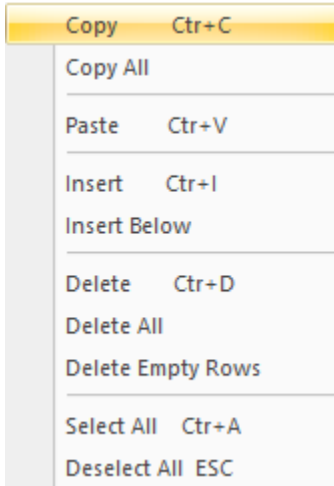
Note that the Dockable Main Menu is available from the Style dropdown, allowing you to use Ribbon and Menu access concurrently.

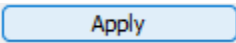
The AI STRUDL TEXT / Command option is discussed below.

12. A new option has been added to allow direct access to running input commands while in GTMenu. The AI STRUDL TEXT / Command window can be accessed from Menu View or Ribbon View.

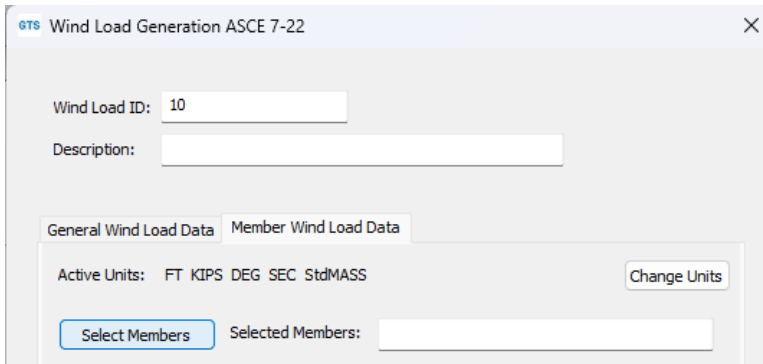


The Command window is an editable text window where you can type or paste commands. Use right-click to bring up the edit options menu.

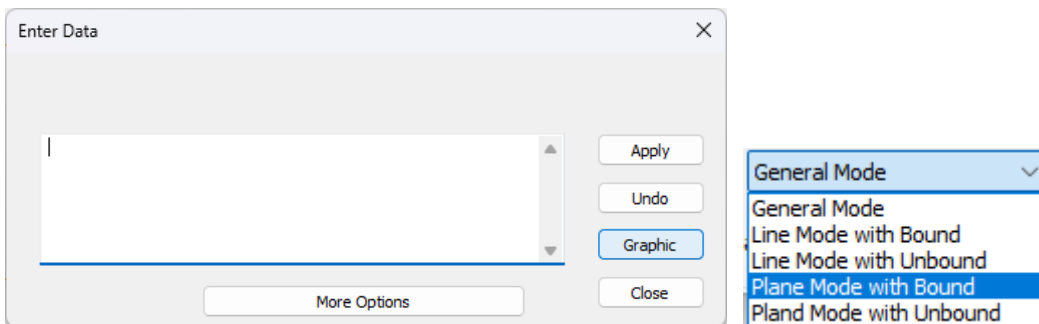


Click the  button to send the commands in the window to the GT STRUDL command processor. This action will be handled in the same manner as File → Read Input File.

- The wind loads on open structures dialog now has enhanced capabilities for member selection to apply the wind loads on.



Previously, the Select Members button only supported List mode. A Graphic option has been added to allow member selection in Hit mode, with Line and Plane options.



General Mode is member-by-member individual selection.

2.4 GTSelos

1. GTSelos has received a UI update, featuring a redesigned icon, updated background coloring, and improved alignment of interface elements. Additionally, newly implemented tooltips provide detailed explanations for each component, enhancing user understanding and navigation

2.5 Base Plate Wizard

1. The Base Plate Wizard now has an improved reporting feature that allows you to generate RTF reports that can be opened using any RTF editor. This feature can be accessed under Results → RTF Summary. Similar to text Results, the feature is greyed out if no analysis has been run.
2. The Base Plate Wizard interface has been updated for a cleaner look, but no other changes to the organization or operation of the Wizard have been made.

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 GTMenu

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

1. Using the 'Place Members by Start, Distance & Rotation' option in a view which would result in the end of the member being outside of the view bounds no longer causes a problem. This problem only occurred for restricted views and not the 4 default views.
2. Support and Section symbol labels are no longer duplicated and scrolled up the right-hand side of the graphical display. Symbol labels are displayed once until a redraw operation is requested.
3. The problem with changing member releases for more than 10 members at a time has been corrected. This problem was intermittent depending on the state of memory at the time of execution but now works properly in all cases.

3.2 General

1. RIGID BODY INCIDENCES will no longer create an element if the command is not valid due to issues with the slave joint properties. Previously, this message was issued:

```
**** WARNING_THEQU -- Master joint 18    previously specified for slave joint 17.  
          New master joint 7    is invalid. Any slave joint may have only  
          one master joint. Scan mode entered and analysis terminated.
```

The RIGID BODY element was created even though errors were reported, and the SCAN flag was set.

The message is still generated and the SCAN flag set, but the element is not created.

Documentation

Section 2.6.5.2 Rigid Bodies, Volume 3 of the GT STRUDL Reference Manual

2. The problem with a projected wind force area total of 0.0 has been corrected. A message is generated, and SCAN is turned on. The loading entry is created but no applied load data is generated.

```
**** ERROR_CMPWNDLDWF -- Wind load calculation cannot proceed for this load case.  
Total solid area is 0.0 for wind load case. No exposed area  
found to apply wind loads. Scan mode entered.
```

3. PRINT WIND LOADS no longer fails if a member is deleted after being included in a wind load.

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 43. These deficiencies have been evaluated and based on our experience, they are seldom encountered or there are workarounds. The following sections describe the known problems or deficiencies by functional area.

4.1 CAD Modeler

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

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

4.2 Finite Elements

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

4.3 General Input/Output

1. Numerical precision problems will occur if joint coordinate values are specified in the JOINT COORDINATES command with more than a total of seven digits. Similar precision problems will occur for joint coordinate data specified in automatic generation commands. (GPRF 2000.16)
2. Internal member results will be incorrect when all of the following conditions are present:
 1. Dynamic analysis is performed (response spectra or time history)
 2. Pseudo Static Loadings are created
 3. Buckling Analysis is Performed
 4. Internal member results are output or used in a subsequent steel design after the Buckling Analysis. In addition, the eigenvalues and eigenvectors from the Dynamic Analysis are overwritten by the eigenvalues and eigenvectors from the Buckling Analysis.

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

Workaround:

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

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

(GPRF 2004.14)

4.4 GTMenu

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

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

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

(GPRF 95.18)

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

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

(GPRF 95.21)

3. 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)

4. GTMenu is limited to 1,000 views. If more than 1,000 views are created, incorrect displays may occur.
(No GPRF issued)
5. 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)
6. GTMenu is limited to 10,000 Member Property Groups. If more than 10,000 property groups are created, incorrect results may occur. We have never encountered a model with more than 10,000 property groups.
(No GPRF issued)
7. The Label Structural Attributes options in the Label Settings dialog will not display if the Inquire Output dialog is open. For instance, if you have checked the Support Status option in Label Structural Attributes, the legend for the support status will disappear if the Inquire Output dialog is open.
8. When using the new Read Input File function in GTMenu, the user should check the input file (.gti file) before reading into GTMenu. In some instances, an abort could occur. At a minimum, the user should check for duplicate data such as joints, members, elements, and loadings as well as other data that could conflict with existing data already in the model in GTMenu.
9. The Generate Input process will generate incorrect commands for the one-dimensional element types NLS4PH and NLSGAP. The command will be missing required information after the TYPE specification. Users will need to manually add the missing information to create a valid input file.

This page intentionally left blank.

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 A new national annex parameter for EC3-2005 steel design code
- 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 Surface Definition Command
- 5.4.4 Export to CAESAR II

5.4.5 Import CAESAR II Pipe Loads

5.4.6 The APPLIED CARDINAL POINTS Command

We encourage the user 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 A new national annex parameter for EC3-2005 steel design code

A new national annex parameter, “Annex”, has been added to the EC3-2005 steel design code. A country name from Table 1.3-7 may be specified which indicates that the national annex of the specified country to be used for the code check or design. Parameter “Annex” is defined in the Table 1.3-1 and the country names are shown in the Table 1.3-7.

Table 1.3-1

EC3-2005 Code Parameters

<u>Parameter Name</u>	<u>Default Value</u>	<u>Meaning</u>
Annex	EC3	Parameter to specify a national annex country name which is used to automatically set the national annex parameters (e.g., GM0 (γ_{M0}), GM1 (γ_{M1}), GM2 (γ_{M2}), Beta (β), and LamdaLT0 ($\bar{\lambda}_{LT,0}$)). The default value of ‘EC3’ for this parameter indicates that the default values shown for national annex parameters GM0, GM1, GM2, Beta, and LamdaLT0 should be used. An alternative country name will reset national annex parameters to the specified country’s national standards. The country names and the parameter values associated to the specified countries are shown in Table 1.3-7. The country names that are not listed in Table 1.3-7 are accepted but a warning message is given that the EC3-2005 default values are used.

Table 1.3-7

Country Names and the National Annex Parameter Values

Country ¹	National Annex Parameter Values
EC3-2005 (defaults)	GM0 = 1.0, GM1 = 1.0, GM2 = 1.25 Beta = 0.75, LamdaLT0 = 0.4
Cyprus, Greece, Netherlands², Slovenia, Spain, and Sweden use above EC3-2005 default values	
Belgium	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Bulgaria	GM0 = 1.05, GM1 = 1.05
Denmark	GM0 = 1.1, GM1 = 1.2, GM2 = 1.35
Finland	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
France	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Germany	GM1 = 1.1
Italy	GM0 = 1.05, GM1 = 1.05 Also see Table 1.3-8 for lateral torsional buckling curve for cross-sections using equation (6.57) of the 1993-1-1:2005(E)
Malaysia	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Norway	GM0 = 1.05, GM1 = 1.05
Poland	GM2 = 0.9(f_u / f_y) 1.1

Note: National annex parameters with different values from the EC3-2005 defaults are shown in Table 1.3-7 for each country.

- 1 The country names that are not listed in Table 1.3-7 are accepted but a warning message is given that the EC3-2005 default values are used.
- 2 Country names more than 8 characters are stored and displayed based on the first 8 characters.

Table 1.3-7 (continued)

Country Names and
the National Annex Parameter Values

Country ¹	National Annex Parameter Values
EC3-2005 (defaults)	GM0 = 1.0, GM1 = 1.0, GM2 = 1.25 Beta = 0.75, LamdaLT0 = 0.4
Portugal	Beta = 1.0, LamdaLT0 = 0.2
Singapore²	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
UK (United Kingdom)	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2 Also see Table 1.3-9 for lateral torsional buckling curve for cross-sections using equation (6.57) of the 1993-1-1:2005(E)

Note: National annex parameters with different values from the EC3-2005 defaults are shown in Table 1.3-7 for each country.

- 1 the country names that are not listed in Table 1.3-7 are accepted but a warning message is given that the EC3-2005 default values are used.
- 2 Country names more than 8 characters are stored and displayed based on the first 8 characters.

Table 1.3-8

**Lateral torsional buckling curve for cross-sections
using equation (6.57) of the 1993-1-1:2005(E)
Italy**

Cross-section	Limits	Buckling curve	
Rolled I cross-sections	$h/b \leq 2$	b	0.34
	$h/b > 2$	c	0.49
Welded I cross-sections	$h/b \leq 2$	c	0.49
	$h/b > 2$	d	0.76
For all other cross-sections		d	0.76

Table 1.3-9

**Lateral torsional buckling curve for cross-sections
using equation (6.57) of the 1993-1-1:2005(E)
UK (United Kingdom)**

Cross-section	Limits	Buckling curve	
Rolled doubly symmetric I and H sections and hot-finished hollow sections	$h/b \leq 2$	b	0.34
	$2 < h/b \leq 3.1$	c	0.49
	$h/b > 3.1$	d	0.76
Angles (for moments in the major principal plane)		d	0.76
All other hot-rolled sections		d	0.76
Welded doubly symmetric sections and cold-formed hollow sections	$h/b \leq 2$	c	0.49
	$h/b > 2$	d	0.76

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.

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 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 if 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.

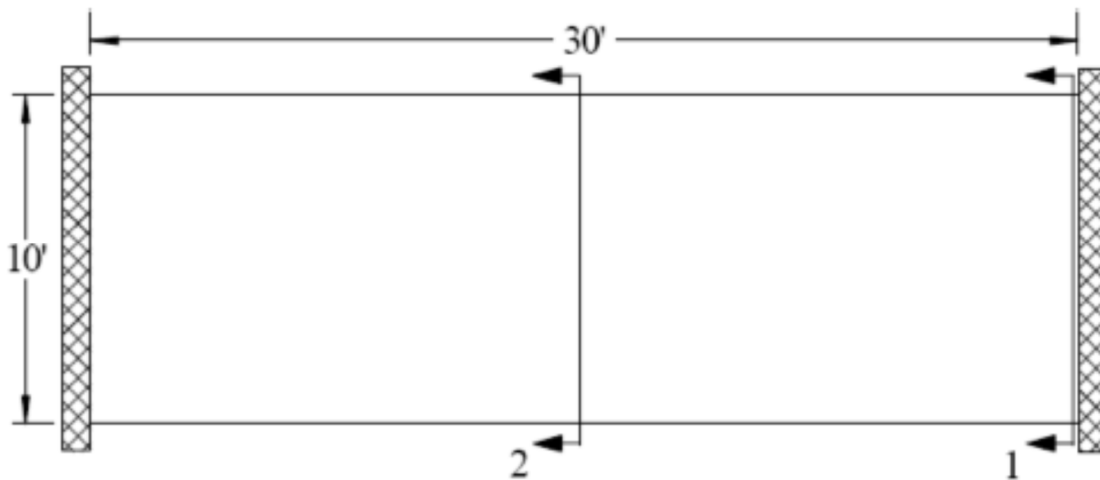


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

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 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.

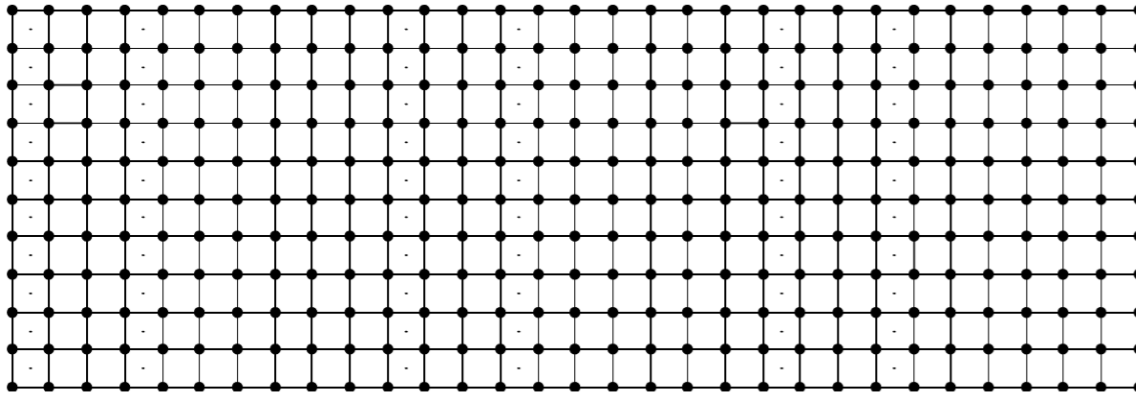


Figure 5.2.3-2 Example Finite Element Model

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 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 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:

CALCULATE ERROR (ESTIMATE) (BASED ON) -

* $\left. \begin{array}{l} \text{ENERGY (NORM)} \\ \text{MAX DIFFERENCE} \\ \text{DIFFERENCE FROM AVERAGE} \\ \text{PERCENT MAX DIFFERENCE} \\ \text{PERCENT DIFFERENCE FROM AVERAGE} \\ \text{NORMALIZED PERCENT MAX DIFFERENCE} \\ \text{NORMALIZED PERCENT DIFFERENCE FROM AVERAGE} \end{array} \right\} -$

(AT) * $\left. \begin{array}{l} \text{TOP} \\ \text{MIDDLE} \\ \text{BOTTOM} \end{array} \right\} (\text{SURFACES}) (\text{FOR}) \left\{ \begin{array}{l} \rightarrow \text{ALL} \\ \text{ELEMENT list} \end{array} \right\}$

The results from this command provide an estimate of the errors in the finite element discretization of the problem. Energy norm (L2 norm) and nodal error estimates are available.

The L2 norm is given by:

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

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

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

where N is the shape functions used for the assumed displacement field of the element. The stress norm uses the average stresses and is given by:

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

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

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

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

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

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

Maximum Difference Method

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

Difference from Average Method

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

Percent Maximum Difference Method

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

Percent Difference from Average Method

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

Normalized Percent Maximum Difference

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

Normalized Percent Difference from Average Method

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

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

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

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

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

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command**General form:**

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

Explanation:

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

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

**** WARNING_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 1o.

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:

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

Elements:

- $i_R / 'a_R'$ = integer or alphanumeric name of the existing independent loading condition whose global components are to be rotated.
- r_1, r_2, r_3 = values in current angle units of the load component rotation angles $\theta_1, \theta_2, \theta_3$ as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

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

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

and illustrated in Figure 2.1.7-1.

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

Taking the specified loading *ir/ar*, 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:

$$\text{REFERENCE (COORDINATE) (SYSTEM) } \left. \begin{array}{l} \{i_1\} \\ \{a_1\} \end{array} \right\} - \left. \begin{array}{l} (\text{ORIGIN } [\underline{X}] v_x [\underline{Y}] v_y [\underline{Z}] v_z) (\text{ROTATION } [\underline{R1}] v_1 [\underline{R2}] v_2 [\underline{R3}] v_3) \\ \left\{ \begin{array}{l} \text{JOINT } \{i_2\} \\ \{a_2\} \\ \underline{X} v_4 \underline{Y} v_5 \underline{Z} v_6 \end{array} \right\} \left\{ \begin{array}{l} \text{JOINT } \{i_3\} \\ \{a_3\} \\ \underline{X} v_7 \underline{Y} v_8 \underline{Z} v_9 \end{array} \right\} \left\{ \begin{array}{l} \text{JOINT } \{i_4\} \\ \{a_4\} \\ \underline{X} v_{10} \underline{Y} v_{11} \underline{Z} v_{12} \end{array} \right\} \end{array} \right\}$$

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 'a1' 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 R1, R2, and R3 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:

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

Explanation:

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

5.4.3 GTMenu SURFACE DEFINITION Command

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

General Form:

GTMenu SURFACE DEFINITION

$$\begin{aligned} & \{ 'a_1' \} \text{ surface-specs}_1 \\ & \cdot \\ & \cdot \\ & \cdot \\ & \{ 'a_n' \} \text{ surface-specs}_n \end{aligned}$$

Elements:

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

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

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

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

where,

'a₁', 'a₂', ..., 'a_n' = 1 to 8 character alphanumeric Surface IDs beginning with S (i.e. S1, S2).

iu, iv = integer values representing the number of drawing

segments to use in directions U and V respectively.

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

Examples:

```

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

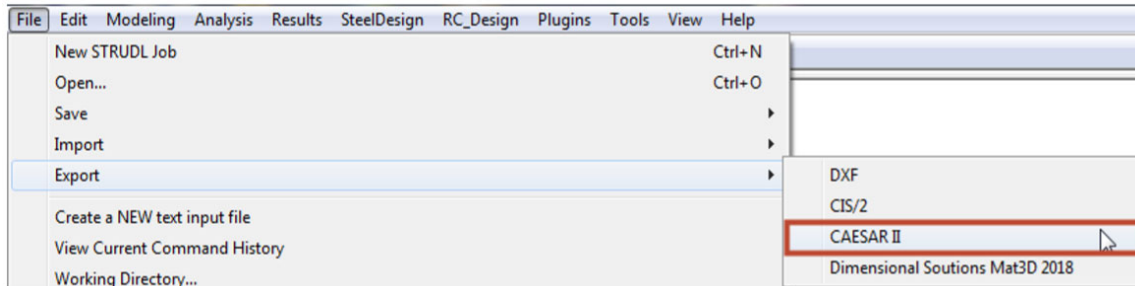
  'S2' SURFACE OF REVOLUTION SPACING 10 20 -
      REVOLUTION ANGLE 60.5 -
      AXIS POINTS 'P1' 'P6' -
      U CURVE 'C2'

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

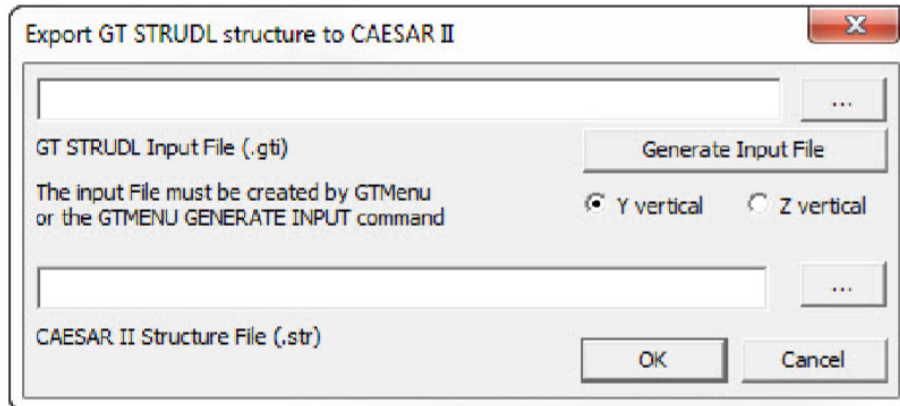
```

5.4.4 Export to CAESAR II

You can export the model from GTShell (Command Window) and from CAD Modeler to CAESAR II. In GTShell this feature is available under the File pulldown menu as shown below:

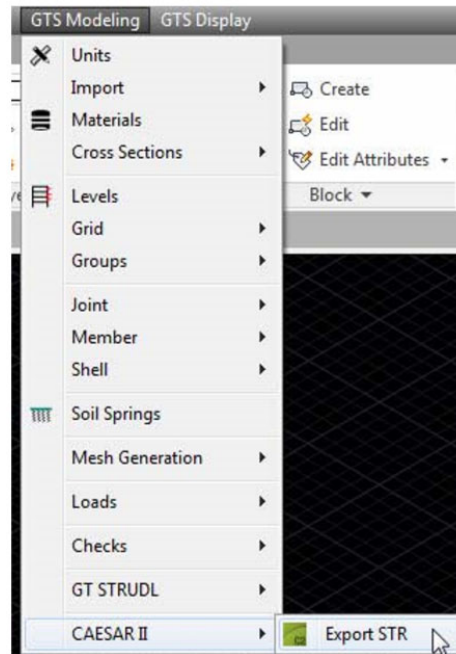


After selecting CAESAR II, the following dialog will pop-up:



This feature will convert a GT STRUDL input file into one or more CAESAR II structure files. The input file must have been created in GTMenu or by using the GTMENU GENERATE INPUT command to ensure a readable (by the translator program) syntax. Note that the GT STRUDL input file must have a ".gti" extension and the specified CAESAR II structural file must have a ".str" extension. You can use the 'Generate Input File' button to create a suitable input file from the current GT STRUDL model. Select the appropriate vertical axis (Y or Z) to generate correct Beta angles for the CAESAR II file. Then click the OK button.

In CADModeler, the current drawing can be exported to the CAESAR II Modeler (.str file) from the GTS Modeling pulldown as shown below:



The current drawing can also be exported to into CAESAR II Modeler (.str file) by typing GTSEXPSTR at the command prompt. Immediately after the log file appears on the screen. A typical log file is:

```
GT STRUDL Version 2018.R1
GTS2CII Version 2018.R1.01
GTS2CII Binary Dir
C:\Program Files (x86)\GTStrudl\2018R1\Utilities\GTS2CII\
Project Dir F:\HexagonPPM\CaesarII\PlantStructure\
Total Number of Sections: 6
Total Number of Joints: 170
Total Number of Members: 233
The model will be saved in 1 STR file(s)
File
F:\HexagonPPM\CaesarII\PlantStructure\PStructure_0708_01.str created
```

If the cross sections used in CAD Modeler (and GT STRUDL) are not available in CAESAR's section library, a warning message such as the one shown below will appear:

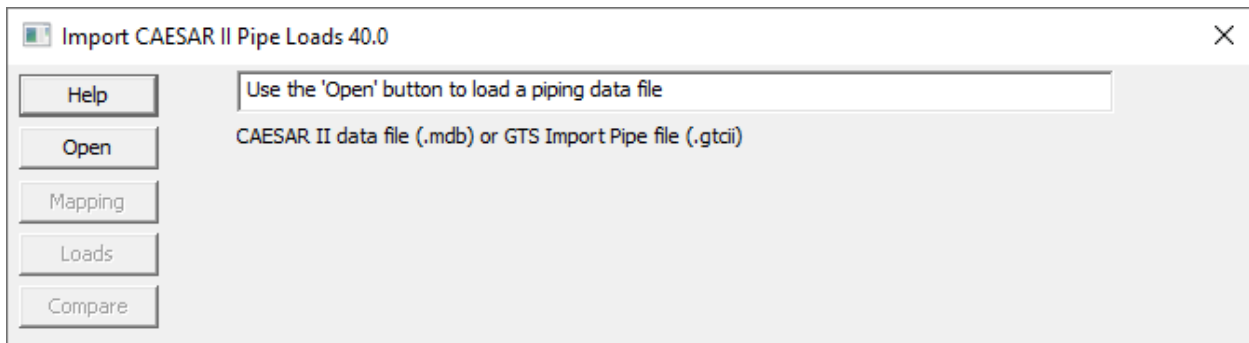
```
WARNING: Section L1x1x1/4 is not available in CII, please use another one
or edit
F:\HexagonPPM\CaesarII\PlantStructure\PStructure_0708_01.str      file
manually
```

The .str file in CAESAR II is limited to 500 members so if your structure contains more than 500 members, the Export function will automatically break the model into 500 member .str files with a limit of 10 such files (maximum of 5000 members in your structure).

5.4.5 Import CAESAR II Pipe Loads

You can import loads from CAESAR II pipe stress analysis into a GT STRUDL analysis model. Reactions from a CAESAR II .mdb file can be assigned to a GT STRUDL member and applied as a concentrated member load at a designated location. Be sure to export as “.mdb” from CAESAR II. While only a single piping system from each .mdb file can be processed, you can add as many loads from piping systems as needed to a GT STRUDL model by appending the generated loading commands into a single, integrated file for processing by GT STRUDL.

Access to the “Import CAESAR II Pipe Loads” dialog is through the File → Import menu in the GT STRUDL Command window, or through the GTMenu Create → CAESAR II Piping Load Import pick. Click the Help button for information about using the Import CAESAR II Pipe Loads feature.



5.4.6 APPLIED CARDINAL POINTS

A new feature to retain Cardinal Points (point on the section profile that is used to place the member) when importing models from CIS/2 has been added as a Pre-Release feature. The Import CIS/2 process will add the new APPLIED CARDINAL POINTS commands for members that have been designated to be placed by a cardinal point other than the centroid. APPLIED CARDINAL POINT information is retained only for the purpose of exporting CIS/2 files and has no effect on the model geometry.

Since GT STRUDL only uses the centroidal axis to place members, MEMBER ECCENTRICITIES commands are used to correctly position the member upon import. Previously, when the model was exported through CIS/2 all cardinal point information was lost. Now, for members with APPLIED CARDINAL POINTS specified, upon export the original cardinal point is used in the generated CIS/2 .stp file. The eccentricities used to position the member due to cardinal point placement are not included in the generated file, but any specified CIS/2 offsets are retained. This allows programs, such as Smart3D and CADWorx Structure, to create a standard physical model with 'top of steel' or other placement points set correctly from a GT STRUDL CIS/2 file.

```
Syntax;
APPLIED CARDINAL POINTS
  list CP ncp [X] vx [Y] vy [Z] vz
  ...
```

Where ncp = cardinal point number. Cardinal points 1 to 10 are accepted.
vx, vy, vz = the global eccentricities due to cardinal point placement

PRINT CARDINAL POINTS (*list*)

Print the existing cardinal point information. If *list* is omitted, the cardinal point information is printed for all members.

DELETIONS

```
APPLIED CARDINAL POINTS (list)
(list)
```

ADDITIONS

Delete cardinal point information for members specified in *list*. Note that if you delete the cardinal point information for any member in a Physical Member, all analytical members in the Physical Member will have cardinal point information deleted to maintain Physical Member integrity.