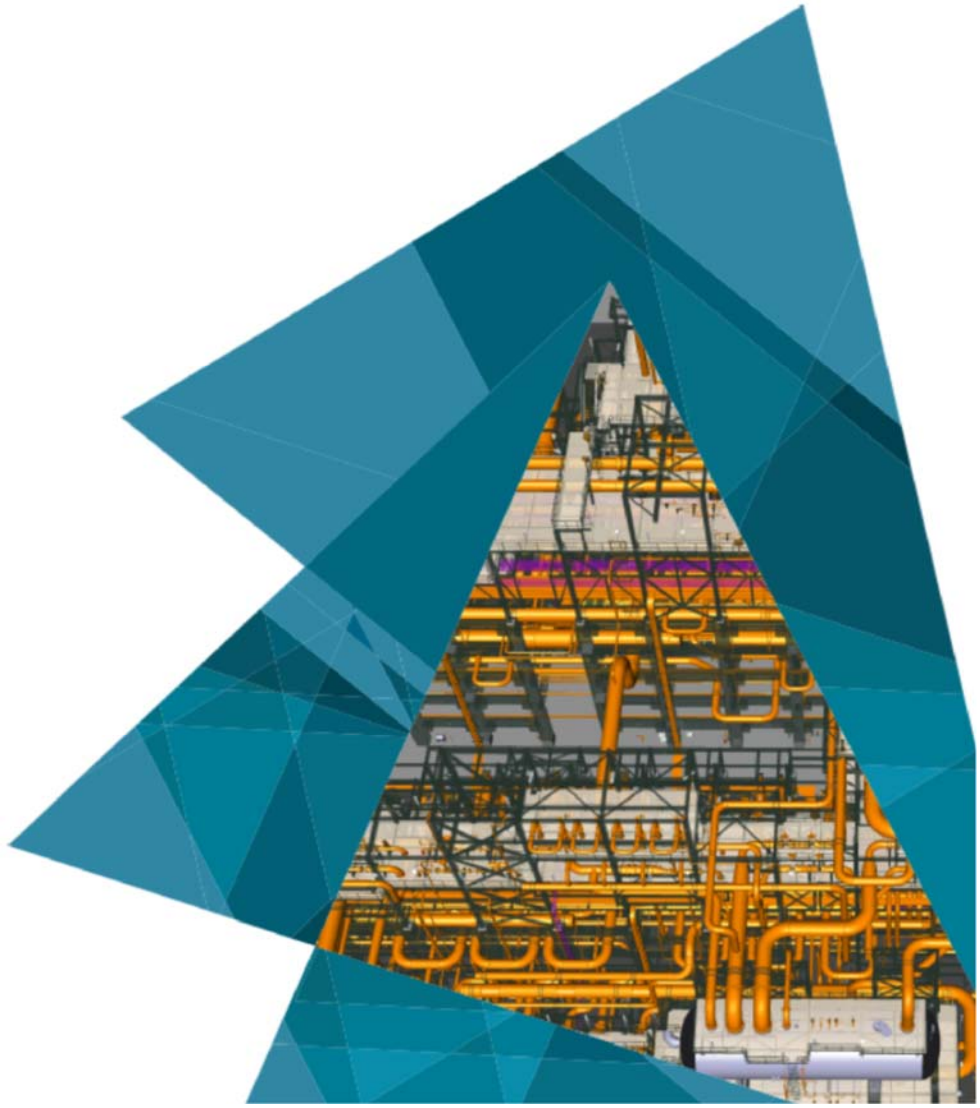
GT STRUDL® Version 41
Release Guide



Release Date: November 2022

HEXAGON
PPM

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

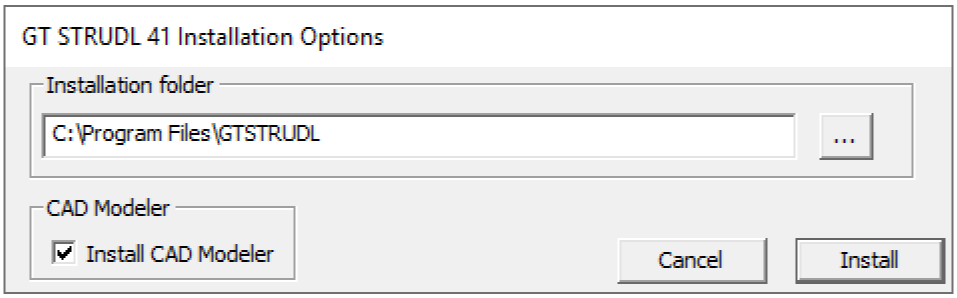
Intergraph®, the Intergraph logo®, Intergraph Smart®, SmartPlant®, SmartMarine®, SmartSketch®, SmartPlant Cloud®, PDS®, FrameWorks®, I-Route, I-Export, Isogen®, SPOOLGEN, SupportManager®, SupportModeler®, SAPPHIRE®, TANK, PV Elite®, CADWorx®, CADWorx DraftPro®, GTSTRUDL®, and CAESAR II® are trademarks 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, ProtoViewCtl, 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 1

Introduction

Version 41 covers GT STRUDL operating on PCs under the Windows 10 x64 based operating systems. Installation requires approximately 2.5 GB of storage for GT STRUDL 41. To interactively install GT STRUDL 41, run Setup.exe. Select an installation folder and choose if you wish to install CAD Modeler at this time by checking or clearing the check box. Click Install and choose a language of the Hexagon PPM licensing agreement, and then click the Proceed button when it appears. See the ReadMe.txt file on the CD for information about installing GT STRUDL 41 in batch mode.



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 from <*installation folder*>\41\CADModeler. AutoCAD® or BricsCAD® must be installed on your computer to be able to use CAD Modeler.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 40. 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 Version 40 release.

Chapter 4 describes known problems with Version 41.

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.

Chapter 2

New Features in Version 41

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 41 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. Additional finite element integrity checking has been added to the consistency phase of stiffness matrix calculation. A more comprehensive check of ELEMENT INCIDENCE data is done to verify that the number of nodes specified for an element corresponds to the specified element type. Incompatible information will generate an error message and turn on the SCAN flag, halting analysis.

   Planar elements will also have new geometrical checking for interior angles, aspect ratio and, for quadrilateral elements, nonplanarity. These geometrical checks will generate warnings and will not terminate analysis. If such warnings are given, it is the user's responsibility to determine if the limit violation is acceptable. See the next entry for information about controlling planar geometry checking.

   Consistency checking without analysis can be requested with the CHECK CONSISTENCY command, Section 2.1.12.4, Volume 1, GT STRUDL Users Reference Manual.

   <u>Examples of the new messages</u>:

```
**** ERROR_FEcheck1 - Element 4        has  5 incidences. Type SBHQ6    requires  4 incidences.
                  SCAN mode is entered.
**** WARN_STDSY1 -- SCAN mode is ON.  Analysis will be halted.


**** WARN_FEcheck1 - Element 1        has an interior angle less than 20.00 degrees.

**** WARN_FEcheck1 - Element 1        has an interior angle greater than 135.00 degrees.

**** WARN_FEcheck1 - Element 2        is more than  5.00 percent out-of-plane.

**** WARN_FEcheck1 - Element 1        has an aspect ratio of  11.0 which is greater than  10.0
```

2. A new command has been added to control planar geometry checking, described above. By default, planar element geometry checking is enabled. To turn off geometry checking, use the ELEMENT LIMITS OFF command as documented below.

$$\underline{\text{ELE}}\text{MENT }\underline{\text{LIMI}}\text{TS}\left\{\begin{array}{l}\underline{\text{OFF}}\\ \underline{\text{ON}}\\ \underline{\text{ANGLE}}\left\{\begin{array}{l}\underline{\text{MAX}}\ v_1\\ \underline{\text{MIN}}\ v_2\end{array}\right\}\\ \underline{\text{ASP}}\text{ECT }(\underline{\text{RAT}}\text{IO})\ v_3\\ \underline{\text{PLA}}\text{NE }(\underline{\text{TOL}}\text{ERANCE})\ v_4\end{array}\right\}$$

Documentation:

Section 2.1.12.23 The ELEMENT LIMITS Command, Volume 1, GT STRDUL Users Reference Manual.

3. The WARNING message that was generated when the INCLUDE FINITE ELEMENTS option was given for the DEAD LOAD or SELF WEIGHT commands is now only generated if the problematic element types exist at the time of the command. For version 40 and earlier:

```
{   88} > DEAD LOAD 1 DIR -Y INCLUDE FINITE ELEMENTS ALL MEMBERS

**** WARNING_STDLCK -- FINITE ELEMENTS are included in the automatic computation
                       of element/joint self-weight loads by the DEAD LOAD
                       command.  Element/joint self-weight loads will be computed
                       for all finite elements except PSRR, UTLQ1, IPCABLE,
                       NLS, NLS4PH, and base isolation elements.
```

Version 41 will not include the warning if PSRR, UTLQ1, IPCABLE, NLS, NLS4PH, or base isolation elements are not included in the current model.

2.2 Loadings

1. New codes have been added to CREATE AUTOMATIC LOAD COMBINATIONS DESIGN. The new code options are ASCE716A, ASCE722L and ASCE722A. ASCE716A is used to create combinations based on the ASCE 7-16 ASD (Allowable Stress Design) standard. ASCE722L is used to create combinations based on the ASCE 7-22 LRFD (Strength Design) standard. ASCE722A is used to create combinations based on the ASCE 7-22 ASD (Allowable Stress Design) standard.

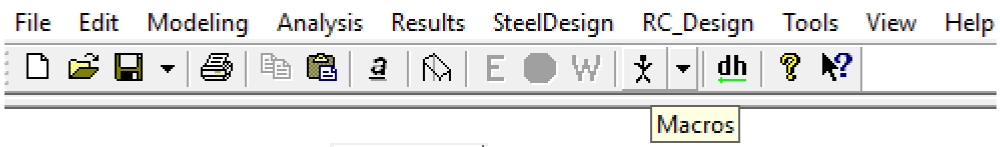
   Documentation:
   2.1.11.3.6.4 The CREATE AUTOMATIC LOAD COMBINATION DESIGN Command

2. Type WT has been added to the STORE DESIGN LOAD VARIABLES types. WT indicates a tornado wind load to be used with ASCE 7-22 design loads.

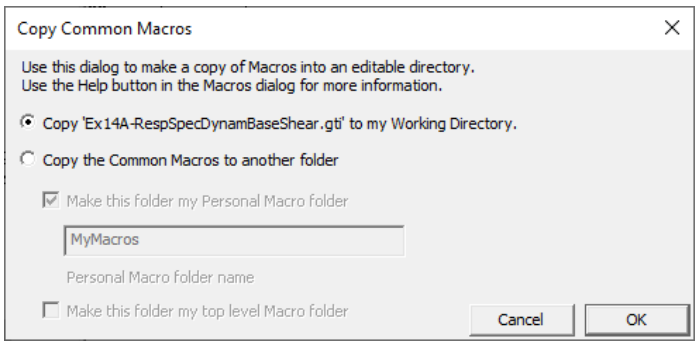
   Documentation:
   2.1.11.3.6.3 The STORE DESIGN LOAD VARIABLES Command

2.3 GT STRUDL Output Window (GTShell)

1. A new function has been added to the Macros dialog to easily copy files from the installed Common Macros to your folder. Either the currently selected Macro or the entire Common Macro folder may be copied. This will allow you to edit and save the macro files.



From the Macros dialog: 



2. The ModelWizard has several improvements.

A Physical Member option has been added to the member wizards Plane Frame, Braced Frame, Space Frame and Truss Bridge. This will add a DEFINE PHYSICAL MEMBER command for each member generated by the wizard, or in the case of the Braced Frame Wizard for all the analytical members between columns.

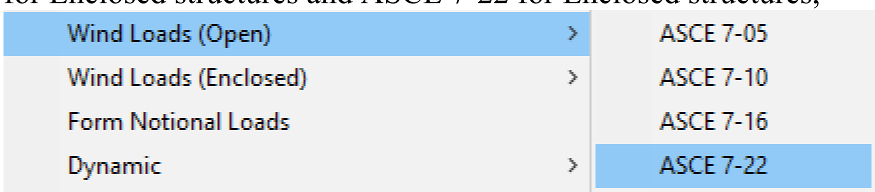
A Stiffness Analysis option has been added to all wizards. This will run the regular in-core analysis engine allowing you to go directly to viewing results without having to run the analysis yourself. This option will not be offered if no loadings are applied in the wizard.

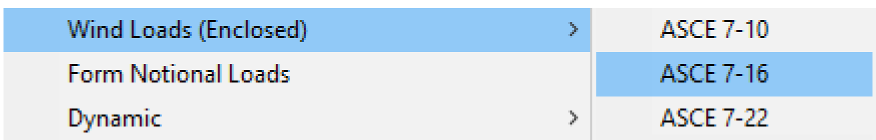
The Space Frame Wizard now offers a ‘Z up’ option. If chosen, additionally the command GTMENU VIEW Z UP is added so when you enter GTMenu the correct orientation is shown. Also, the option to add beam loads and transverse joint loads has been added to match the options in the Plane Frame Wizard.

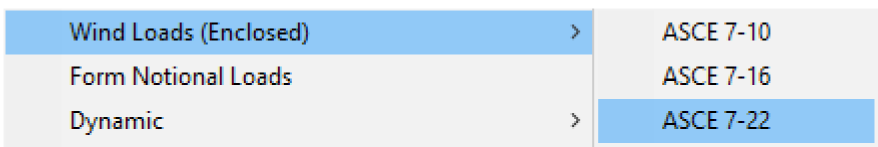
Default profiles are now taken from Table W-AISC15. Previously they came from WBEAM9 and WCOLUMN9, tables based on the 1989 AISC 9th Edition.

The ModelWizard DLL format has been changed from ActiveX to a standard C++ DLL for better compatibility with GTMenu (see below). The appearances of the dialogs have changed but functionality, other than the improvements noted above, are unchanged.

3. Three new Wind load types have been added. ASCE 7-22 for Open structures, ASCE 7-16 for Enclosed structures and ASCE 7-22 for Enclosed structures,







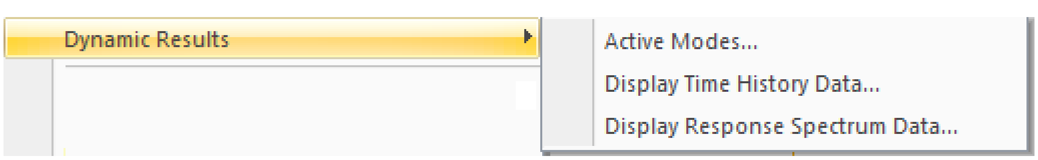
2.4 GTMenu

1. The top menu bar has been changed from

to

The “Coordinate System” menu has been moved to the Display menu. “Utilities” has been changed to “External Links”.

2. Dynamic Results has been added to the Results menu

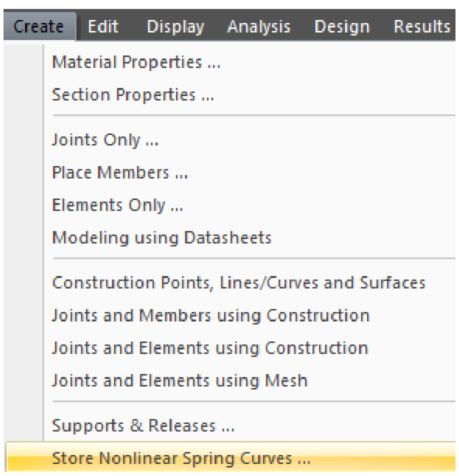


These dialogs are the same as the ones available in Command mode.
Active Modes is used to inactivate inconsequential modes prior to modal analysis results.
Display Time History will display plots of existing Time History data.
Display Response Spectrum will display plots of existing Response Spectrum data.

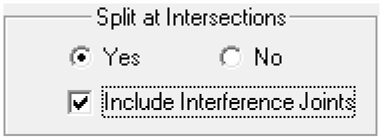
3. Nonlinear Spring creation and review have been added to GTMenu.

Display Nonlinear Spring Curves has been added to the Display Model dialog. Select a curve from the dropdown list and click the Display button to bring up the Plot dialog where you edit titles and axis labels, plus print or send to the Scope Editor.

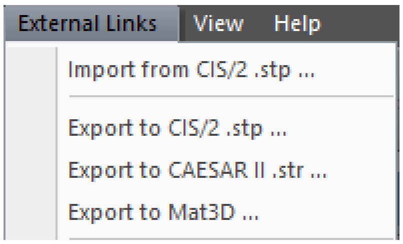
Store Nonlinear Spring Curves has been added to the Create menu. This is the same dialog as is available in Command mode. You can create, review and plot nonlinear spring curves from this dialog.



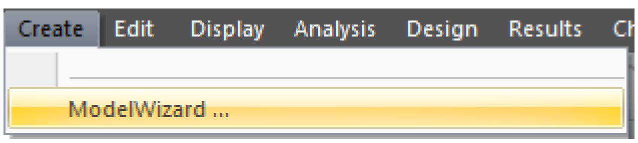
4. A new option has been added to the Create → Place Members dialog to allow splitting at interference joints. Interference joints are joints that lie within a specified tolerance of the new member's center line but are not the start or end of another member. For example, when placing a new member that spans a set of finite element incidences, the element incidences would be detected as interference joints. Previously the element incidence joints would be ignored. You may choose to include interference joints by checking the box when 'Split at Intersections' is set to 'Yes'.



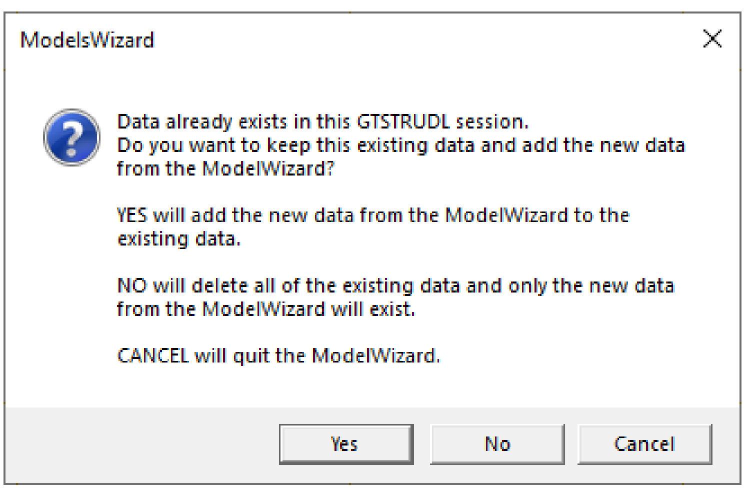
5. **Import from CIS/2** has been added to the new External Links menu. **Export to CIS/2**, **CAESAR II** structural format (.str) and **Mat 3D** from Dimensional Solutions have been added to the new External Links menu. Choosing these options will bring up the same dialogs that are available from the File menu in Command mode.



6. The Model Wizard is now available from GTMenu, from the bottom of the Create menu.



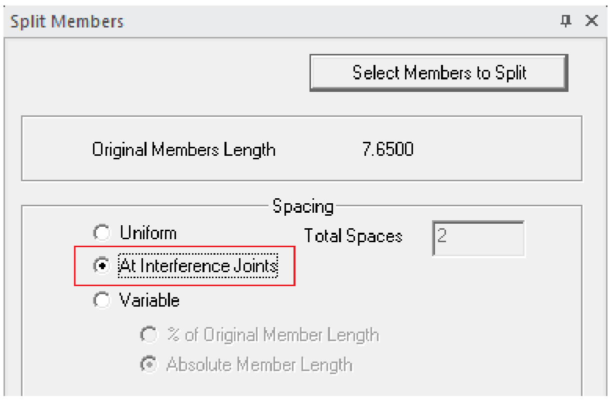
All of the changes and improvements to the Model Wizard noted above are included. When you use the Model Wizard, at completion all data is committed so that the GTMenu data base and the Command mode data base are equal. If there is an existing model, the following query will be asked, as it is in Command mode.



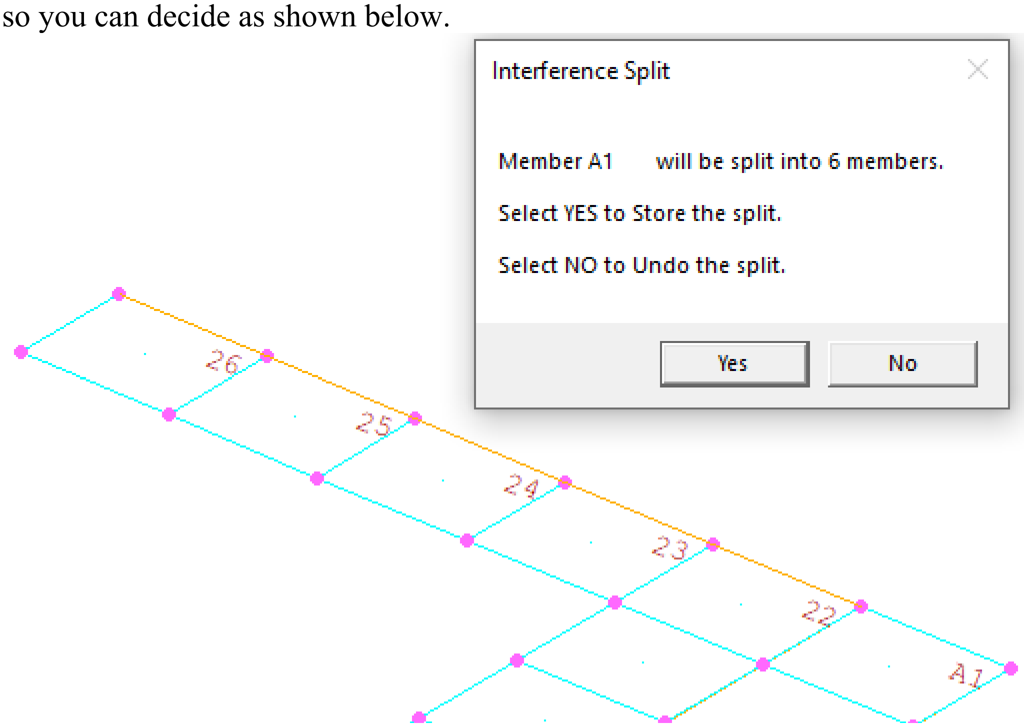
If 'Yes' is chosen to retain the existing data, the assumed starting names for joints, members, elements and loads will be set to the next available GTMenu integer name. You can choose to override the starting names if you wish.

If you choose 'No' to delete the existing data, it will be deleted upon completion of the Model Wizard without a further prompt.

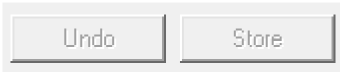
7. A new option has been added to the Edit → Split Members… dialog to allow you have GTMenu detect Interference Joints and split the selected members at those points. Interference Joints are joints that lie close to the member's centroid but are not the start or end joints. This feature is especially useful after a finite element mesh has been added to a frame structure.



Because the number of spacings due to Interference Joints is not predetermined as it is for Uniform and Variable, for each selected member you will be queried to Store or Undo the split as the selected member list processed. You will be shown the newly created members so you can decide as shown below.

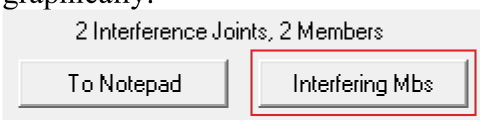


Unlike Uniform and Variable splitting, the Undo and Store buttons will not be available after splitting at Interference Joints.

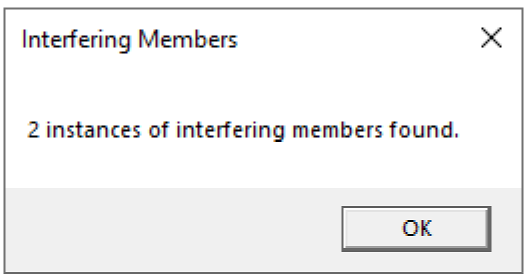


As with Uniform and Variable splitting, Physical Member definitions will be maintained after Store so that the Physical Member will contain the newly created analytical members.

8. The **Interference Joints** option in the **Check → Duplicates & Floating Joints** dialog has a new button **Interfering Mbs**. This new feature is used to find modeling problems in a large structure by identifying member-to-member problems which can be difficult to find graphically.



Click the **Interfering Mbs** button to check the list of interfering joints to find where the start or end joint of a member lies on the centroid of another member. A message will appear reporting the number of such cases.



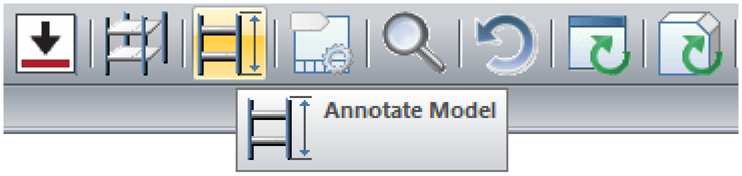
If there are Interfering members, a report is generated in the Working Directory named "InterferingMembersReport.txt". A new **Member Report** button will appear. Click this button to view the report using Notepad.

The report contains information to help locate problems member by member. As an example, a structure with two overlapping members:

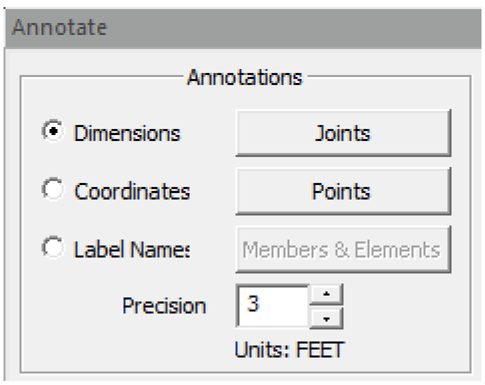
Generates this report:

```
Member   Joint     S/E  interferes with Member   Overlap?
-------- -------- -----                 -------- ---
2        2        Start                 1        Yes
1        3        End                   2        Yes
```

Note that overlapping members will be denoted in the report with a 'Yes'. This situation can be very difficult to find graphically. You can locate members by name using the LM hot button (type LM when focus is in the graphics) or the Annotate dialog in List mode:

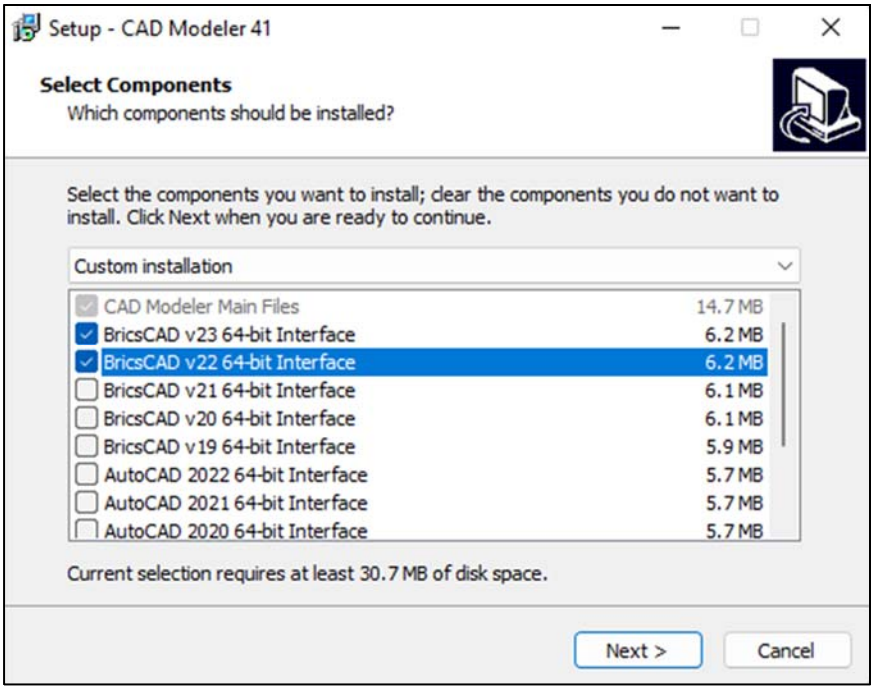


9. The Annotate dialog (see toolbar image above) now allows you to set the desired number of decimal places to use when annotating Dimensions or Coordinates. Previously the precision was fixed as 2.



2.5 CAD Modeler

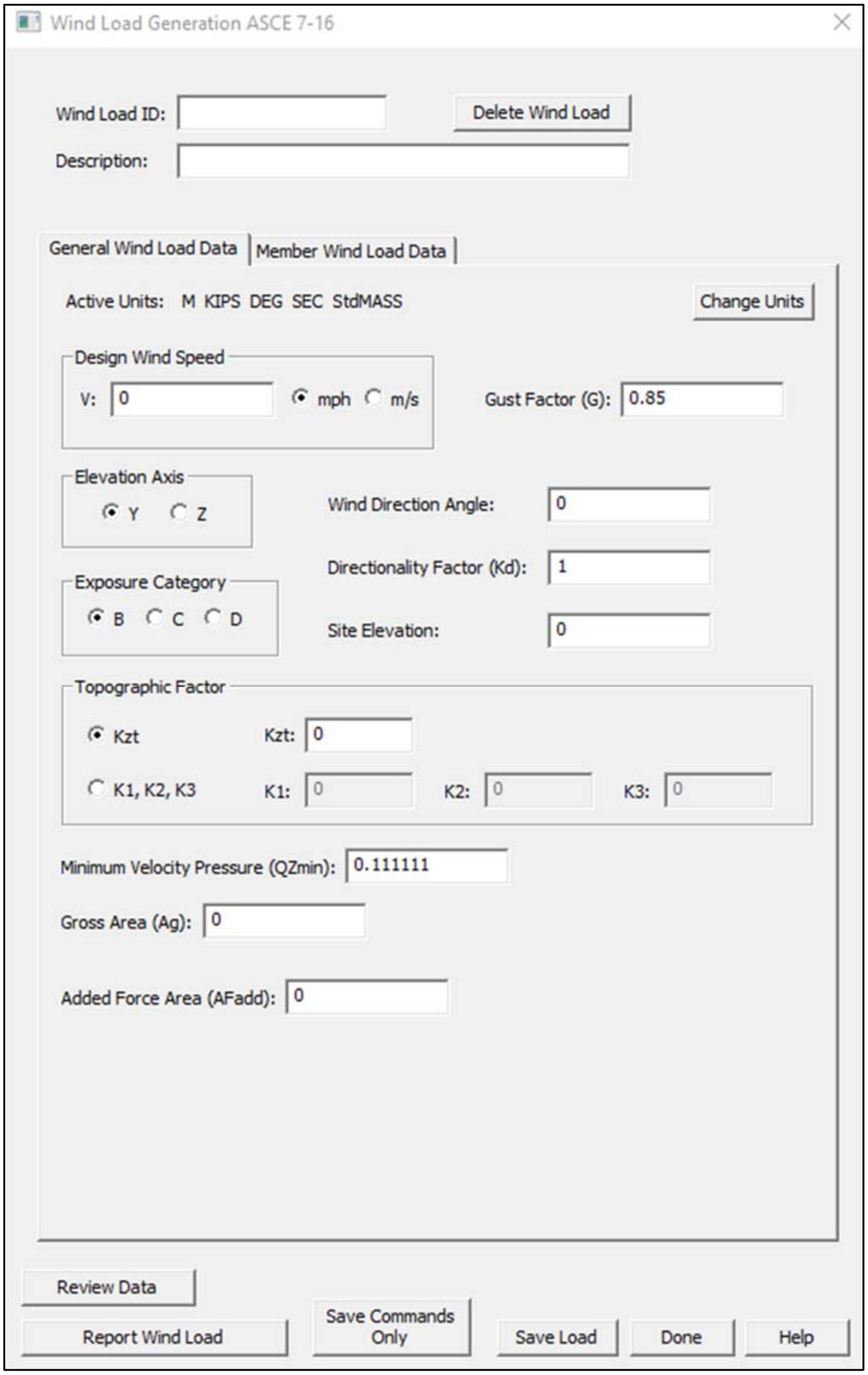
1. CAD Modeler now supports BricsCAD version 2022 and version 2023.

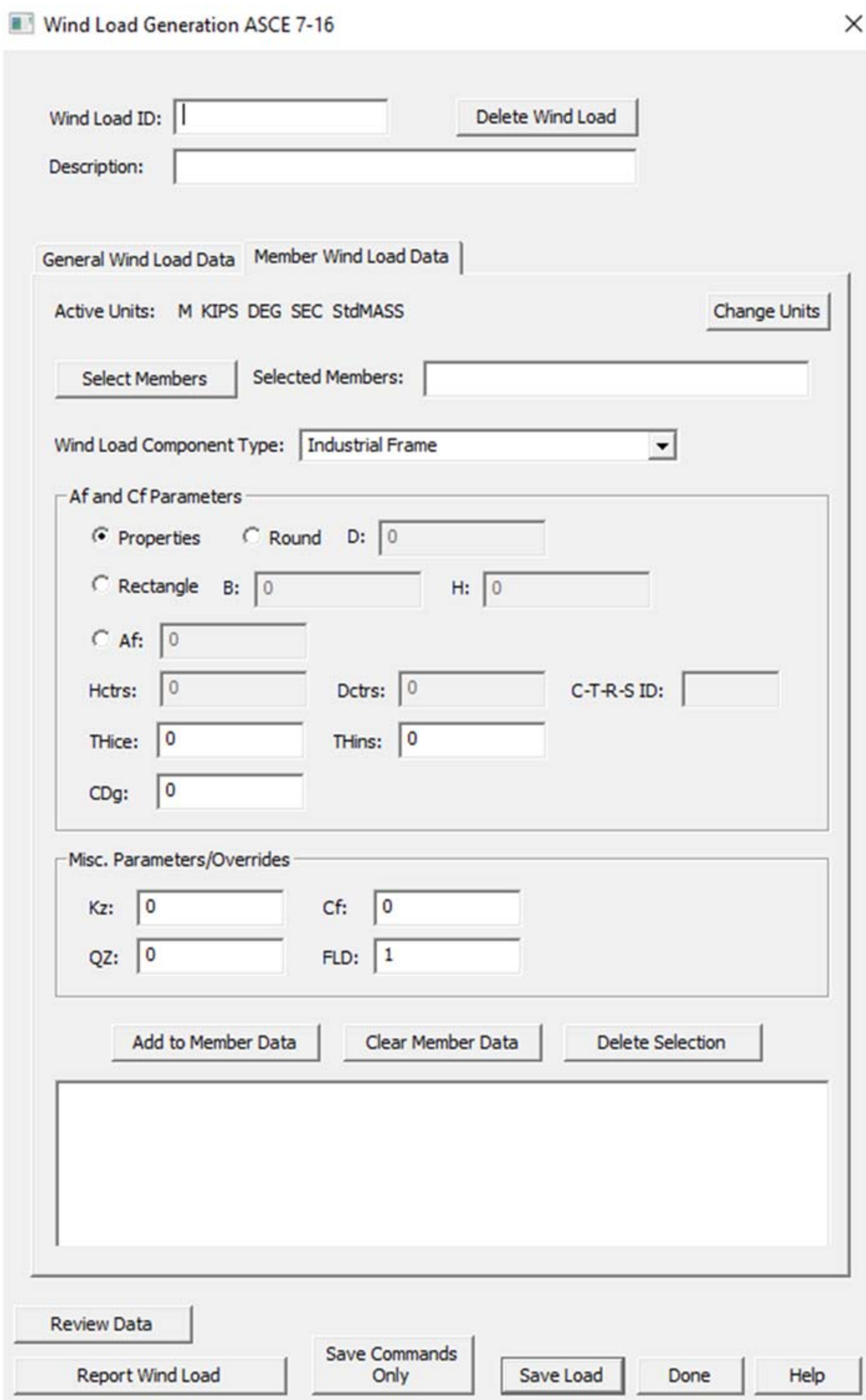


2. The ability to create both ASCE 7-16 Open Structure Wind Loads has been added.

A Wind Load can be entered from the ribbon command 

or from the menu *"GTS Modeling>Loads>Wind Load ASCE 716"* or by typing `GTSWindLoadsASCE716` at the command prompt.

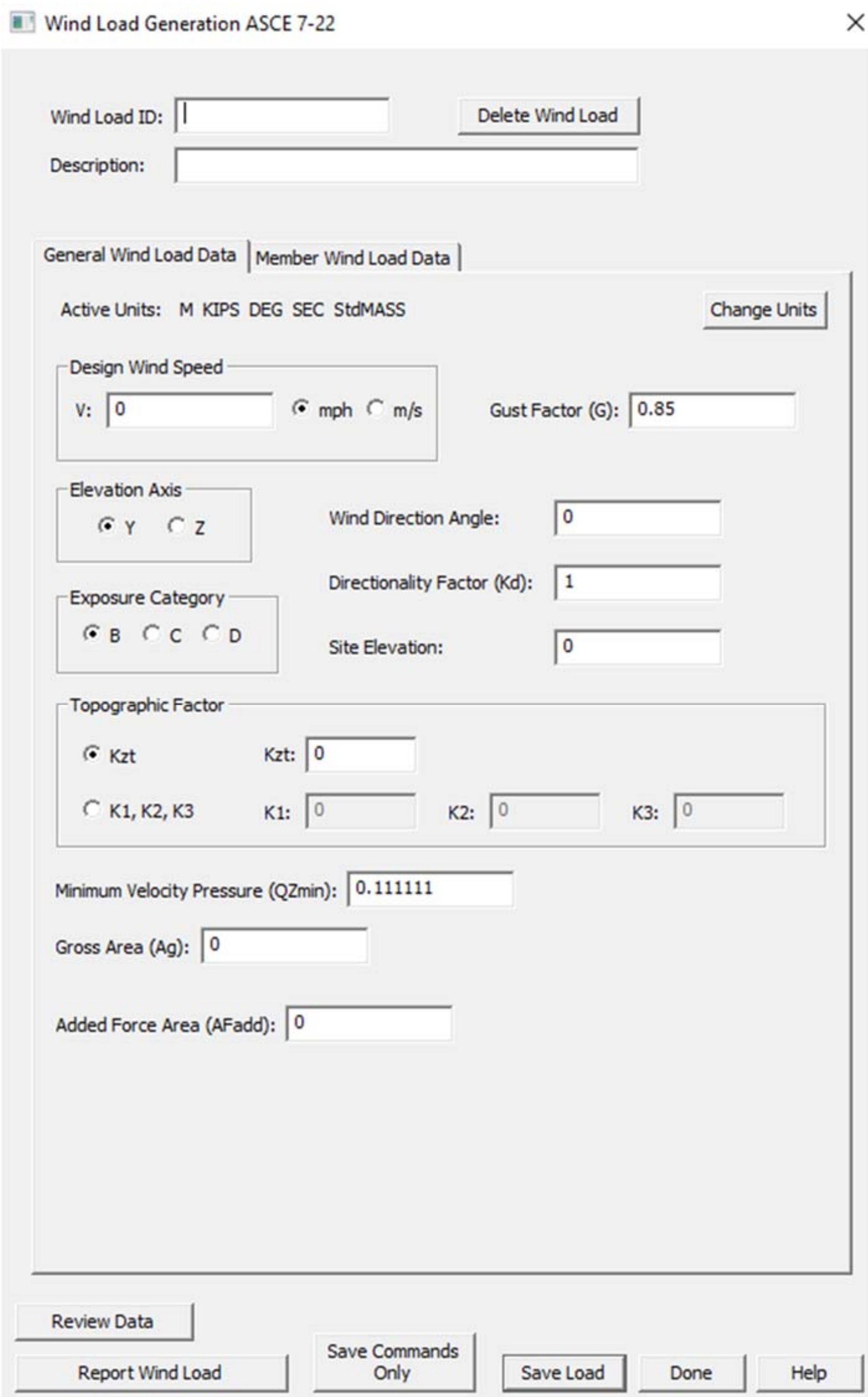

Wind Load Generation ASCE 7-16
Wind Load ID:
Delete Wind Load
Description:
General Wind Load Data
Member Wind Load Data
Active Units: M KIPS DEG SEC StdMASS
Change Units
Design Wind Speed
V: 0
mph
m/s
Gust Factor (G): 0.85
Elevation Axis
Y
Z
Wind Direction Angle: 0
Directionality Factor (Kd): 1
Exposure Category
B
C
D
Site Elevation: 0
Topographic Factor
Kzt
Kzt: 0
K1, K2, K3
K1: 0
K2: 0
K3: 0
Minimum Velocity Pressure (QZmin): 0.111111
Gross Area (Ag): 0
Added Force Area (AFadd): 0
Review Data
Report Wind Load
Save Commands Only
Save Load
Done
Help

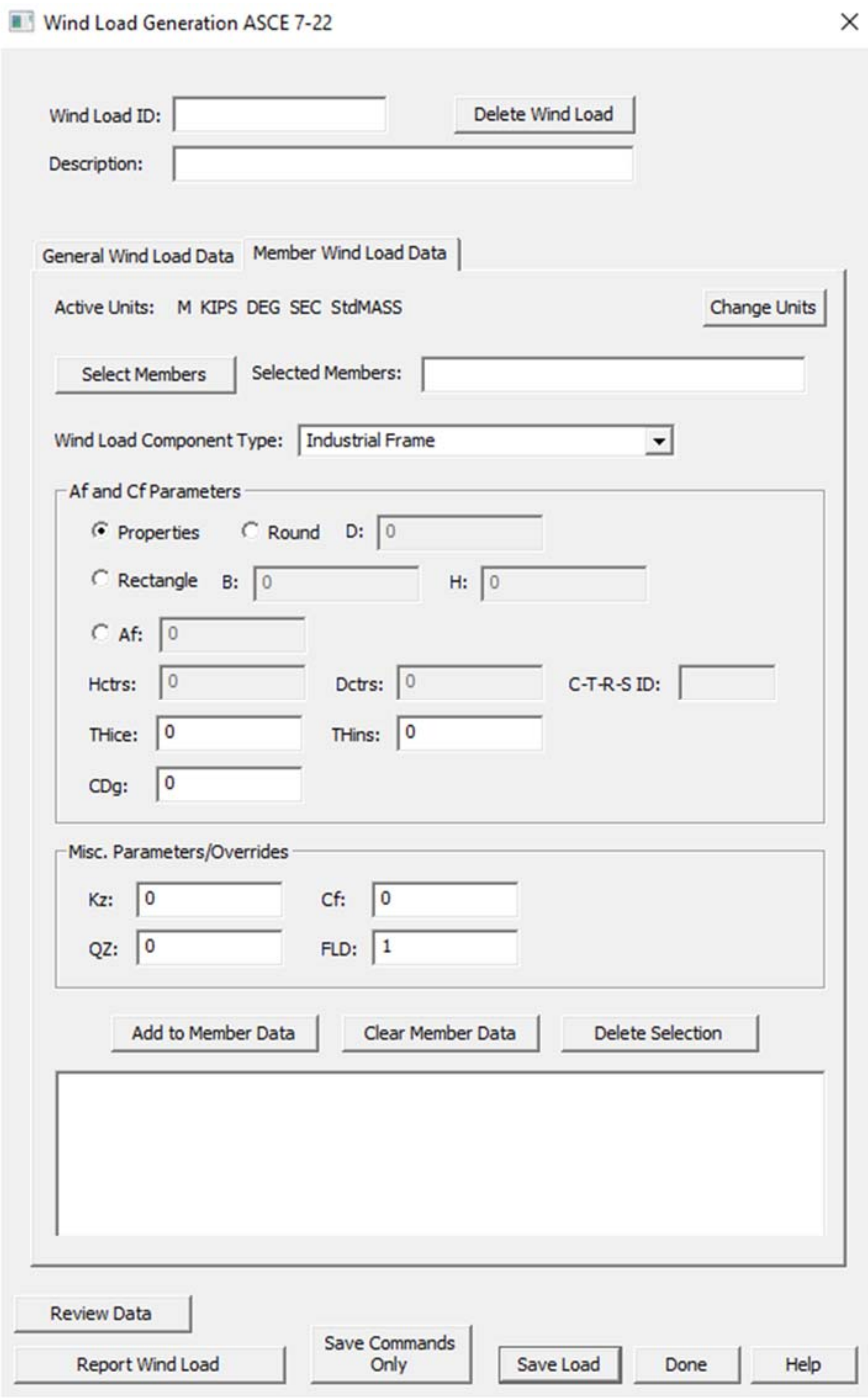

Wind Load Generation ASCE 7-16
Wind Load ID:
Delete Wind Load
Description:
General Wind Load Data
Member Wind Load Data
Active Units: M KIPS DEG SEC StdMASS
Change Units
Select Members
Selected Members:
Wind Load Component Type: Industrial Frame
Af and Cf Parameters
Properties
Round
D: 0
Rectangle
B: 0
H: 0
Af: 0
Hctrs: 0
Dctrs: 0
C-T-R-S ID:
THice: 0
THins: 0
CDg: 0
Misc. Parameters/Overrides
Kz: 0
Cf: 0
QZ: 0
FLD: 1
Add to Member Data
Clear Member Data
Delete Selection
Review Data
Report Wind Load
Save Commands Only
Save Load
Done
Help

3. The ability to create both ASCE 7-22 Open Structure Wind Loads has been added.

A Wind Load can be entered from the ribbon command 

or from the menu *"GTS Modeling>Loads>Wind Load ASCE 722"* or by typing `GTSWindLoadsASCE722` at the command prompt.



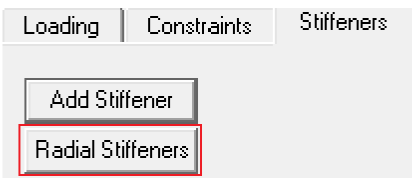

Wind Load Generation ASCE 7-22
Wind Load ID:
Delete Wind Load
Description:
General Wind Load Data
Member Wind Load Data
Active Units: M KIPS DEG SEC StdMASS
Change Units
Select Members
Selected Members:
Wind Load Component Type:
Industrial Frame
Af and Cf Parameters
Properties
Round
D:
Rectangle
B:
H:
Af:
Hctrs:
Dctrs:
C-T-R-S ID:
THice:
THins:
CDg:
Misc. Parameters/Overrides
Kz:
Cf:
QZ:
FLD:
Add to Member Data
Clear Member Data
Delete Selection
Review Data
Report Wind Load
Save Commands Only
Save Load
Done
Help

2.6 Base Plate Wizard

1. Triangular meshing using the GTMesh algorithm is now available in the Base Plate Wizard. Click the 'Triangular plate meshing only' box on the FE Mesh tab to use GTMesh. See the Base Plate Wizard User Guide or Release Guide for more details.

2. Circular plates are now available in the Base Plate Wizard. Click the 'Circular' radio button on the Plate tab. See the Base Plate Wizard User Guide or Release Guide for more details.

3. A Radial Stiffeners option has been added to the Stiffeners tab if Pipe attachments exist. Click this button to add one or more radial Stiffeners to a Pipe. This option will only appear if one or more Pipe attachments have been added to the plate. See the Base Plate Wizard User Guide or Release Guide for more details.



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. In the **Check → Duplicates & Floating Joints** dialog, using the **Interference Joints** option, the **To Notepad** button now generates a file without extra blank lines.

3.2 Loadings

1. The problem with mislabeling the dialog for ASCE wind loads on open structures has been corrected. Now the dialog title will correctly show the selected code, whether ASCE 7-10 or ASCE 7-16. In addition, if wind loads for ASCE 7-10 were created after creating loads for ASCE 7-16, the generated commands would specify a STANDARD of ASCE 7-16 instead of ASCE 7-10. This problem has also been corrected.

3.3 CAD Modeler

1. When sections have been selected from Section Profiles dialog, the selected sections have not been updated properly in the Place Member dialog. The error in Place Member dialog to update Section Profiles has been corrected.

2. When multiple sections have been selected from Section Profiles dialog, the selected sections have not been set up properly as selected in the Section Profile dialog. The error in Section Profiles to be set as selected has been corrected.

3. When the 'Create Load' button was clicked with a single associated dead load in Equation 1, the standard load combination was not created. This error in the Standard Load combination has been corrected.

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 41. 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.

4.5 Dynamic Analysis

1. If the MODAL DAMPING… command is given after the DYNAMIC ANALYSIS EIGENVALUE command, subsequent PERFORM TRANSIENT ANALYSIS, PERFORM RESPONSE SPECTRUM ANALYSIS, PERFORM STEADY STATE ANALYSIS, or PERFORM HARMONIC ANALYSIS commands will fail and SCAN ON will be set.
   (PRF 2022.01)

   Workaround:
   Move the MODAL DAMPING… command prior to the DYNAMIC ANALYSIS EIGENVALUE command.

2. When using the GT64M solver, the LIST HARMONIC MAXIMUM DISPLACEMENTS, VELOCITIES, or ACCELERATIONS command may produce unreliable phase angle results when the previous COMPUTE HARMONIC

DISPLACEMENTS, VELOCITIES or ACCELERATIONS command specifies a partial list of joints.

(PRF 2022.02)

Example:
COMPUTE HARMONIC DISPLACEMENTS VELOCITIES JOINTS –
'Gm' 'Gc' 'Gcyl1' 61 5 24688 24751
LIST HARMONIC MAX DISPL JOINT 'Gc'

Workaround:
Reference all joints in the COMPUTE command by omitting the joint list:

COMPUTE HARMONIC DISPLACEMENTS VELOCITIES

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:

5.4.5 Export to CAESAR II

5.4.6 Import CAESAR II Pipe Loads

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

Parameter Name	Default Value	Meaning
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**, **Netherlands**[2], **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
Singapore[2]	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 ≤ 2	b	0.34
	h/b > 2	c	0.49
Welded I cross-sections	h/b ≤ 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 ≤ 2	b	0.34
	2 < h/b ≤ 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 ≤ 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.

The form of the command is as follows:

DESIGN SLAB (REINFORCEMENT) (USING) -

$$\left\{\begin{matrix} \text{WOOD (AND) (ARMER)} \left\{\begin{matrix} \rightarrow\text{AVERAGE} \\ \text{MAXIMUM} \end{matrix}\right\} \\ \text{CALCULATE (RESULTANT) (ELEMENT) (FORCES)} \end{matrix}\right\} \text{(ALONG) -}$$

$$\text{(CUT} \left\{\begin{matrix} \text{'a'} \\ i_1 \end{matrix}\right\}\text{)} \left\{\begin{matrix} \text{JOINTS} \\ \text{NODES} \end{matrix}\right\} \text{list}_1 \text{ ELEMENT list}_2 \text{ (TABLE} \left\{\begin{matrix} \rightarrow\text{ASTM} \\ \text{UNESCO} \end{matrix}\right\}\text{) -}$$

$$*\left\{\begin{matrix} \text{TOP (FACE) (BARS } i_2\text{) (SPACING } v_1\text{)} \\ \text{BOTTOM (FACE) (BARS } i_3\text{) (SPACING } v_2\text{)} \\ \text{BOTH (FACE) (BARS } i_4\text{) (SPACING } v_3\text{)} \end{matrix}\right\} \text{-}$$

$$\left\{\begin{matrix} \rightarrow\text{INNER (LAYER)} \\ \text{OUTER (LAYER)} \end{matrix}\right\} \text{(COVER } v_4\text{) (LINEAR (TOLERANCE) } v_5\text{) -}$$

(TORSIONAL (MOMENT) (WARNING) v_6)

where,

‘a’ or i_1 refer to an optional alphanumeric or integer cut name

$list_1$	=	list containing ID’s of the start and end node of the cut
$list_2$	=	list containing the ID of an element in the plane of the cut
i_2	=	bar size to be used for bars on the top surface of the slab
i_3	=	bar size to be used for bars on the bottom surface of the slab
i_4	=	bar size to be used for both the top and bottom surfaces of the slab
v_1	=	reinforcing bar spacing to be used on the top surface of the slab
v_2	=	reinforcing bar spacing to be used on the bottom surface of the slab
v_3	=	reinforcing bar spacing to be used on both surfaces of the slab
v_4	=	optional user-specified cover distance for reinforcing bars
v_5	=	linear tolerance used in element selection rules for moment computation
v_6	=	optional ratio of torsion to bending moment allowed on the cross-section
TOP	=	element surface with +Z PLANAR coordinate
BOTTOM	=	element surface with -Z PLANAR coordinate

Explanation:

The DESIGN SLAB command allows the user to communicate all data necessary for the reinforcing steel design. This information is processed, and a design is calculated based on the input. The command is designed to provide varying levels of control for the user 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.

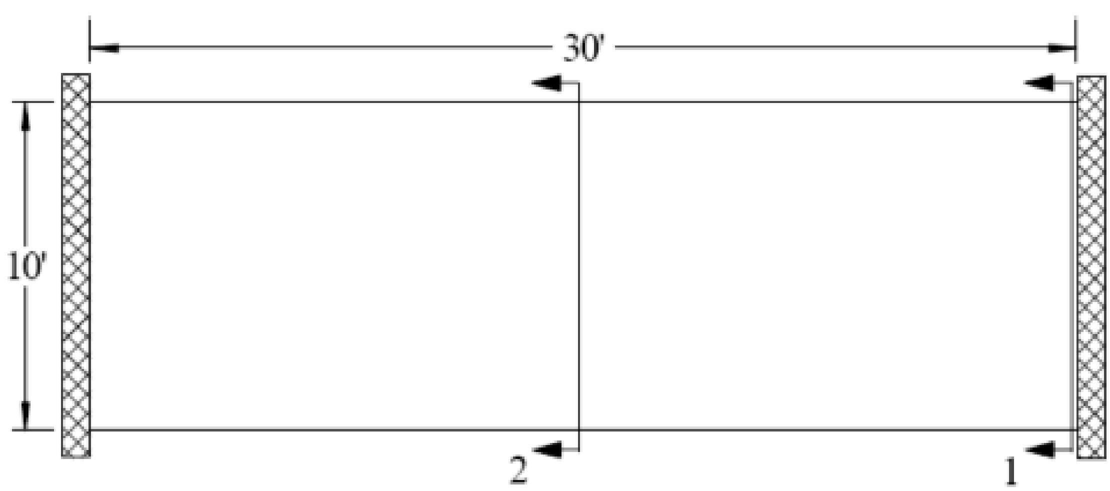


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

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.

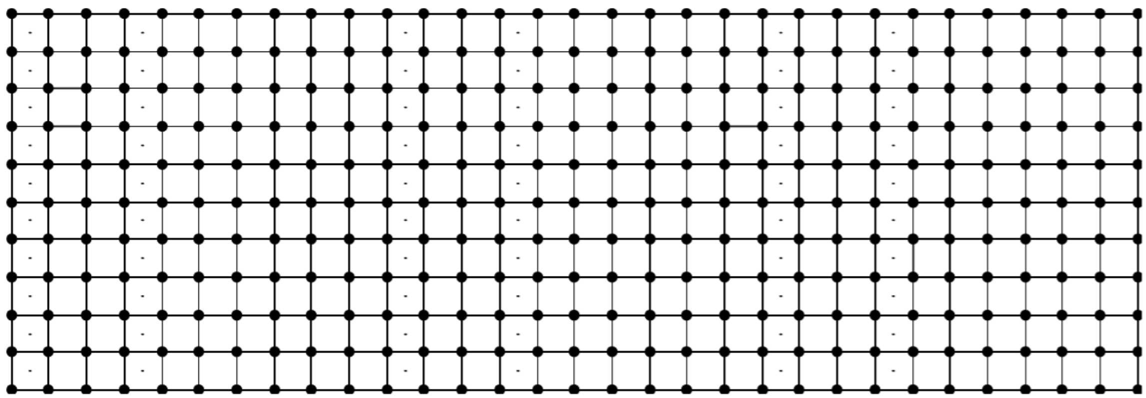


Figure 5.2.3-2 Example Finite Element Model

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) -

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

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

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,

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

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:

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

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

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

Difference from Average Method

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

Percent Maximum Difference Method

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

Percent Difference from Average Method

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

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

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

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

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

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

1. Enter GTMenu.

2. Select Results, Finite Element Contours, and then Energy & Stress Error Estimates.

3. Select the Estimate Method including Value, Surface, and Stress Component.

4. Select the Loading.

5. Select Display (solid colors or lines) to produce a contour of the error estimate.

6. Select Legend to place a legend on the screen indicating the type of error estimate, loading, and surface.

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

General form:

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

Explanation:

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

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

**** WARNING_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:

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

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 θ_1, θ_2, θ_3 as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

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

The ROTATE LOADING command is used to take the global applied loading components from an existing loading condition, rotate them through a set of rotation angles, and copy the new rotated global components to a new or modified different destination loading condition. The existing independent loading condition, the ROTATE load, from which the rotated global load components are computed is specified by the loading name i_R/'a_R'. The angles of rotation are specified by the values r_1, r_2, r_3. These rotation angles are defined according to the same conventions as those that define the local support release directions in the JOINT RELEASE command described in Section 2.1.7.2, Volume 1 of the GT STRUDL User Reference Manual, and illustrated in Figure 2.1.7-1.

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

Taking the specified loading i_R/'a_R', the ROTATE LOADING command performs the following operations and copies the results into the destination loading condition:

1. Rotate all joint loads, including applied joint support displacements.

2. Rotate all member force and moment loads defined with respect to the global coordinate system. Member force and moment loads defined with respect to the member local coordinate system are simply copied without rotation.

3. Rotate all element force loads defined with respect to the global coordinate system. Element force loads defined with respect to any applicable local or planar coordinate systems are copied without rotation.

4. All other types of loads such as member temperature loads, member distortions, joint temperatures, etc. are copied without changes.

Examples:

1.
```
UNITS DEGREES
LOADING 2 'ROTATED LOADING'
MEMBER DISTORTIONS
     1 TO 10 UNIFORM FR LA 0.0 LB 1.0 DISPL X 0.001
ROTATE LOADING 1 ANGLES T1 45.0
```

The applied loads from previously defined loading 1 will be processed according to Steps 1 to 4 above and copied into the new destination loading 2, which includes the specified member distortion loads applied to members 1 to 10.

2.
```
UNITS DEGREES
CHANGES
LOADING 3
ADDITIONS
ROTATE LOAD 4 ANGLES T2 -30.0
```

Previously defined loading 3 is specified in CHANGES mode, followed by a return to

ADDITIONS mode. The ROTATE LOAD command is then given to add the components of load 4, including appropriate rotations, to loading 3.

Error Messages:

Incorrect data given in the ROTATE LOADING command will cause the following error conditions to be identified and error messages printed:

1. The following error message is printed if the ROTATE loading name is identical to the name of the destination load. An example of the commands that produce this error are also included:

```
{   114} > LOADING 201
{   115} > ROTATE LOAD 201 T1 45.0

**** ERROR_STROLO -   The ROTATE loading is illegally the same as the destination
                      loading.
                      Command ignored.
```

   Loading 201 is illegally named as both the destination load and the loading whose components are rotated.

2. In the following error example, loading 51 is undefined.

```
{   111} > LOADING 201
{   112} > ROTATE LOAD 51 T1 45.0

**** ERROR_STROLO -   Loading to be rotated undefined.
                      Command ignored.
```

3. The following error message is produced because loading 4, specified as the ROTATE load, is a load combination, or dependent loading condition. The ROTATE load must be an independent loading condition.

```
{   141} > LOADING 108
{   142} > ROTATE LOADING 4 T3 45.0

**** ERROR_STROLO -   Rotated Loading 4 is an illegal dependent load.
                      Command ignored.
```

4. This error condition and message is caused by the fact that the destination load 108 is defined as a loading combination.

```
{   144} > LOAD COMB 108 ‘ALL’ COMBINE 1 1.5 2 1.0 3 1.0
{   145} > ROTATE LOADING  1 T3 45.0

**** ERROR_STROLO -   Destination independent loading not defined.
                      Rotated load components not computed.
```

5.4.2 REFERENCE COORDINATE SYSTEM Command

General form:

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

$$\left\{ \begin{array}{l} (\underline{\text{ORI}}\text{GIN } [\underline{\text{X}}]\ v_x\ [\underline{\text{Y}}]\ v_y\ [\underline{\text{Z}}]\ v_z)\ (\underline{\text{ROT}}\text{ATION } [\underline{\text{R1}}]\ v_1\ [\underline{\text{R2}}]\ v_2\ [\underline{\text{R3}}]\ v_3) \\ \\ \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{\text{X}}\ v_4\ \underline{\text{Y}}\ v_5\ \underline{\text{Z}}\ v_6 \end{Bmatrix} \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_3 \\ 'a_3' \end{Bmatrix} \\ \underline{\text{X}}\ v_7\ \underline{\text{Y}}\ v_8\ \underline{\text{Z}}\ v_9 \end{Bmatrix} \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_4 \\ 'a_4' \end{Bmatrix} \\ \underline{\text{X}}\ v_{10}\ \underline{\text{Y}}\ v_{11}\ \underline{\text{Z}}\ v_{12} \end{Bmatrix} \end{array} \right\}$$

Explanation:

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

At the present time, this command is used to specify additional coordinate systems which may be used in GTMenu (see Volume 2 of the GT STRUDL Release Guide) to facilitate the creation of the structural model. Reference Coordinate systems created using the above command will be available as Local systems in GTMenu. In a future release, the user will be able to output results such as joint displacements and reactions in a Reference Coordinate System.

There are two optional means of specifying a Reference Coordinate System:

(1) Define the origin and rotation of coordinate axes of the reference system with respect to the global coordinate system, and

(2) define three joints or the coordinates of three points in space.

In either case, i_1 or 'a_1' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x, v_y, and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x, v_y, and v_z, are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive directions of these axes; v_1, v_2, and v_3 are the rotation angles 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 'a_2' or v_4, v_5, and v_6) defines the origin of the reference system; the X-axis of the reference system is determined by the first and second joints (i_3 or 'a_3' or v_7, v_8, and v_9). The positive X-axis is directed from the first to the second joint. The third joint (i_4 or 'a_4' or v_{10}, v_{11}, and v_{12}) is used to define the XY-plane of the reference system. The positive Y-axis is directed toward the third joint. The Z-axis then is determined by the right-hand rule.

Only one reference system can be specified in one command, but the command may be used any number of times.

Modifications of Reference Systems:

In the changes mode, the translations of the origin and/or the rotations of the axes of the reference system from those of the overall global system can be changed. Only that information supplied in the command is altered. The other data that might be supplied in the command remains unchanged. The CHANGES mode, however, does not work for the second option discussed above (i.e., define a reference coordinate system by three joints or the coordinate of three points in space). The reason is that data for these joints are not stored permanently in GT STRUDL. When this option is used, a reference system is created and its definitions of the system origin, rotation angles, as well as the transformation matrix between the global coordinate system and the reference system are generated and stored as would be for the first option. Therefore, if any of the coordinates for the joints used to specify a reference system is changed after the REFERENCE COORDINATE SYSTEM command has been given, the definition of the reference system remains unchanged. For this reason, care must be taken in using the three joints option in conjunction with the changes of joint coordinates. The reference system should be deleted first if any of the coordinates of the joints used to define the reference system are to be changed. Under the DELETIONS mode, the complete definition of the reference coordinate system is destroyed.

Examples:

a)
```
UNITS DEGREES
REFERENCE COORDINATE SYSTEM 'FLOOR2'   -
   ORIGIN 0.0 15.0 0.0 R1 30.
```

This command creates a Reference Coordinate System called FLOOR2 at Y=15 with the axes rotated 30 degrees about global Z.

b)
```
REF COO 1   -
     X    120   Y   120   Z   -120  -
     X    120   Y   240   Z      0  -
     X   -120   Y   120   Z      0
```

This command creates Reference Coordinate System 1 with its origin at 120, 120, -120 and its X-axis from this origin to 120, 240, 0 and its Y axis is the plane defined by the two previous coordinates and the third coordinate, -120, 120, 0, with the positive Y-axis directed toward the third coordinate.

c)
```
REFERENCE COORDINATE SYSTEM 2   -
    JOINT 10 JOINT 20 JOINT 25
```

This command creates Reference Coordinate System 2 with its origin located at Joint 10 and its X-axis directed from Joint 10 toward Joint 20. The XY plane is defined by Joints 10, 20, and 25 with the positive Y-axis directed toward Joint 25.

d)
```
CHANGES
REFERENCE COORDINATE SYSTEM ‘FLOOR2’   -
    ORIGIN   10   20   30
ADDITIONS
```

The above commands change the origin of the Reference System FLOOR2 defined in a) above. The rotation RI = 30 remains unchanged.

e)
```
DELETIONS
REFERENCE SYSTEM 2
ADDITIONS
```

The above command deletes Reference System 2.

5.4.2-1 Printing Reference Coordinate System Command

General form:

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

Explanation:

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

5.4.3 GTMenu 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

{'a_1'} surface-specs$_1$
•
•
•
{'a_n'} surface-specs$_n$

Elements:

surface-specs = $\begin{Bmatrix} \text{(PATCH SURFACE SPACING) iu iv patch-specs} \\ \text{(SURFACE OF) REVOLUTION (SPACING) iu iv sor-specs} \end{Bmatrix}$

patch-specs = U (CURVES) 'b_1' . . . 'b_n' V (CURVES) 'c_1' . . . 'c_m'

sor-specs = (REVOLUTION ANGLE) v axis-specs U (CURVE) 'b_1'

axis-specs = (AXIS) $\begin{Bmatrix} \text{POINTS 'd}_1\text{' 'd}_2\text{'} \\ \text{COORDINATES START } x_1\ y_1\ z_1 \text{ END } x_2\ y_2\ z_2 \end{Bmatrix}$

where,

'a_1', 'a_2', ..., 'a_n'	=	1 to 8 character alphanumeric Surface IDs beginning with S (i.e. S1, S2).
iu, iv	=	integer values representing the number of drawing

segments to use in directions U and V respectively.

‘b_1’, ‘b_2’, ..., ‘b_n’ = 1 to 8 character alphanumeric Line/Curve IDs for U direction. n must be greater than or equal to 1 and less than or equal to 10. Line/Curve IDs begin with C (i.e. C1,C2).

‘c_1’, ‘c_2’, ..., ‘c_m’ = 1 to 8 character alphanumeric Line/Curve IDs for V direction. m must be greater than or equal to 1 and less than or equal to 10. Line/Curve IDs begin with C (i.e. C1,C2).

v = real number representing the angle of revolution.

‘d_1’, ‘d_2’ = 1 to 8 character alphanumeric Point IDs for start and end points of the axis of revolution respectively. Point IDs begin with P (i.e. P1,P2).

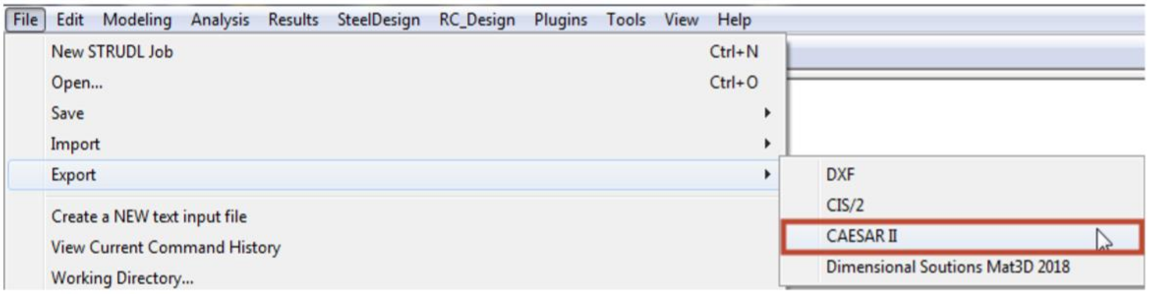
x_i, y_i, z_i = real values representing coordinates for global directions X, Y, Z respectively of the start and end points of the axis of revolution.

Examples:

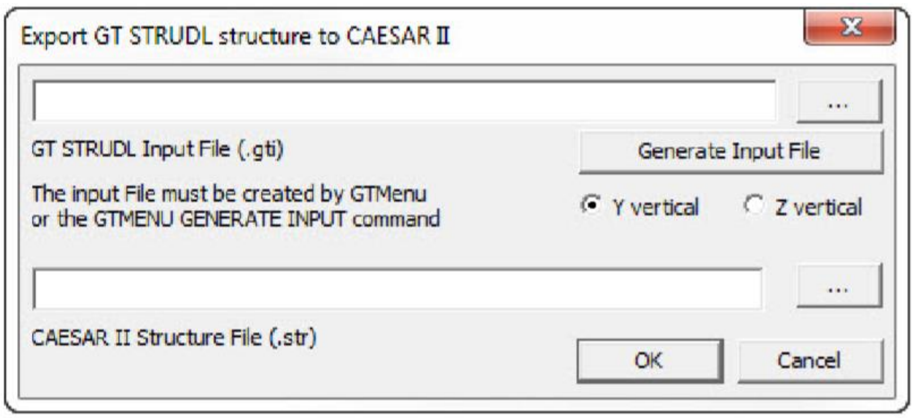
```
GTMenu SURFACE DEFINITION
    'S1' PATCH SURFACE SPACING 10 20 -
        U CURVES 'C1' -
        V CURVES 'C2'

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

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

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:

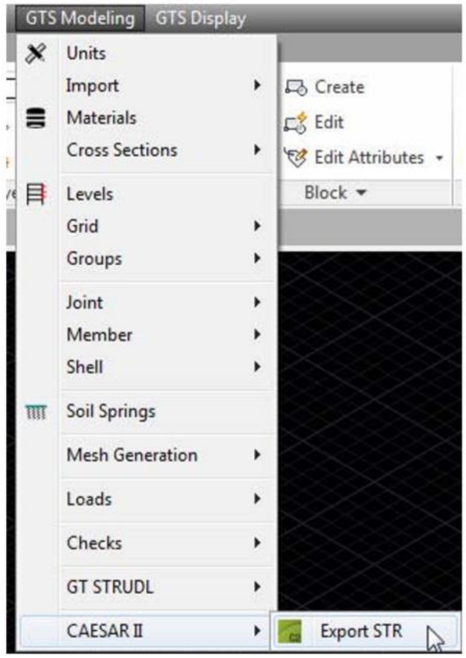


After selecting CAESAR II, the following dialog will pop-up:



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 GTSExportSTR 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:

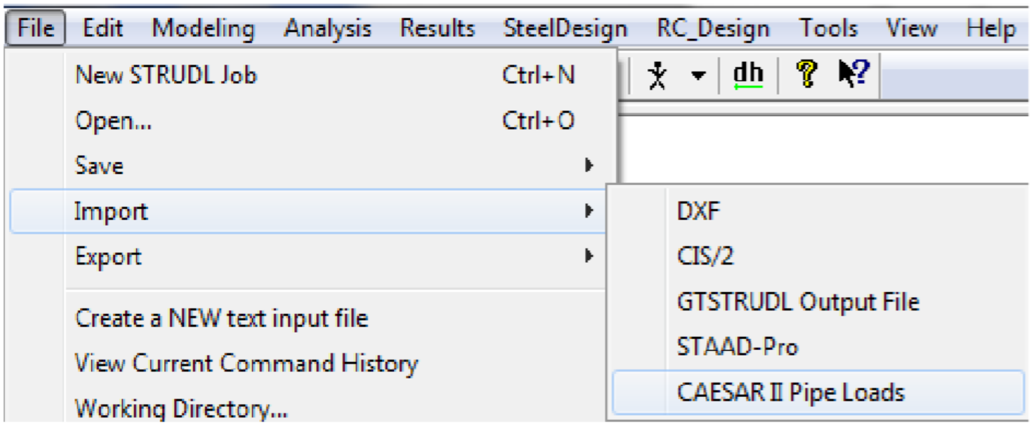
```
WARNING: Section L1x1x1/4 is not available in CII, please use another one
or edit
F:\\HexagonPPM\CaesarII\PlantStructure\\PStructure_0708_01.str       file
manually
```

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).

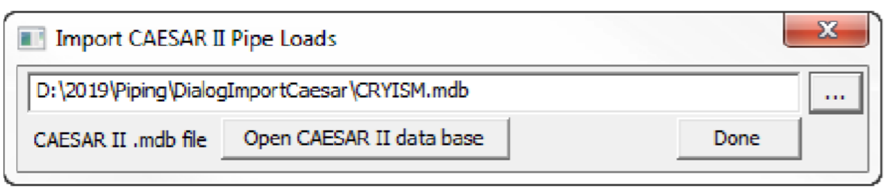
5.4.5 Import CAESAR II Pipe Loads

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:



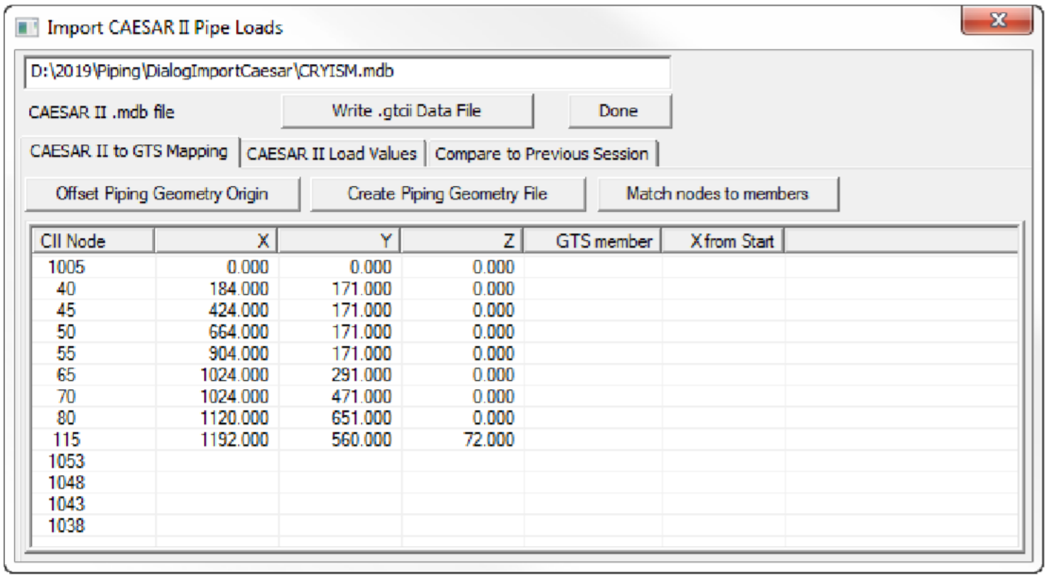
Clicking the menu item will launch the opening dialog, and if there is structural data in the GT STRUDL data base, write files so the dialog can get the structural model data. Choose a .mdb file by typing or browsing with the button and then click the 'Open CAESAR II data base' button.



This will open to the full "Import CAESAR II Pipe Loads" dialog, which is organized into three tabs: "CAESAR II to GTS Mapping", "CAESAR II Load Values", and "Compare to Previous Session". Each tab is discussed in the following User Guide. Each tab will have a spreadsheet portion, which will be referred to as a "grid". You can select and copy from these grids, and then paste into another program, such as Microsoft Excel. You can paste into user entry portions of the grid. Data is expected to be separated by Tabs for pasting. Pasting allows you to save data from one session of the dialog to another, such node coordinates not included in the CAESAR II .mdb file or load values you have edited to be conservative.

Use the F1 key to open a Help file describing the use of this feature. The online Help will contain details about using the functions and dialogs of the Import CAESAR II Pipe Loads feature.

The "CAESAR II to GTS Mapping" Tab



CII Node	X	Y	Z	GTS member	X from Start
1005	0.000	0.000	0.000		
40	184.000	171.000	0.000		
45	424.000	171.000	0.000		
50	664.000	171.000	0.000		
55	904.000	171.000	0.000		
65	1024.000	291.000	0.000		
70	1024.000	471.000	0.000		
80	1120.000	651.000	0.000		
115	1192.000	560.000	72.000		
1053					
1048					
1043					
1038					

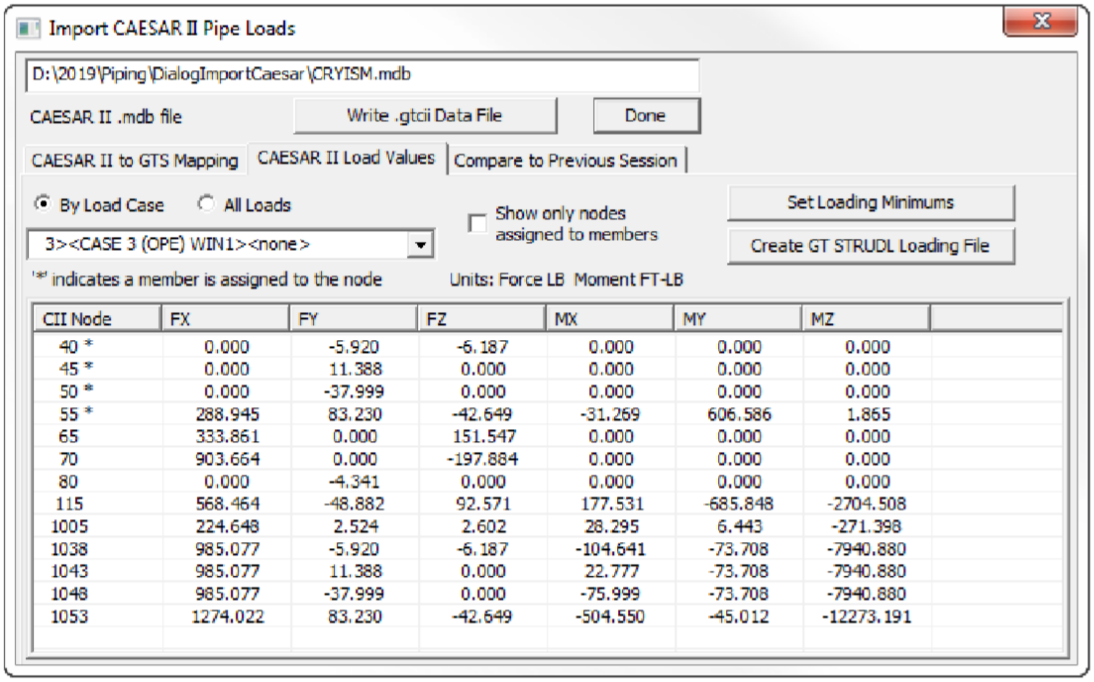
This tab is used to assign the CAESAR II nodes to a location along a GT STRUDL member. Only reactions for nodes with a 'GTS member' assignment can be exported to a loading file. You can type member names and fractional distance along the local X axis into the grid by double-clicking the cell. Press Enter or click away from the cell to stop editing. Alternately, you can paste into a block of cells.

The "Offset Piping Geometry Origin" function is used to align the CAESAR II coordinate system with the GT STRUDL global coordinate system.

The "Create Piping Geometry File" function is used to create a GT STRUDL input file with joints and members representing the piping system. This can be helpful in deciding which member should be assigned to a node.

The "Match nodes to members" function will make an initial assignment of members to nodes, but user verification is required to ensure accurate placement of CAESAR II reactions as GT STRUDL loads.

The "CAESAR II Load Values" Tab



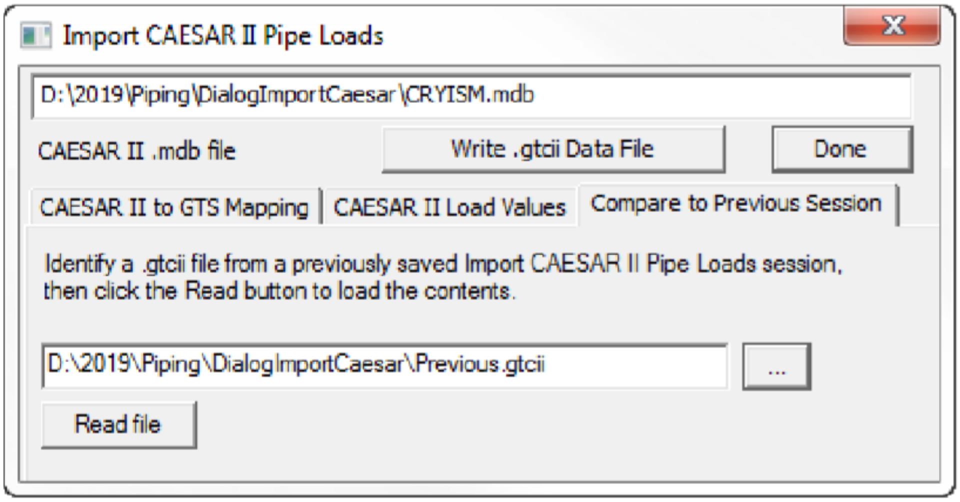
Reaction values can be displayed with only one load case in the grid, or all loads in the grid. A dropdown menu is provided to shift between load cases when in single load case mode. Only nodes with assigned GT STRUDL members, those with an entry in the "GTS member" column, can be exported into a loading file. The relevant nodes are indicated with " *" after the node number in the grid.

By default, the load value grid is created from the reaction values found in the CAESAR II .mdb file. You can override the values from the .mdb file, either with the "Set Loading Minimums" function or by double-clicking in a field and typing in a user specified value. Run "Set Loading Minimums" first if you are going to enter values as the minimums function will possibly change your entered data.

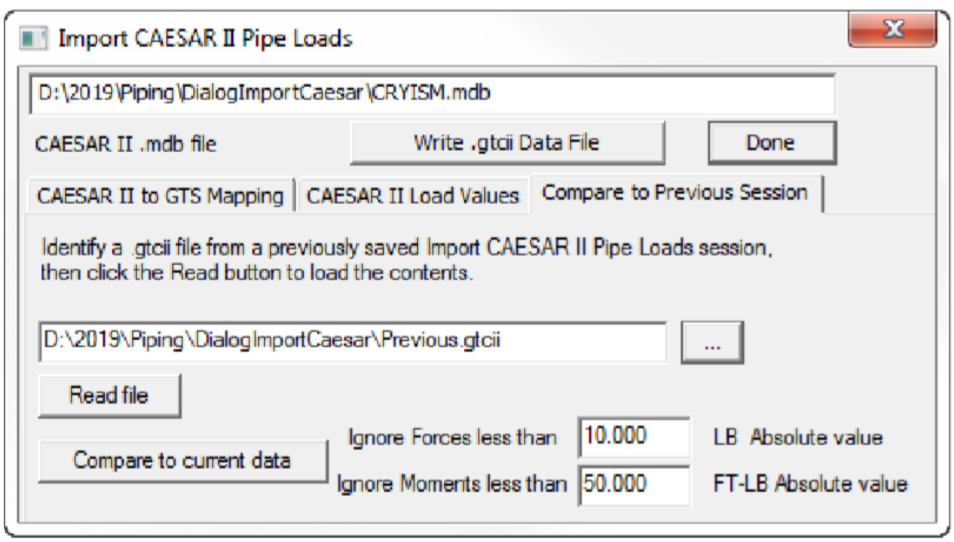
If you want to be able to compare this load data with a subsequent CAESAR II pipe stress analysis, use the "Write .gtcii Data File" button to save the current data to used with the next tab.

The "Compare to Previous Session" Tab

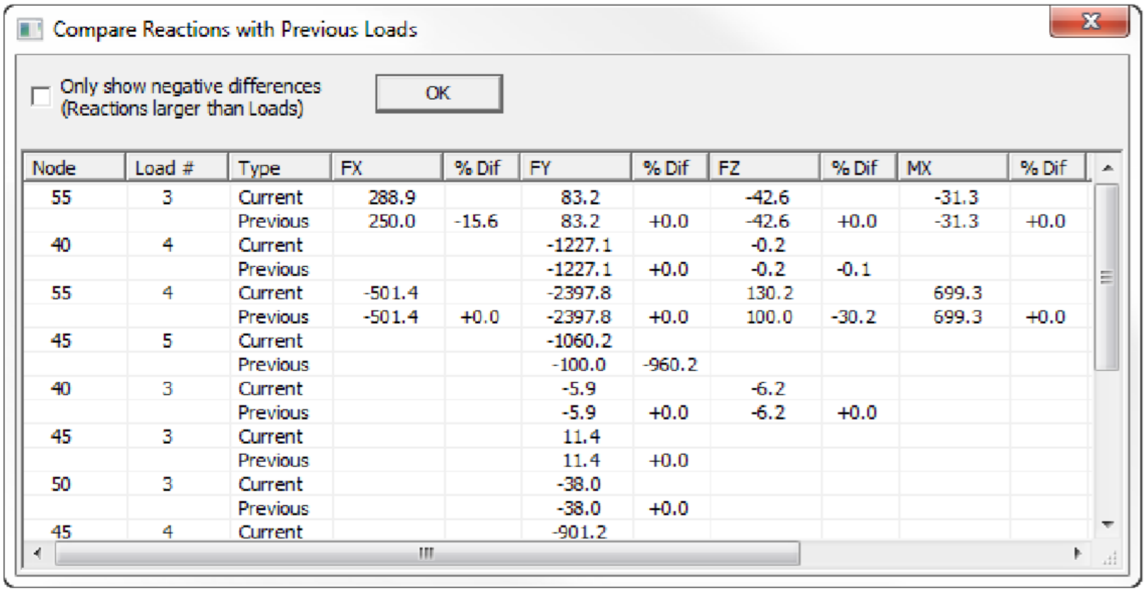
If you have saved a previous data file (.gtcii extension), you can use that to compare with a new CAESAR II analysis to verify that the new reaction values do not exceed the loading values sent to GT STRUDL. Enter or browse for the .gtcii file and then click the 'Read file' button.



If the .gtcii file is compatible with the current CAESAR II ,mdb file, then the 'Compare to current data' will be enabled.



Previous loads values that are smaller than the current CAESAR II reactions will be reported as "negative". Nodes with negative compare results will be at the top of the grid. There is an option to see only the nodes with negative compare results. Check the "% Dif" columns to see the comparison results.



Node	Load #	Type	FX	% Dif	FY	% Dif	FZ	% Dif	MX	% Dif
55	3	Current	288.9		83.2		-42.6		-31.3	
		Previous	250.0	-15.6	83.2	+0.0	-42.6	+0.0	-31.3	+0.0
40	4	Current			-1227.1		-0.2			
		Previous			-1227.1	+0.0	-0.2	-0.1		
55	4	Current	-501.4		-2397.8		130.2		699.3	
		Previous	-501.4	+0.0	-2397.8	+0.0	100.0	-30.2	699.3	+0.0
45	5	Current			-1060.2					
		Previous			-100.0	-960.2				
40	3	Current			-5.9		-6.2			
		Previous			-5.9	+0.0	-6.2	+0.0		
45	3	Current			11.4					
		Previous			11.4	+0.0				
50	3	Current			-38.0					
		Previous			-38.0	+0.0				
45	4	Current			-901.2					