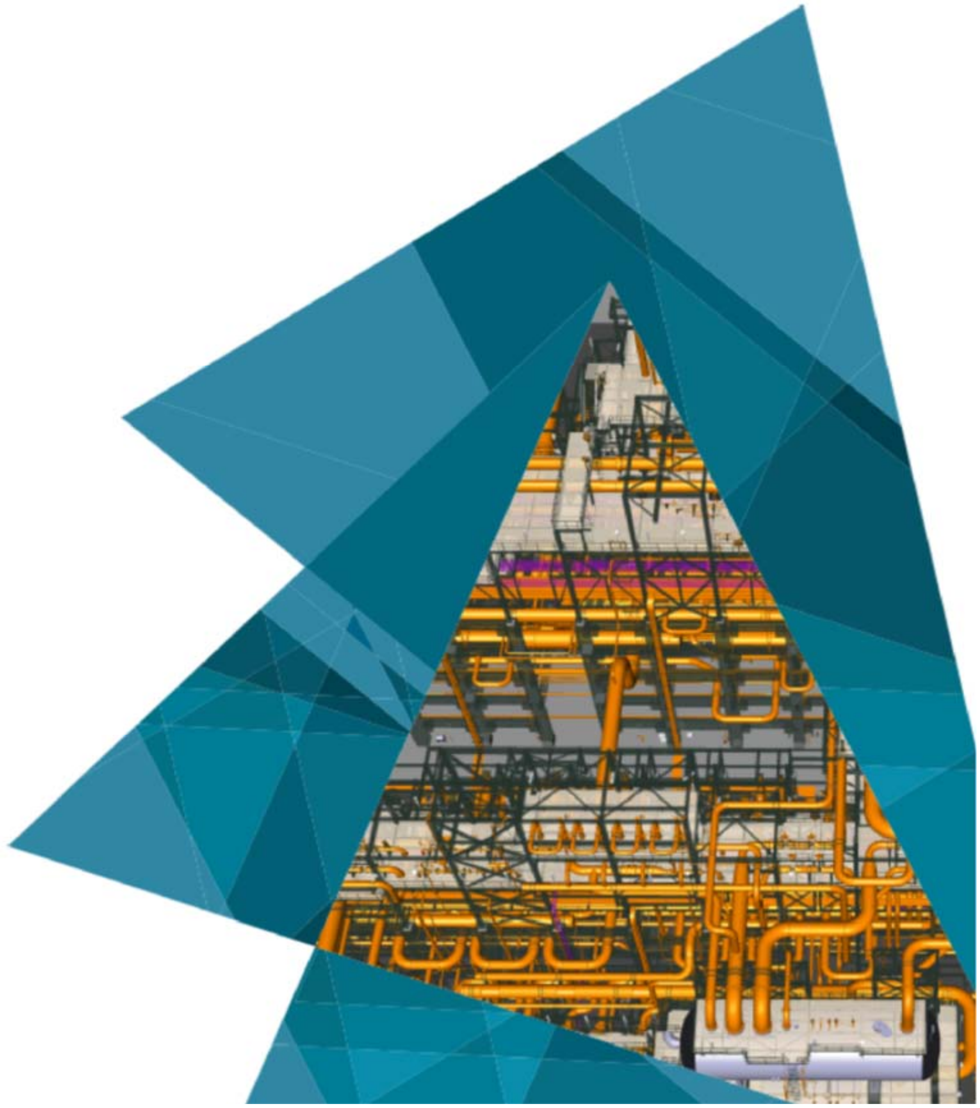
GT STRUDL® Version 2020 Release Guide



Release Date: May 2020

HEXAGON PPM

Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 2020 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

U.S. Government Restricted Rights Legend

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the Software License Agreement ("SLA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a SLA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee’s internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee’s internal, non-commercial use. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the SLA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

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

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

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

Limitation of Damages

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

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

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

Export Controls

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

a. to Cuba, Iran, North Korea, the Crimean region of Ukraine, or Syria, or any national of these countries or territories.

b. to any person or entity listed on any United States government denial list, including, but not limited to, the United States Department of Commerce Denied Persons, Entities, and Unverified Lists, the United States Department of Treasury Specially Designated Nationals List, and the United States Department of State Debarred List. Visit www.export.gov for more information or follow this link for the screening tool: https://legacy.export.gov/csl-search.

c. to any entity if Customer knows, or has reason to know, the end use of the software product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.

d. to any entity when Customer knows, or has reason to know, that an illegal reshipment will take place.

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

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

Trademarks

Intergraph®, the Intergraph logo®, Intergraph Smart®, SmartPlant®, SmartMarine®, SmartSketch®, SmartPlant Cloud®, PDS®, FrameWorks®, I-Route, I-Export, Isogen®, SPOOLGEN, SupportManager®, SupportModeler®, SAPPHIRE®, TANK, PV Elite®, CADWorx®, CADWorx DraftPro®, GTSTRUDL®, and CAESAR II® are trademarks or registered trademarks of Intergraph Corporation or its affiliates, parents, subsidiaries. Hexagon and the Hexagon logo are registered trademarks of Hexagon AB or its subsidiaries. Microsoft and Windows are registered trademarks of Microsoft Corporation. ACIS is a registered trademark of SPATIAL TECHNOLOGY, INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, 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 **Page**

Chapter 1

Introduction

Version 2020 covers GT STRUDL operating on PC's under the Windows 10 x64 based operating system. For users who are accustomed to our older version numbering system, the version is internally known as Version 39.0.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2019. In particular, Chapter 2 briefly describes an extensive list of new features including the following new features of which a few are highlighted below:

- GT STRUDL is now a 64bit program. Which allows for the allocation of memory beyond 4Gb; therefore, modeling of larger models.

- GB-2017 Steel Design Code based on People's Republic of China National Standard, GB 50017–2017, Code for Design of Steel Structures, Published in December 12, 2017.

- Numerous improvements to CAD Modeler including supporting BricsCAD version 20, and the use of Shell dialogs such as Load Combinations, Wind Loads, Seismic Loads, and Model Wizard.

- Numerous improvements to GTMenu including the ability to now create and edit Joints, Members, and Elements through spreadsheets.

Chapter 3 provides you with details regarding error corrections that have been made since the Version 2019 release.

Chapter 4 describes known problems with Version 2020.

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 2020

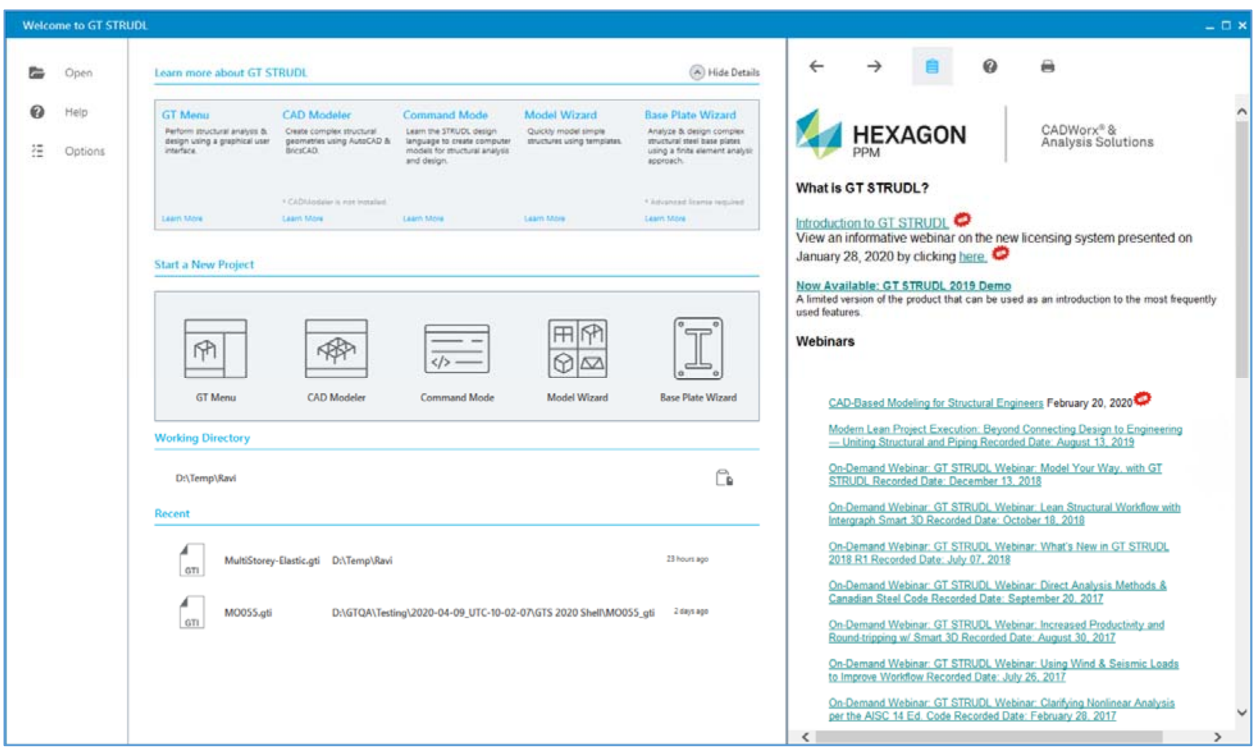
This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GT STRUDL in Version 2020. This release guide is also available online upon execution of GT STRUDL under menu "Help → Reference Documentation → GT STRUDL Release Guide".

2.1 General

1. GT STRUDL is now a 64bit Application. This allows handling of larger models as well as performance improvements.

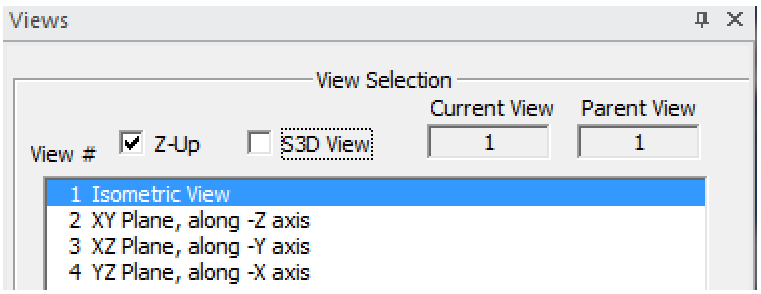
2. Intergraph Smart License has been added to GT STRUDL 2020. It becomes the sole licensing method for commercial usage of GT STRUDL. Intergraph Smart License is a separate product from GT STRUDL and must be installed prior to running GT STRUDL 2020.

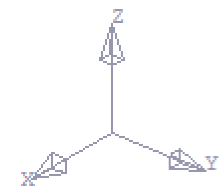
3. The welcome screen has been redesigned to show all options in a single page, including news and GT STRUDL Help.

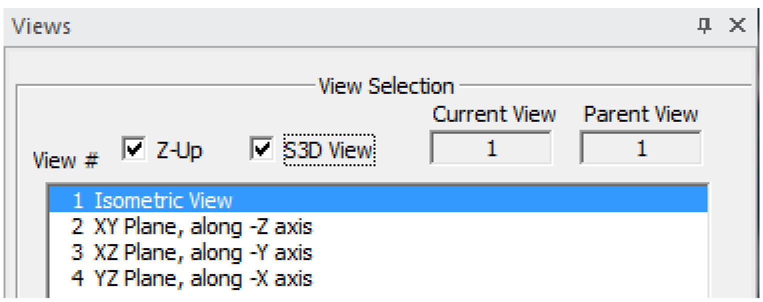


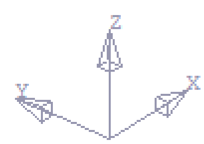
2.2 GTMenu

1. A new option "S3D View" has been added to the Views dialog for "Z-up" (the global Z axis as the gravity or vertical axis) to set View #1 to an orientation rotated 180 degrees around the Z axis. This view corresponds to the Smart3D isometric view and allows for easier visual checking of imported models.

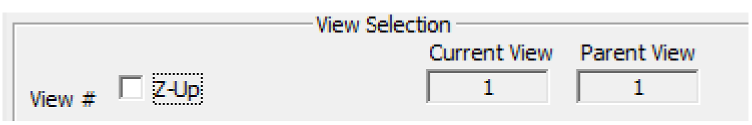






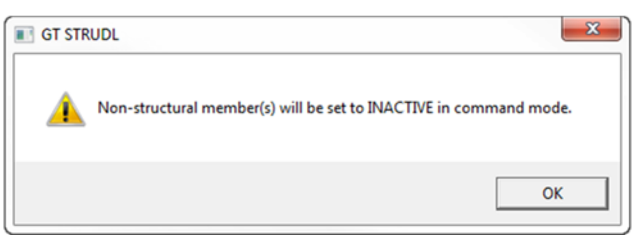


The "S3D View" option does not appear when the "Z-Up" checkbox is unchecked.

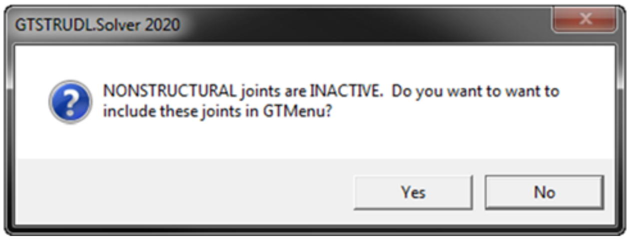


2. To aid in the viewing of piping systems imported from CAESAR II, the piping geometry members are declared as NONSTRUCTURAL. Nonstructural members in GTMenu are drawn in the color assigned to Member Display. To help distinguish them, nonstructural members are drawn with dashed lines in wire-frame mode and as 1-color members in solid mode. In addition, joints associated with the piping system will be nonstructural and displayed in the Member Display color instead of the Joint Display color.

When exiting GTMenu, all active nonstructural members will be made inactive:



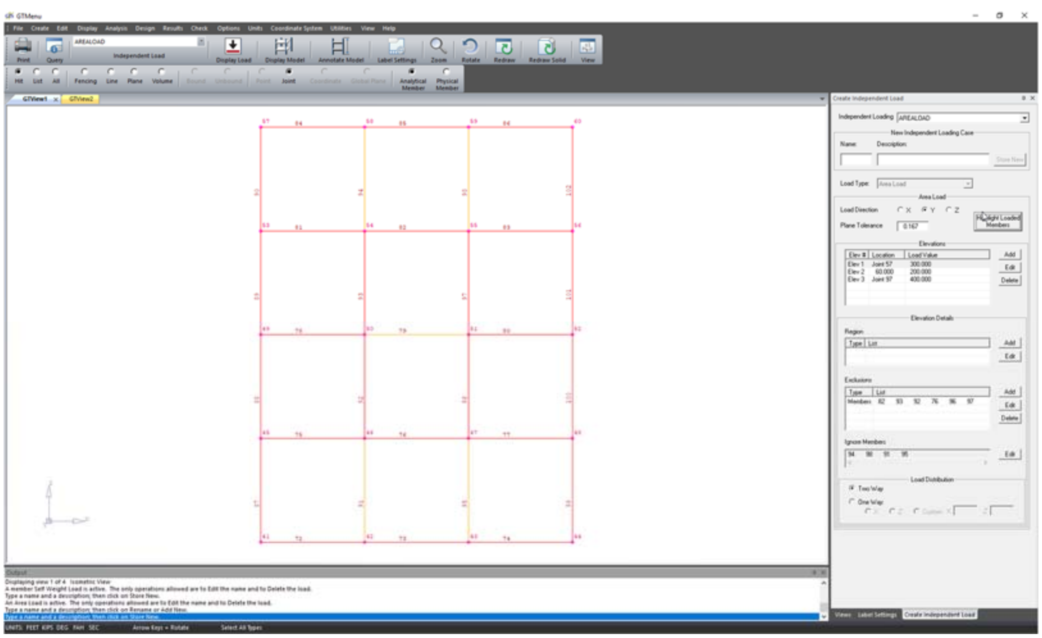
When entering GTMenu, you will be queried whether you want to view nonstructural members and joints:

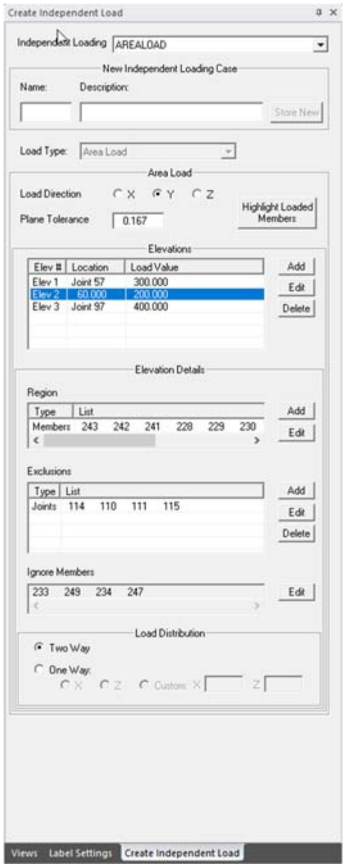


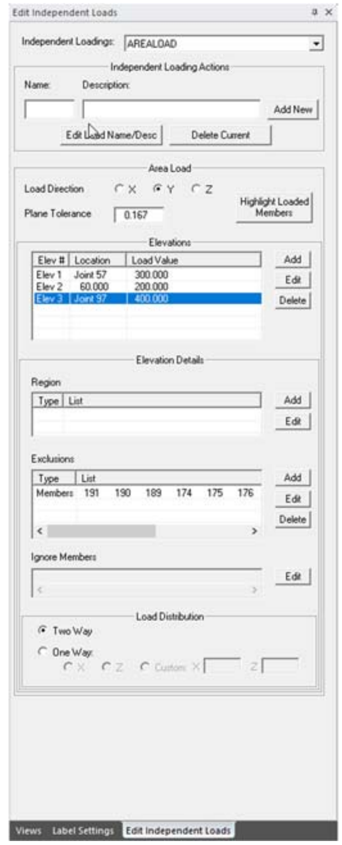
If nonstructural components exist when a GT STRUDL input file is created, the nonstructural components will be included in the generated input file.

3. A new Area Load creation and editing interface has been added to GTMenu. In previous versions, an Area Load could only be created using the AREA LOAD command or the GTShell Area Load dialog and could only be edited using the command.

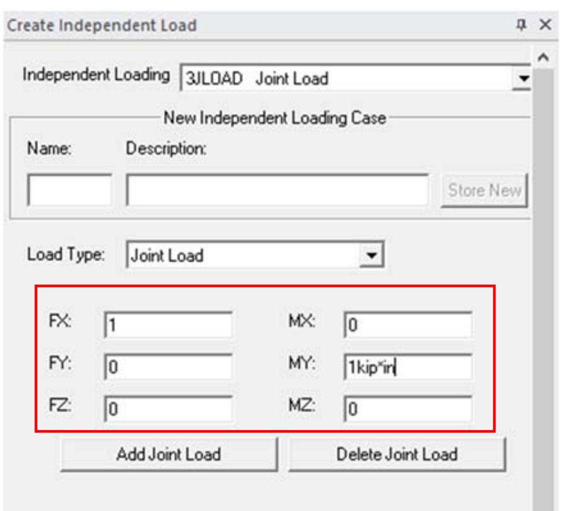
An Area Load can now be fully managed in GTMenu with its full graphic support to define bounding areas. Creation of an Area Load can be done through the menu "Create → Loads → Area Loads" (or Activate Independent Load) and selecting "Area Load" for Load Type. Editing can be done through the menu "Edit → Loads → Activate/Edit Independent Load". On the Create/Edit Loads dialog, the Area Load creation/editing interface will be displayed if the Load Type "Area Load" is selected or an existing and defined Area Load is selected. If an elevation is selected from the Elevations list on the Area Load interface, the GTMenu view will display a "plan view" at the selected elevation to help easily define the bounding areas at the elevation. The members loaded by Area Load will be highlighted as any change to the bounding areas is made.

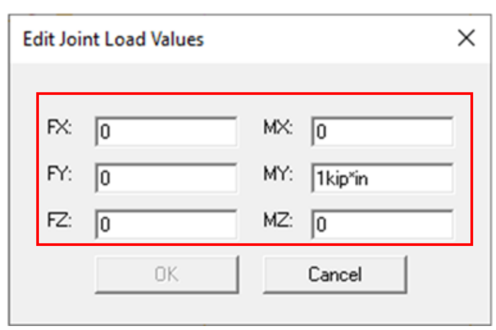




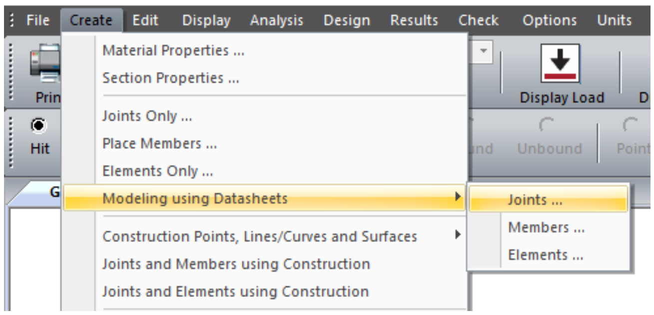


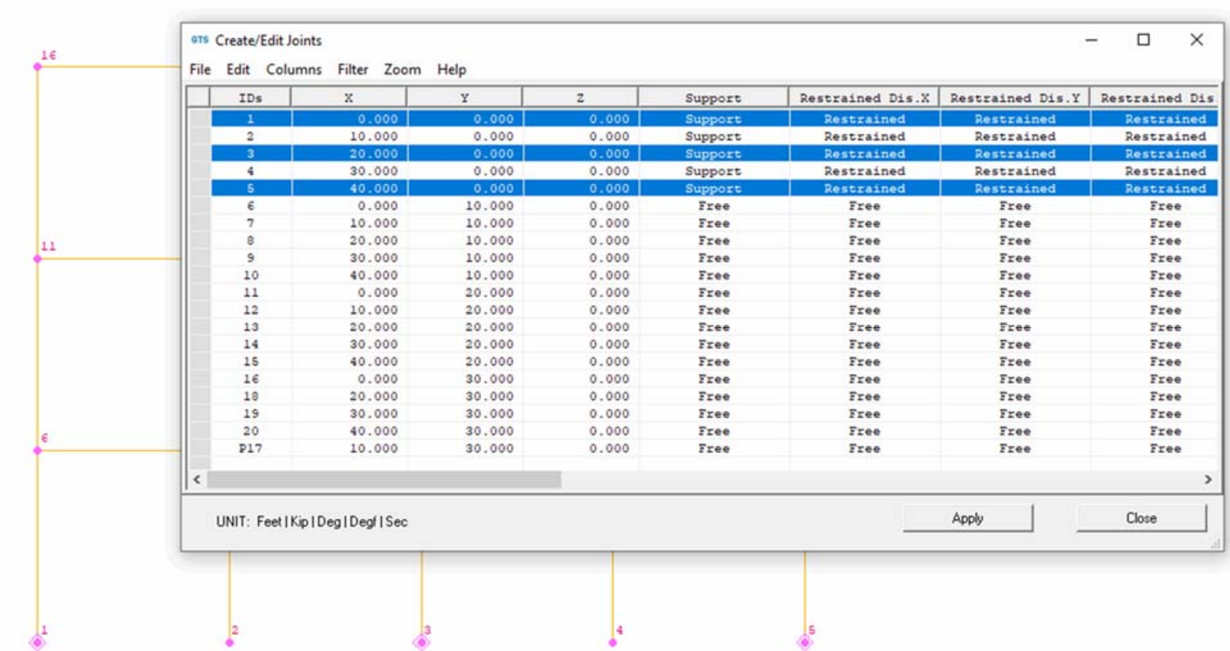
4. Force and Moment Units of Measure are now available in the Edit/Create Joint Load dialogs.





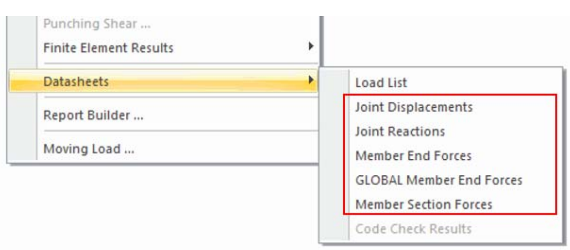
5. Modeling Datasheets have been added to GTMenu. The user can now create, edit, and see the properties of Joints, Members and Elements in Datasheets. In addition, selected objects in the datasheet are highlighted in the model graphics. Created and edited objects will be updated after clicking Apply button. For more information, you can use help menu from the datasheet.

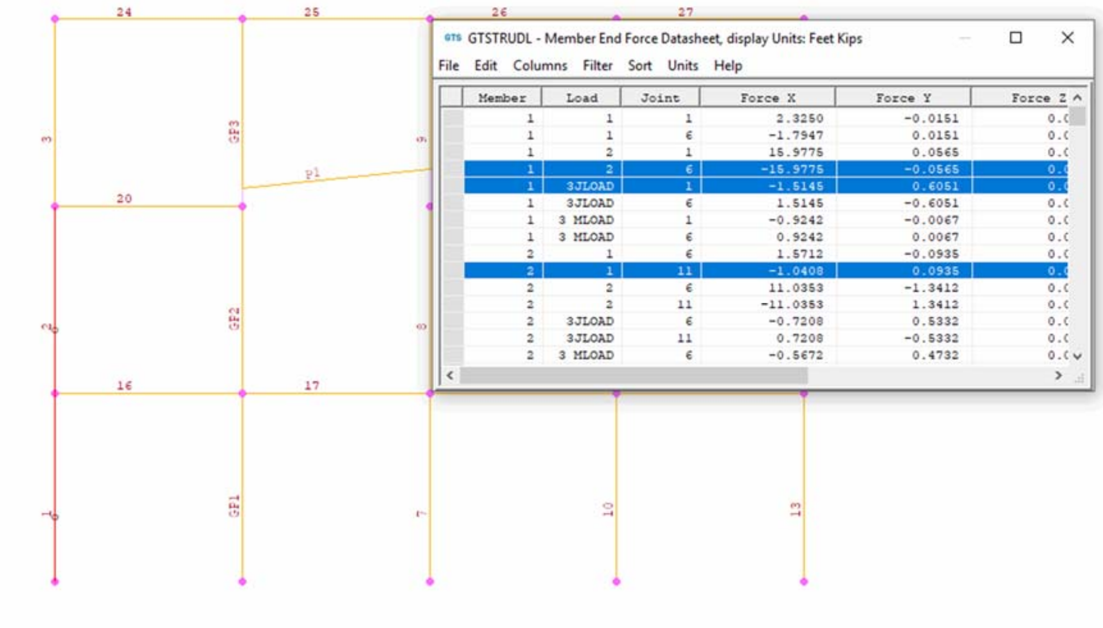




IDs	X	Y	Z	Support	Restrained Dis.X	Restrained Dis.Y	Restrained Dis
1	0.000	0.000	0.000	Support	Restrained	Restrained	Restrained
2	10.000	0.000	0.000	Support	Restrained	Restrained	Restrained
3	20.000	0.000	0.000	Support	Restrained	Restrained	Restrained
4	30.000	0.000	0.000	Support	Restrained	Restrained	Restrained
5	40.000	0.000	0.000	Support	Restrained	Restrained	Restrained
6	0.000	10.000	0.000	Free	Free	Free	Free
7	10.000	10.000	0.000	Free	Free	Free	Free
8	20.000	10.000	0.000	Free	Free	Free	Free
9	30.000	10.000	0.000	Free	Free	Free	Free
10	40.000	10.000	0.000	Free	Free	Free	Free
11	0.000	20.000	0.000	Free	Free	Free	Free
12	10.000	20.000	0.000	Free	Free	Free	Free
13	20.000	20.000	0.000	Free	Free	Free	Free
14	30.000	20.000	0.000	Free	Free	Free	Free
15	40.000	20.000	0.000	Free	Free	Free	Free
16	0.000	30.000	0.000	Free	Free	Free	Free
18	20.000	30.000	0.000	Free	Free	Free	Free
19	30.000	30.000	0.000	Free	Free	Free	Free
20	40.000	30.000	0.000	Free	Free	Free	Free
P17	10.000	30.000	0.000	Free	Free	Free	Free

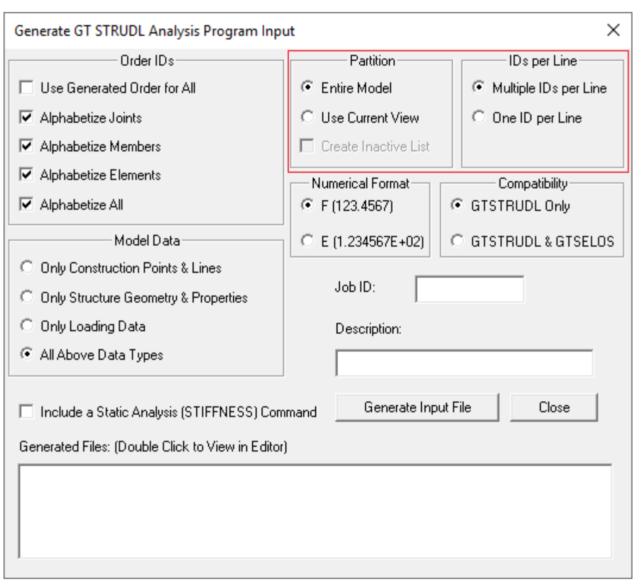
6. The Results Datasheets have been enhanced to highlight selected joints or members in the graphics area.



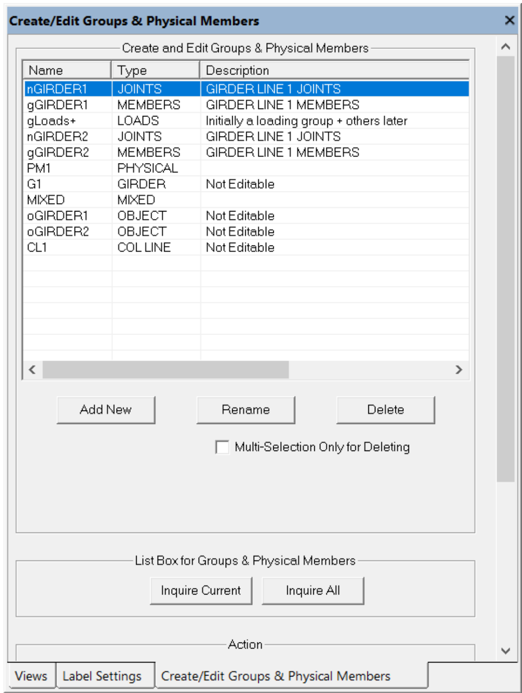


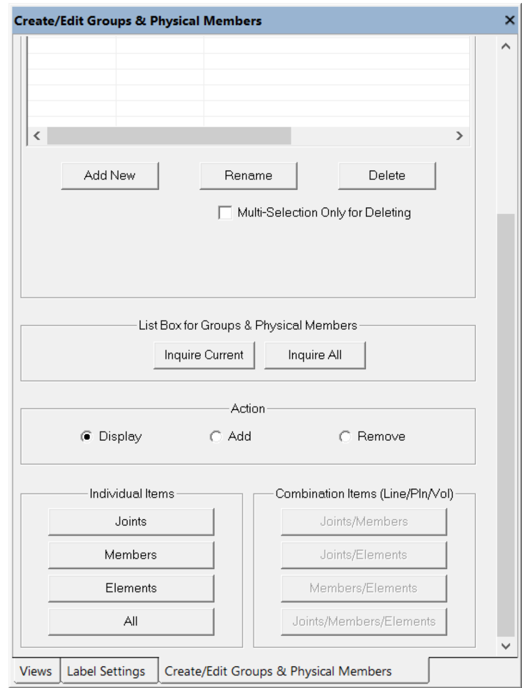
7. A new feature has been added to the Generate Input File dialog which will output one joint, member or element per line. This feature facilitates the comparison of two input files with a Text Diffing tool to easily locate changes, additions or deletions in the model.

In addition, a new feature has been activated in the dialog which will allow to create an input file of the full model when in a view but will inactivate all the entities that are not included in the view.



8. The GTMenu Create/Edit Groups & Physical Members Dialog has been updated and improved as shown below:

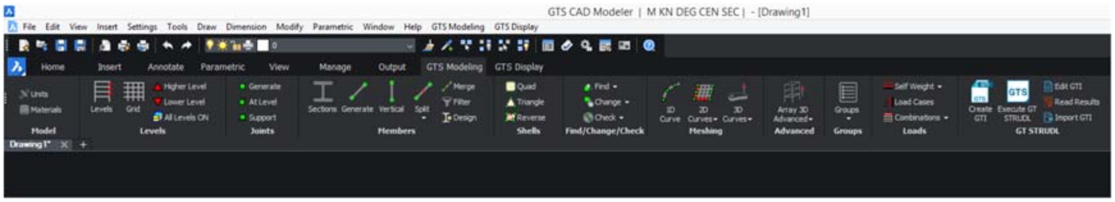




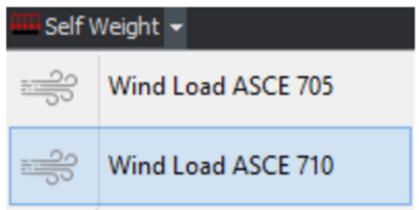
In the group table, the Type column terms have been updated to explicitly indicate the current type of each listed group when it was entered into the table. The Description for OBJECT, COLUMN LINE (COL LINE), and GIRDER has been given as “Not Editable,” indicating that these groups cannot be changed by clicking the edit buttons below, i.e., Rename, and Delete, and the Add and Remove Action items. All groups, including OBJECT, COL LINE, and GIRDER, can be listed using the Inquire All and Inquire Current buttons and used in the list selection functions for joints, members, finite elements, and loads.

2.3 CAD Modeler

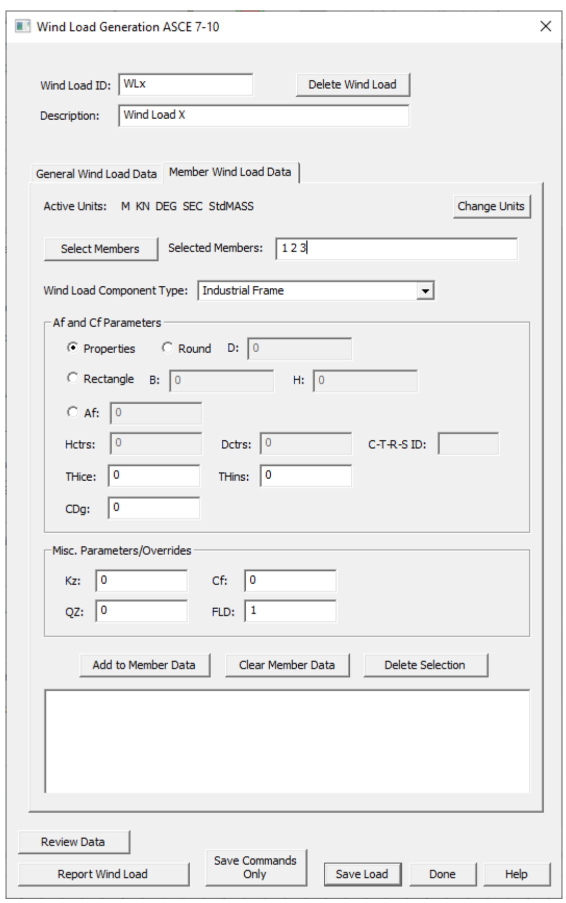
1. CAD Modeler now supports BricsCAD version 20 including all new functionality and the dark scheme.



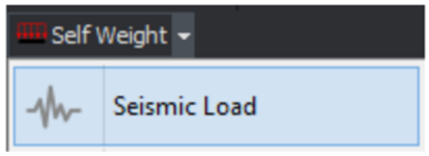
2. You can define Wind Loads using a similar form as GT Shell.



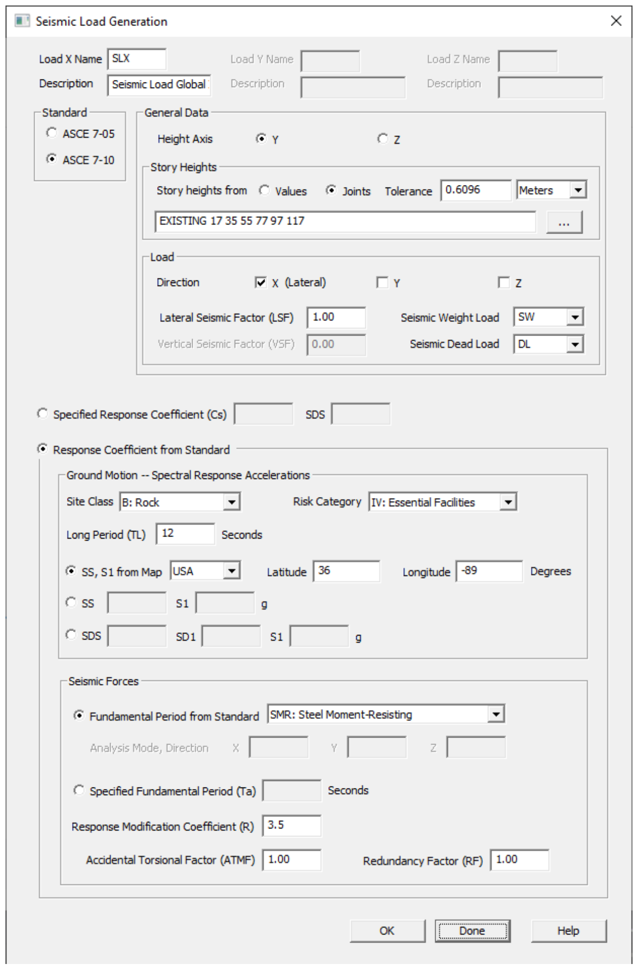
In addition, you can select members (to apply loads) interactively with mouse picks in the CAD environment.



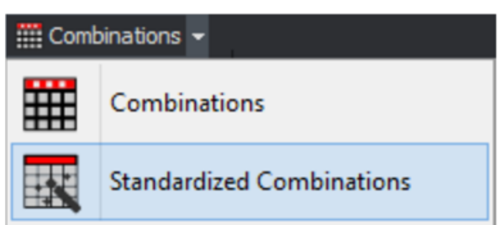
3. You can define Seismic Loads using a similar form as GT Shell.

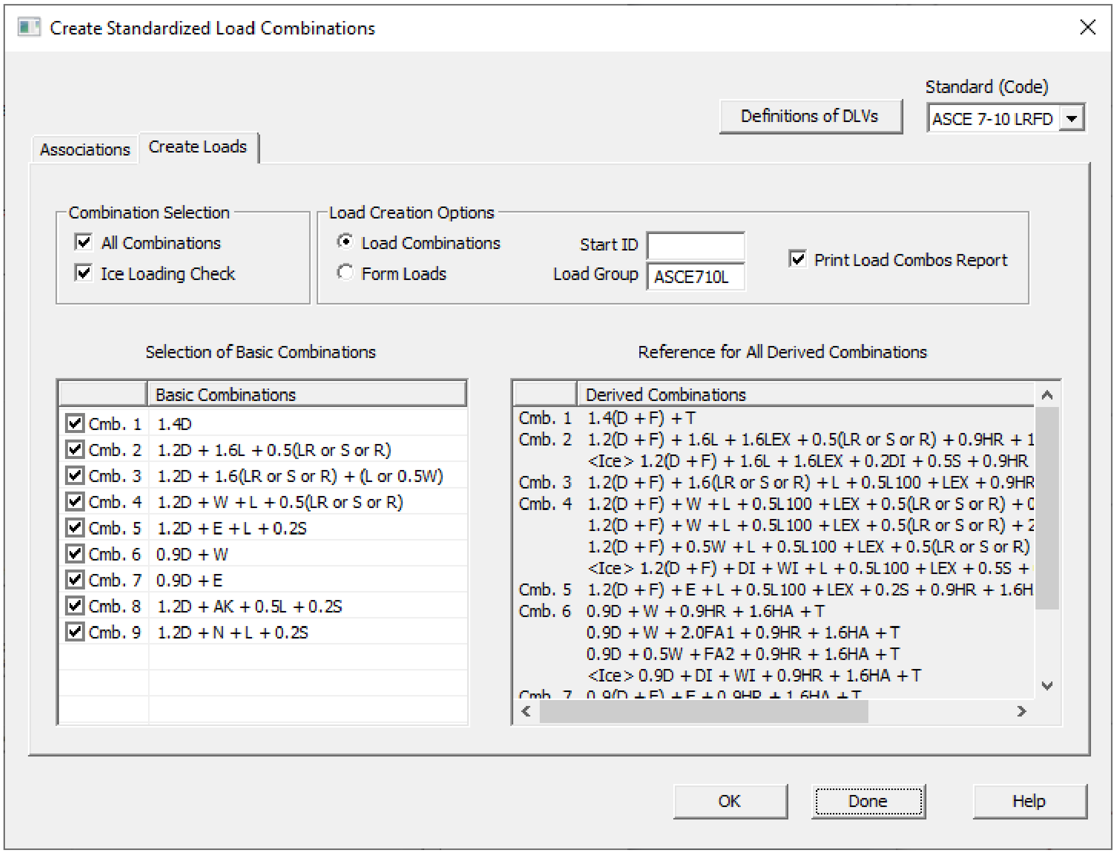


In addition, you can select joints (to define story heights) interactively with mouse picks in the CAD environment.

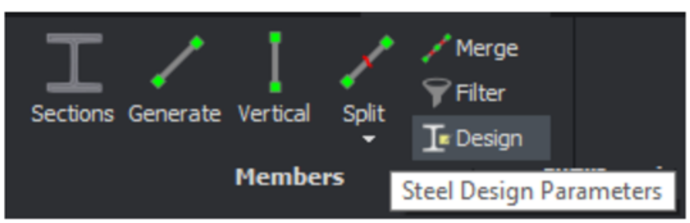


4. You can Create Standardized Combinations using the same dialog as GT Shell.

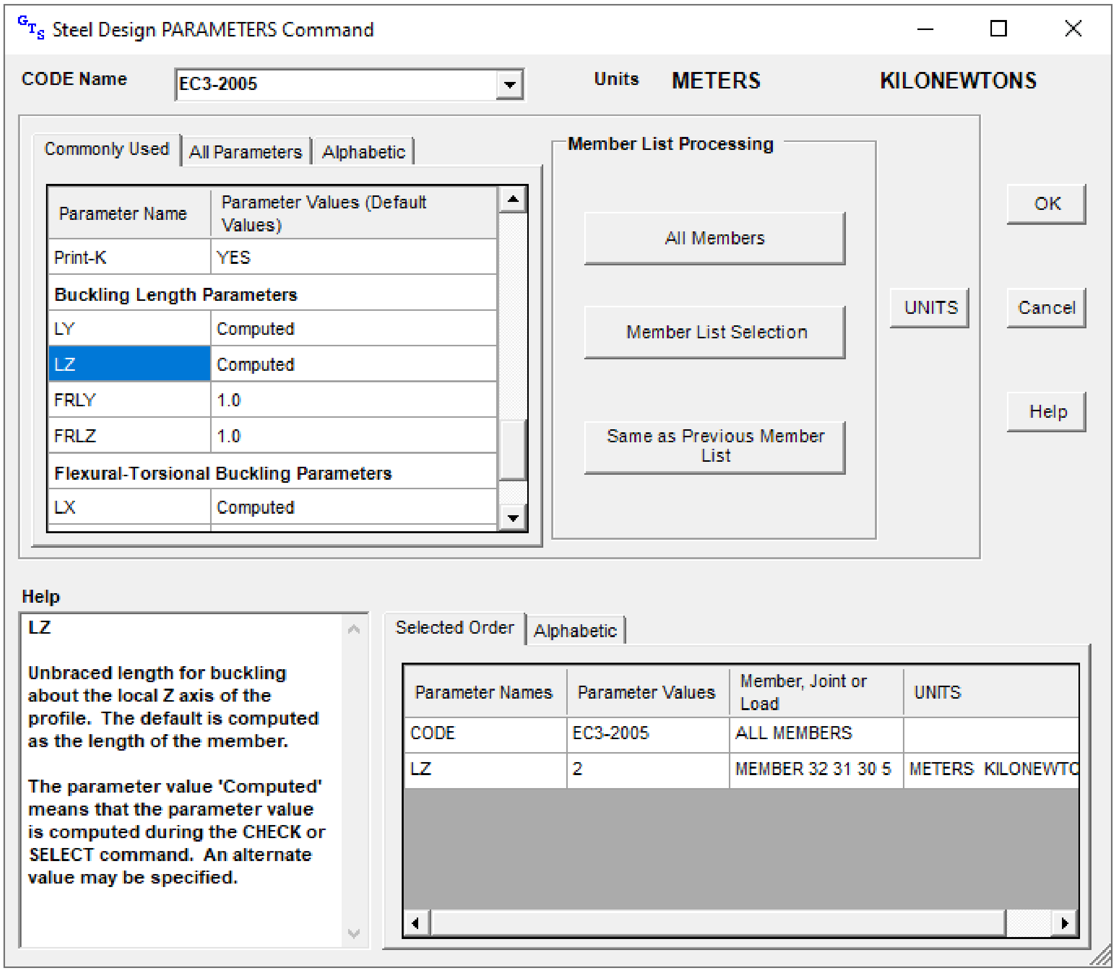




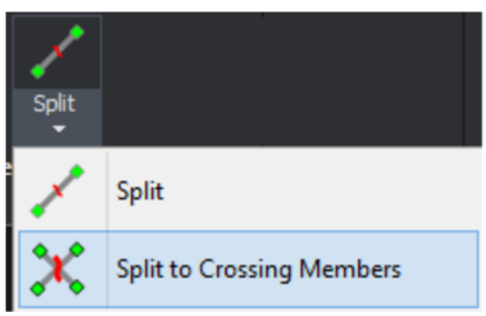
5. You can specify steel design parameters for AISC14, EC3, IS800, CSA-2014 and all other codes.



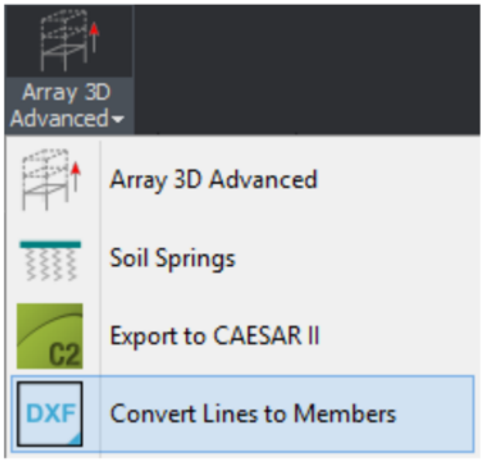
Parameters can be applied to ALL members or to specific members that can be selected interactively with mouse picks in the CAD environment.



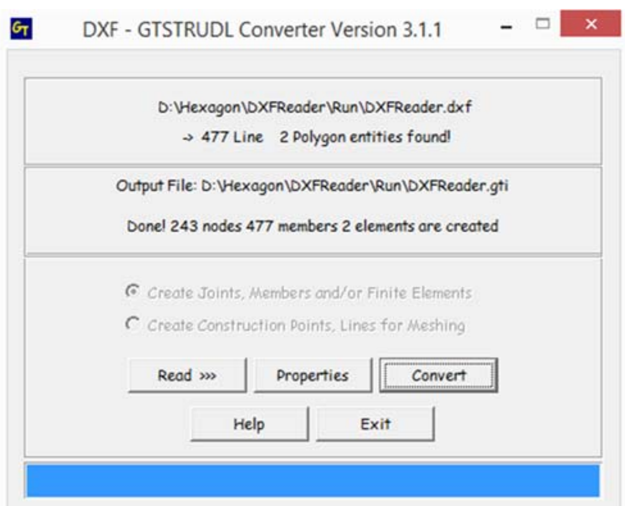
6. You can Split Members to Crossing Members, after being placed. You select the member to be spit and if there are any crossing members the selected member is split automatically at their intersections.

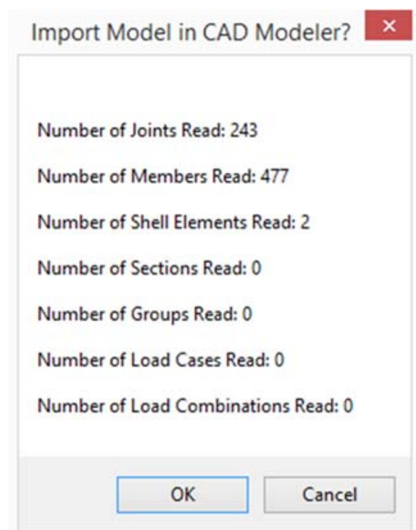


7. Convert lines and polylines to members and shell elements through the utility “DXF converter”.

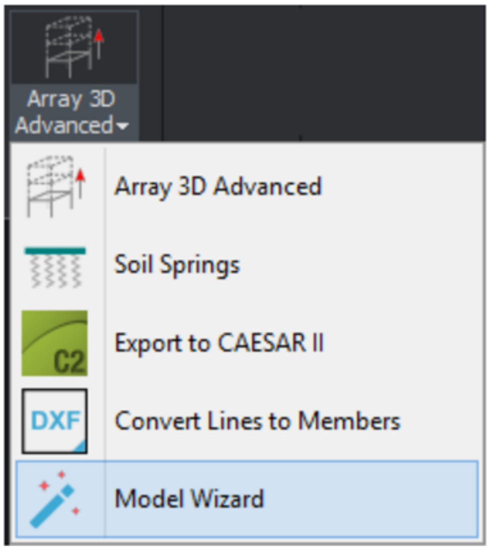


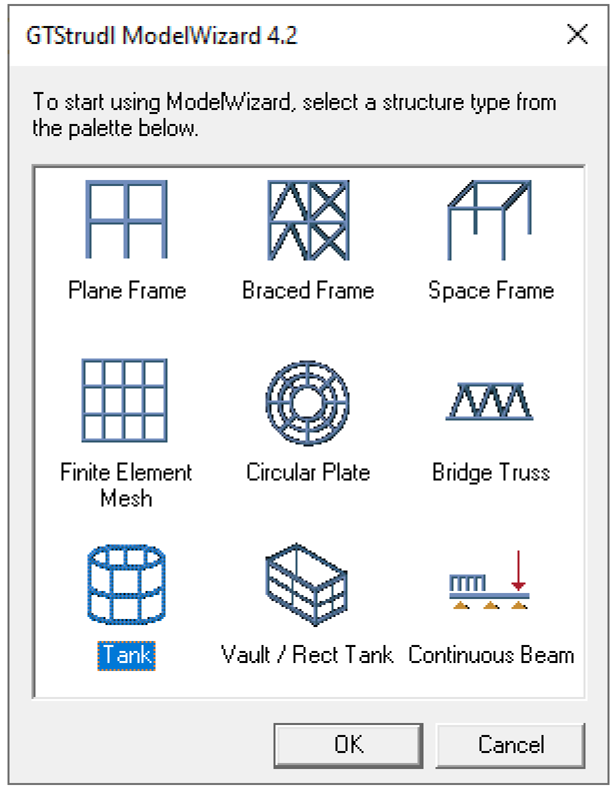
This command is not limited to DXF Files, but you can create new lines and polylines on the DWG that is currently open and convert them to structural members. Or open an existing DXF or DWG, select some lines and polylines, and convert them to members and elements.





8. You can now use Model Wizard to create and import the model to CAD Modeler. This option is very useful if you want to create a typical tank and further edit it in CAD Modeler or append it into an existing model.

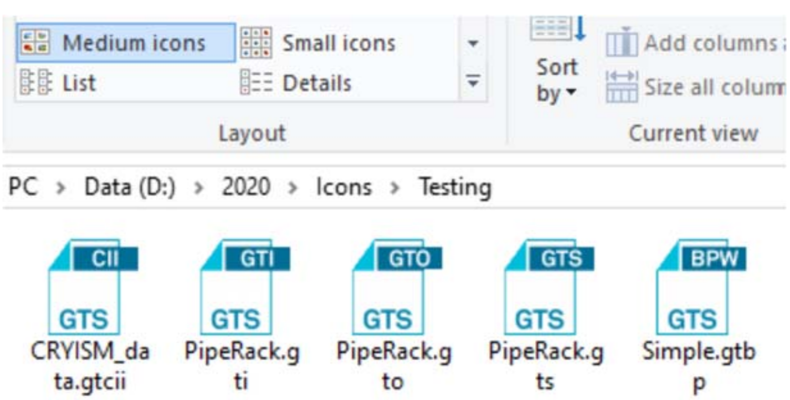




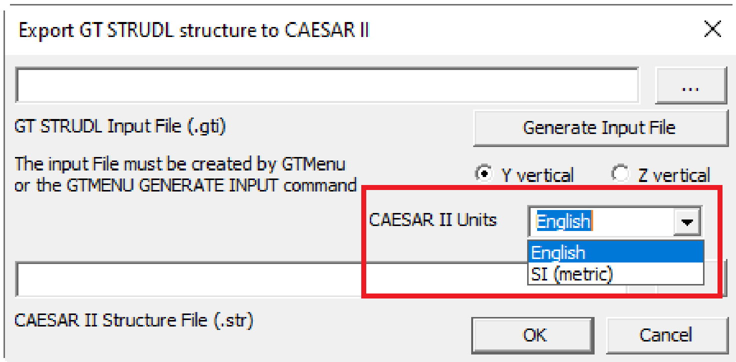
2.4 GT STRUDL Output Window (GTShell)

1. New icons have been added. The new icon for GT STRUDL is shown as a shortcut:

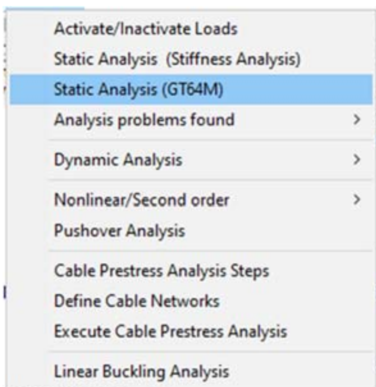
Also, new icons have been added for GT STRUDL file extensions, as shown below. These new icons will aid in knowing the file type when extensions are not displayed.



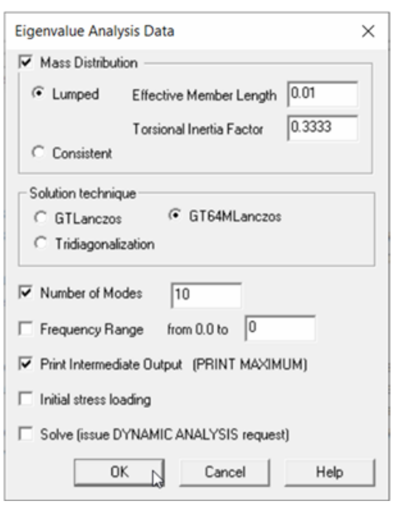
2. SI (metric) units have been added to "File → Export → CAESAR II". The current GT STRUDL units will be converted to the specified CAESAR II units upon export.



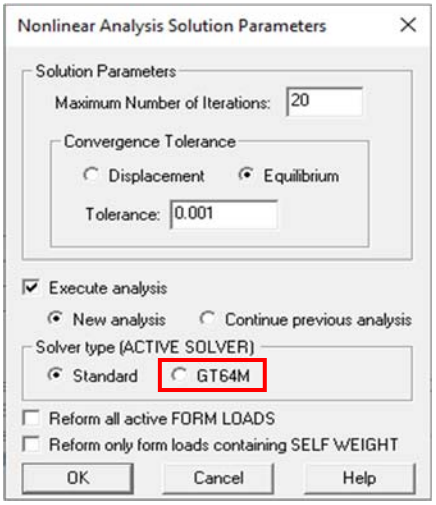
3. The menu option and dialogs that provide for the selection of a high-performance GTSES/GT64M solver option for stiffness, dynamic eigenvalue, and nonlinear static analyses have been modified as shown below. These now provide only the GT64M, 64bit, high-performance solver option. The GTSES, 32-bit option has been eliminated.



The Static Analysis (GT64M) selection from the GTShell Analysis main menu.



The GT64MLanczos selection in the Eigenvalue Analysis Data dialog from the GTShell Analysis/Dynamic Analysis/Eigenvalues (Frequencies) main menu.

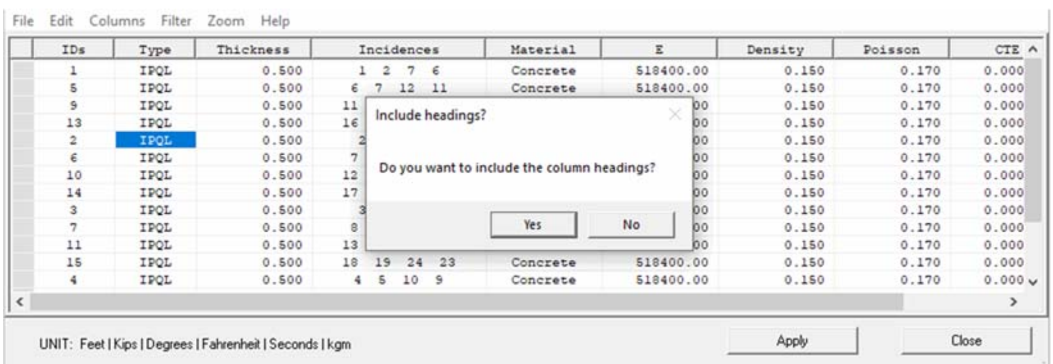


The GT64M selection in the Nonlinear Analysis Solution Parameters dialog from the GTShell "Analysis → Nonlinear/Second order → Nonlinear analysis and solution parameters" main menu.

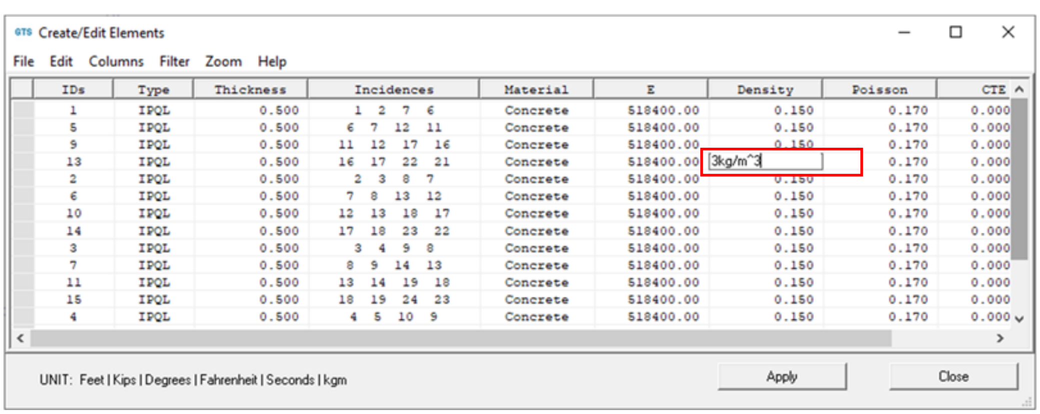
4. Enhancements to "Modeling → Data Sheets for Joints", Members and Elements.

The datasheets are now able to update changes after clicking the Apply button without closing the dialog. In addition, the current units are displayed at the bottom of the dialog.

The Copy function from Datasheet dialogs can now include column headings.



Units of Measure can now be used for numerical values.



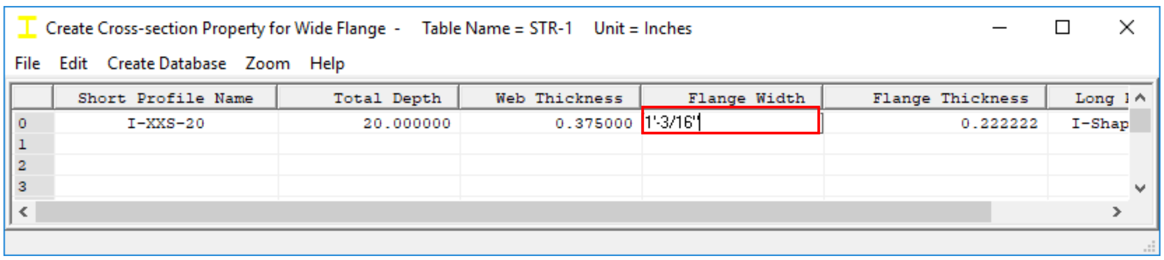
Changing font type, size, and style is now available in the datasheets.

Undo using Ctrl + Z is now available. In addition, redo using Ctrl +Y is also available.

5. Enhancements to Create Cross-Section Property Database Dialog

The section property database creator which is accessed through the menu "SteelDesign → Create, Transfer, and View Tables Profiles → Create Cross-Section Property Database", has been updated with features such as units of measure, zoom and copy/paste.

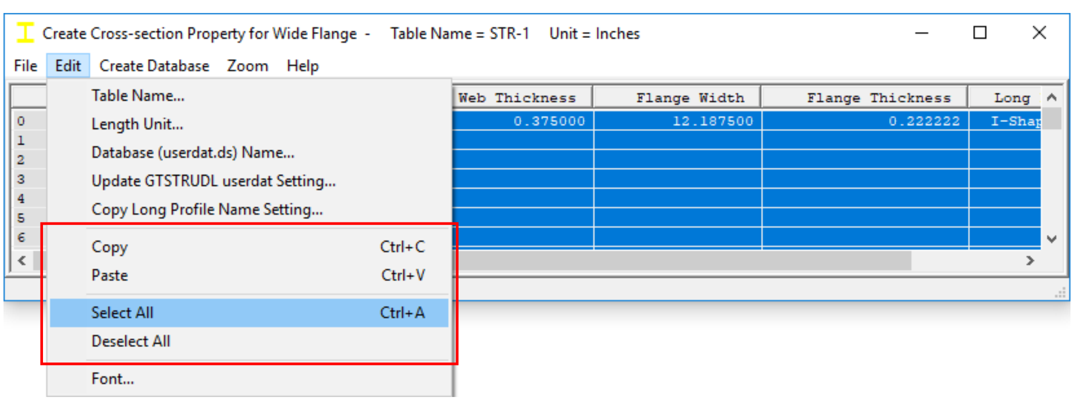
Units of Measure can now be used to enter numerical values for section properties.



The data can be entered as units of length for section dimensions and properties. For example, you can now specify 1′-3/16″ and it will automatically be converted to 12.1875 inches if your current active unit is inches.

Changing font size is now available by Zoom menu in the database creator. The hotkeys Ctrl + and Ctrl - can also be used to zoom in and zoom out, respectively. Zoom 100% takes font size to default setting.

Under Edit menu, Copy, Paste, Select All, and Deselect All functions are available now for data arrangement.



The column headings can now be included in the data selection by the Copy function, and the copied data can be used in other text tools.

2.5 IO

1. As per AISC Design Guide 29, Appendix D, three new GT STRUDL commands with the **TRANSFER CONNECTION** keyword have been added for connection design with transfer forces: DEFINE TRANSFER CONNECTION, PRINT TRANSFER CONNECTION, and LIST TRANSFER CONNECTION FORCES. The DEFINE TRANSFER CONNECTION command defines a connection with a joint and members connected at the joint where the members can be user-specified depending upon how the transfer forces should be calculated for a specific connection design. The PRINT TRANSFER CONNECTION command reports the defined connections with connection details. The LIST TRANSFER CONNECTION FORCES command adds up member forces of the specified members at a connection in order to calculate transfer forces at the connection. Under this command, all transfer forces are calculated for all load cases so that which load case governs for each connection.

```
DEFINE TRANSFER CONNECTION ‘CONN1’ ‘Connection 1’ –
     AT JOINT ‘JNT1’ –
     MEMBERS ‘MEM4’ ‘MEM7’ ‘MEM11’

PRINT TRANSFER CONNECTION ‘CONN1’

LIST TRANSFER CONNECTION FORCES CONNECTIONS ‘CONN1’
```

The data generated by the TRANSFER CONNECTION commands can be exported to DBX files so that external programs for connection design will be able to communicate with GT STRUDL through the DBX files.

Documentation
“2.1.12.22 TRANSFER CONNECTION Commands”, Volume 1, GT STRUDL User Reference Manual

2.6 Dynamic Analysis

1. The ASSEMBLE FOR STATICS/DYNAMICS Command

The ASSEMBLE FOR STATICS/DYNAMICS command feature allowing both STATIC and DYNAMIC specifications to be selected has been eliminated. From Version 2020 forward, the assembly of the static and dynamic equations of equilibrium must be requested in separate ASSEMBLE commands:

```
ASSEMBLE FOR STATICS

ASSEMBLE FOR DYNAMICS
```

Documentation
Assembly of Equilibrium Equations, Section 2.4.5.5.1, Volume 3, GT STRUDL User Reference Manual.

2.7 GTSES/GT64M High-Performance Solvers

1. As of Version 2020, the 32-bit GTSES and GTSELANCZOS high-performance static and eigenvalue analysis solvers are no longer supported and no longer can be selected by specifying the GTSES option of the ACTIVE SOLVER command.

$$\underline{\text{ACT}}\text{IVE } \underline{\text{SOLV}}\text{ER} \begin{Bmatrix} \underline{\text{GT64M}} \\ \underline{\text{STAN}}\text{DARD} \end{Bmatrix}$$

The high-performance static and eigenvalue analysis solutions are now provided exclusively by the 64-bit GT64M and GT64MLANCZOS solvers, which continue to be selected by the ACTIVE SOLVER GT64M command.

Documentation

The ACTIVE SOLVER command, Section 2.1.13.4, Volume 1, GT STRUDL User Reference Manual.

2. Prior versions of dynamic mode superposition analysis based on frequencies and mode shapes computed by the high performance GTSES/GT64M eigenvalue analysis, i.e. response spectrum analysis, modal transient analysis, harmonic analysis and steady state analysis, supported only the lumped mass matrix model. As of Version 2020, these analyses have been extended and improved by the addition of support for the sparse consistent mass matrix model. The existing INERTIA OF JOINTS CONSISTENT command is used to select this mass matrix model in the same way that it is used for standard analyses. Support for all the INERTIA OF JOINTS and MEMBER ADDED INERTIA command functions remains unchanged as well.

Documentation

Inertia Specification Command, Section 2.4.3.1, Volume 3, GT STRUDL User Reference Manual.

The Member Added Inertia Command, Section 2.4.3.2, Volume 3, GT STRUDL User Reference Manual.

2.8 DBX

1. A new DBX data format for connection design with transfer forces has been added and the format can be used to write the data about connections and their transfer forces. Note that the connection data set must be generated by the DEFINE TRANSFER CONNECTION command. The command to write the connection data into DBX files is in the following format:

```
WRITE TRANSFER CONNECTION 'filename' ALL CONNECTIONS
```

Note that all load cases will be considered for each connection in order to help assess which load case governs a connection.

2.9 Steel Design

1. A new design code for the Chinese National Standards GB 50017-2017 has been implemented. The GT STRUDL code name is GB-2017. It is based on the People's Republic of China National Standard, GB 50017–2017, Code for Design of Steel Structures, Published in December 12, 2017, Ministry of Housing and Urban-Rural Development of the People's Republic of China. This new code, GB-2017, may be used to select or check any of the following shapes:

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

   To use the GB-2017 code, you specify the Code Parameter as shown below:

   ```
   PARAMETERS
     CODE   GB-2017   ALL
   ```

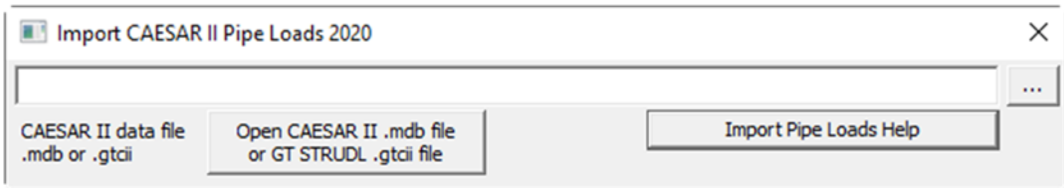
   The documentation for the GB-2017 code and additional parameters may be found upon execution of GT STRUDL by selecting menu "Help → Reference Documentation → Reference Manuals → Steel Design → GB-2017" in the GT STRUDL Output Window.

2. Welded connection design has been modified to compute allowable weld stress based on nominal tensile strength (allowable weld stress = 0.3×nominal tensile strength). Prior to GT STRUDL Version 2020, the allowable weld stress was computed based on minimum tensile strength (allowable weld stress = 0.3×minimum tensile strength).

3. PRINT LOADING DATA command has been enhanced to recompute self-weight automatically. If SELF WEIGHT command has been specified prior to PRINT LOADING DATA command, the self-weight is re-computed automatically as when STIFFNESS ANALYSIS or NONLINEAR ANALYSIS is given.

2.10 Import CAESAR II Pipe Loads

1. Several changes have been made to the Import CAESAR II Pipe Loads dialog, including the ability to read a .gtcii (dialog data base) file created during an earlier session. This simplifies the process of comparing subsequent CAESAR II analyses with the loads sent to GT STRUDL. Use the “Import Pipe Loads Help” button to access the documentation for this feature.



2. The .gtcii file now contains node-to-member mapping data and load case re-naming data if entered before creating the file to allow a complete recovery of an earlier session. This will allow you to interrupt a session before completing mapping or loading and return later to finish before creating a GT STRUDL loading file.

3. When creating a Piping Geometry file, the nodes and elements (JOINTS and MEMBERS in the .gti file) are declared as NONSTRUCTURAL. See the NONSTRUCTURAL note in the GTMenu section.

4. Other changes that were included with version 19.3 (compatible with GT STRUDL 2019) and exist in version 2020 are:

   a. When generating a Piping Geometry .gti file you may create GROUPS for the nodes and elements. You can also choose your own Location load name if desired. Also, piping that lies completely outside of the current structure can be ignored, which is useful for a long piping system spanning multiple racks.

   b. Help in creating useful GT STRUDL load names (8 characters) and descriptions (64 characters) is added to the Rename Load Cases dialog

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 GT STRUDL Commands

1. The problem with using the IGNORE MEMBERS option with AREA LOAD LIMITS specified by coordinates has been corrected. The problem never occurred when LIMITS were specified by joint ID or member list.

   Documentation:

   2.1.11.3.7.1, Vol. 1. The AREA LOAD Command

2. When performing a sequential construction analysis where part of the model was inactive for some of the loading cases, the Create Load Combination command could report that results are missing for some of the joints and members when performing a loading combination. The warning message indicating that results were missing for some of the joints in the model would incorrectly output the loading combination ID instead of the independent loading ID. This problem has been corrected.
   (No GPRF issued)

3. The application of LEX (Exclusive Live load) loading condition for Standardized Load Combination has been corrected for the following two cases:

   - AISC 14 ASD Load Combination equation 2 with Ice condition
     - Before this error correction, any defined LEX loads were not applied to the specified load combination even if they are active and defined by the STORE DESIGN LOAD TYPES command.
     - Now, every defined LEX load is properly applied to the load combinations as an exclusive load

   - All applicable combination equations of all four standards (ASCE710L, ASCE710A, AISC14L, AISC14A) especially when only a single LEX loading condition is defined
     - Previously, if only a single independent LEX load was defined, it was not applied to most of the applicable load combinations for all standards, although

if two or more LEX loads were defined, they were properly applied without any problem.

- Now, the LEX loads are properly applied to all applicable load combinations under all standards no matter how many LEX loads are defined

Documentation:

2.1.11.3.6.4, Vol. 1. The CREATE AUTOMATIC LOAD COMBINATION DESIGN Command

4. Three incorrect load factors applied to Standardized Load Combination equations have been corrected. The three factors are corrected as below:

- ASCE 7-10 LRFD Load Combination equation 4 with Ice condition
  - The factor for F (Fluid loads): 1.0 → 1.2

- ASCE 7-10 ASD Load Combination equation 5 with FA1 or FA2
  - The factor for FA1 (Flood loads in coastal A-Zones): 2.0 → 1.5
  - The factor for FA2 (Flood loads in noncoastal A-Zones): 1.0 → 0.75

Documentation:

2.1.11.3.6.4, Vol. 1. The CREATE AUTOMATIC LOAD COMBINATION DESIGN Command

5. If JOINT TEMPERATURE loads are included in a FORM LOAD, created by the FORM LOAD command, an error will occur when the FORM LOAD is reformed, which is the recalculation of individual component loads for joints, members and elements based on the original FORM LOAD specifications. JOINT TEMPERATURE loads, on the other hand, will not be recalculated, but rather erroneously incremented. This error has been corrected in Version 2020. FORM LOADS are reformed only when a specific instruction is issued with the FORM LOAD REFORM, STIFFNESS ANALYSIS REFORM, and NONLINEAR ANALYSIS REFORM commands.

GPRF: 2020.01

Documentation

The FORM LOAD REFORM Command; Section 1.11.3.2, Vol.1, Rev. 2019 and earlier.

The STIFFNESS ANALYSIS Command; Section 2.1.13.2, Vol. 1, Rev. 2019 and earlier.

The NONLINEAR ANALYSIS Command; Section 2.5.4.3, Vol. 3, Rev. 2019 and earlier.

3.2 GT STRUDL Output Window

1. Import CIS/2 has corrected the problem that occurred when the last profile table used has been deleted or moved. Now, if the last used profile table is not found, the latest supplied profile table will be assumed. The user can then change the assumed profile table to any available profile table before executing the import process.

3.3 GTMenu

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

1. In GT STRUDL 2019, there will be a crash during adding a second joint or member loads after defining new joint or member loads from the joint or member load list box at joint or member load datasheet. The Error has been corrected.

2. In GT STRUDL 2019, there was an error when adding a new datasheet line from Element 2D Spacing Dialog. If the Distance column was empty in the dialog, a new line would not be added to the datasheet. The Error has been corrected.

3. There was a reselection error in the Sum Force at a Cut dialog. The error occurred when a cut was selected and accepted, but then attempted to be reset. Member and elements are not selected. The Line and Plane modes on the Mode Bar will be grayed out and when the Reselect Cut button is selected, a message pop-up indicating that Line or Plane Mode needs to be selected first, but they will be grayed out. The Error has been corrected.

4. When generating an input file which contained nonlinear effects, the last line of the list of members could be omitted from the input file if the number of members which had nonlinear effects was a multiple of 8. This problem has been corrected.

5. Occasional crashes when reentering GTMenu. This problem has been corrected.

6. When using the Split Member option of the Refine Mesh Dialog in order to Split members along edges of subdivided element in a Refine Mesh operation, GTMenu may crash. This problem has been corrected.

7. Results in GTMenu are provided in the Cartesian Coordinate System. However, the Menu entries for selecting other coordinate systems were enabled when showing results. This will mislead the user into thinking that the results shown are in the selected coordinate system. In order to prevent this confusion, now the alternate Coordinate Systems entries in the Menu are disabled when showing Results.

8. GTMenu could become unresponsive when a value lower that 10 was entered for the animation speed in the Static Joints Displacements dialog has been corrected.

9. In GT STRUDL versions prior to 2020, GTMenu will crash when entered and when OBJECT groups containing only GROUP items were previously defined. This error has been corrected. OBJECT GROUPS are special-purpose groups that can be defined only by command for the primary purpose of model building using the OBJECT MOVE and COPY command functions.

   Documentation

   Object Creation and Manipulation, Section 2.1.6.7, Volume 1, GT STRUDL User Reference Manual, Version 2020.

3.4 GTShell

1. After creating new element constants from command line such as E, Density, CTE, Poisson Ratio, the element material constants are updated but the new name of materials are not updated. The problem results in missing material name in Element Datasheet dialog. The Error has been corrected.

3.5 CAD Modeler

1. Fixed problems when launching an UNDO command after the ARRAY command.

2. Fixed problems when extruding a polyline along a line to generate shell elements.

3. Removed a warning message about "AC_ATLAS.cuix" when CAD Modeler starts.

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 2020. 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, element and loadings as well as other data that could conflict with existing data already in the model in GTMenu.

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:

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:

$$\underline{\text{DES}}\text{IGN }\underline{\text{SLAB}}\text{ (}\underline{\text{REIN}}\text{FORCEMENT) (}\underline{\text{US}}\text{ING) -}$$

$$\left\{\begin{array}{l}\underline{\text{WOOD}}\text{ (}\underline{\text{AND}}\text{) (}\underline{\text{ARM}}\text{ER)}\left\{\begin{array}{l}\rightarrow\underline{\text{AVE}}\text{RAGE}\\ \underline{\text{MAX}}\text{IMUM}\end{array}\right\}\\ \\ \underline{\text{CALC}}\text{ULATE (}\underline{\text{RES}}\text{ULTANT) (}\underline{\text{ELE}}\text{MENT) (}\underline{\text{FOR}}\text{CES)}\end{array}\right\}\text{ (}\underline{\text{AL}}\text{ONG) -}$$

$$\text{(}\underline{\text{CUT}}\left\{\begin{array}{c}\text{'a'}\\ i_1\end{array}\right\}\text{)}\left\{\begin{array}{l}\underline{\text{JOI}}\text{NTS}\\ \underline{\text{NOD}}\text{ES}\end{array}\right\}\text{ list}_1\ \underline{\text{ELE}}\text{MENT list}_2\text{ (}\underline{\text{TABLE}}\left\{\begin{array}{l}\rightarrow\underline{\text{ASTM}}\\ \underline{\text{UNESCO}}\end{array}\right\}\text{) -}$$

$$*\left\{\begin{array}{l}\underline{\text{TOP}}\text{ (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S } i_2\text{) (}\underline{\text{SPA}}\text{CING } v_1\text{)}\\ \underline{\text{BOT}}\text{TOM (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S } i_3\text{) (}\underline{\text{SPA}}\text{CING } v_2\text{)}\\ \underline{\text{BOTH}}\text{ (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S } i_4\text{) (}\underline{\text{SPA}}\text{CING } v_3\text{)}\end{array}\right\}\text{ -}$$

$$\left\{\begin{array}{l}\rightarrow\underline{\text{IN}}\text{NER (}\underline{\text{LAY}}\text{ER)}\\ \underline{\text{OUT}}\text{ER (}\underline{\text{LAY}}\text{ER)}\end{array}\right\}\text{ (}\underline{\text{COV}}\text{ER } v_4\text{) (}\underline{\text{LIN}}\text{EAR (}\underline{\text{TOL}}\text{ERANCE) } v_5\text{) -}$$

$$\text{(}\underline{\text{TOR}}\text{SIONAL (}\underline{\text{MOM}}\text{ENT) (}\underline{\text{WARN}}\text{ING) } v_6\text{)}$$

where,

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

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

Explanation:

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

The user must first define the cut. A cut is defined by a start and end node ID, and an element ID in the plane of the cut. The user has the option of giving each cut an alphanumeric name for organizational purposes. The purpose of the required element ID is to determine the appropriate plane to design if multiple planes of finite elements intersect along the cut, as defined by the start and end node. An example where this might occur is the intersection of a slab with a shear wall. In this case, a misleading design could be generated if the slab was designed using the forces in the shear wall. The cut definition constitutes all information required to compute the resultant forces acting along the cut.

The total moment acting on a cut cross-section is computed using one of two methods. The use of moment resultants, also known as the Wood and Armer method, is implemented as the default method. In this method, the moment resultants MXX, MYY, and MXY are resolved on a per node basis along the cut, and either the average effect or the maximum effect on the cut is applied to the entire cross-section.

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

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

The user may also specify which layer of bars to be designed, using the modifier INNER or OUTER. These refer to the location of reinforcing bars on each surface. At most slab locations, reinforcement is placed in two perpendicular directions on both surfaces of the slab. Since each layer of reinforcement cannot occupy the same space, one layer must be placed on top of the other. OUTER refers to the layer closest to the surface, while INNER refers to the layer nearest the center of the slab.

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

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

Requirements:

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

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

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

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

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

Usage:

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

Usage Example: Description of Example Structure

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

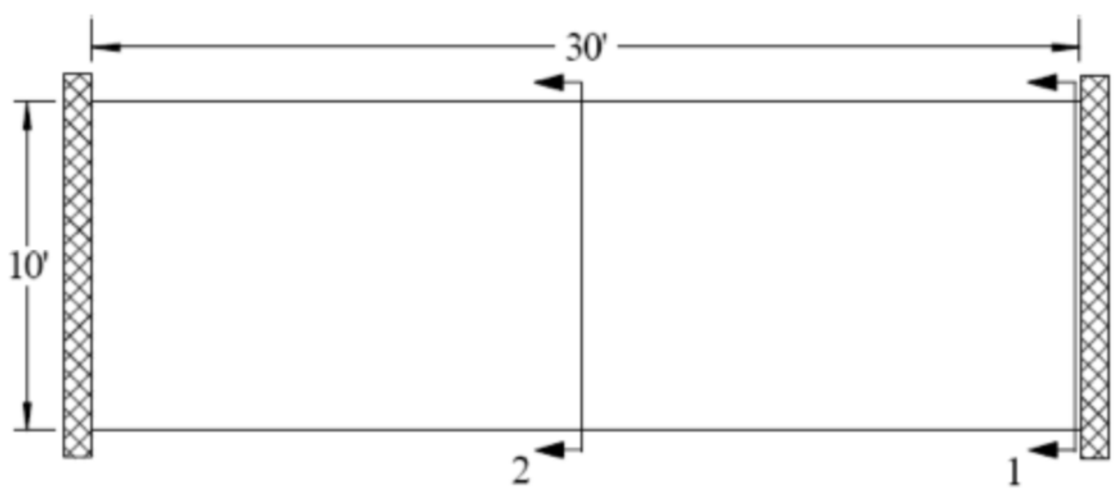


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

GT STRUDL Finite Element Model

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

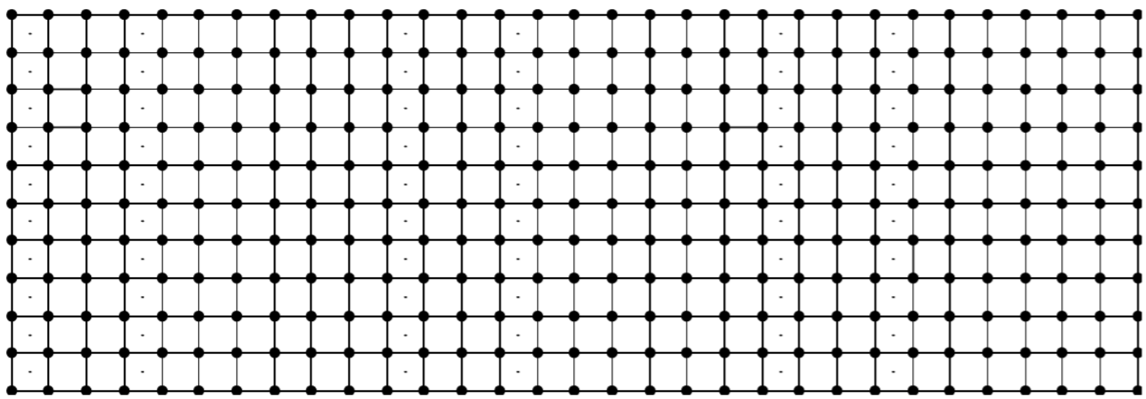


Figure 5.2.3-2 Example Finite Element Model

Definition of Cut Cross-Sections

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

Cut 1-1:

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

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

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

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

The GT STRUDL output for this example is as follows:

```
#
 ** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **

      PROBLEM - VFE103    TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS

      RELEVANT ACTIVE UNITS:  INCH LB

      NUMBER OF ACTIVE LOADINGS:     1

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


 ** ELEMENT FORCE IMPLEMENTATION **

 ** DESIGN MOMENT ENVELOPE **

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

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


 ** SLAB CROSS-SECTION **

      Width        Depth          FCP           FY         Cover        Layer
     ________________________________________________________________________

     120.00       12.00        4000.00       60000.00     0.750        Inner

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

      Face        Bar    Spacing   AS PROV'D    MOMENT STRENGTH    MOMENT REQ'D    STATUS
     _____________________________________________________________________________________

      TOP         # 5    13.000      2.862        1561006.4280    1354381.4844     PASSES

      BOTTOM            ( Reinforcement Not Required )
```

Cut 2-2:

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

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

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

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

The GT STRUDL output for this example is as follows:

```
  ** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **

       PROBLEM - VFE103    TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS

       RELEVANT ACTIVE UNITS:  INCH LB

       NUMBER OF ACTIVE LOADINGS:     1

       REINFORCEMENT ORIENTATION PERPENDICULAR TO A CUT BEGINNING AT NODE 166
         AND TERMINATING AT NODE 176      AND IN THE PLANE OF ELEMENT 141


  ** WOOD & ARMER IMPLEMENTATION **

       Design using average result acting on section.


  ** DESIGN MOMENT ENVELOPE **

       NEGATIVE MOMENT =            0.00   DUE TO LOAD   150psf
       POSITIVE MOMENT =       671358.19   DUE TO LOAD   150psf


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


  ** SLAB CROSS-SECTION **

       Width        Depth          FCP           FY         Cover        Layer
      ________________________________________________________________________

      120.00       12.00        4000.00       60000.00     0.750        Outer

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

       Face       Bar    Spacing   AS PROV'D    MOMENT STRENGTH    MOMENT REQ'D    STATUS
      _________________________________________________________________________________________

       TOP                ( Reinforcement Not Required )

       BOTTOM     M14    10.000      2.864        1664920.7190     671358.1875     PASSES
#
```

5.2.3 ASCE4805 Code for the Design of Steel Transmission Pole Structures

The steel design code, ASCE4805, which is based on the 2005 edition of the ASCE/SEI, *Design of Steel Transmission Pole Structures* Specification has been implemented as a pre-release feature. The ASCE/SEI 48-05 Specification is based on ultimate strength methods using factored loads.

The ASCE4805 Code may be used to select or check any of the following shapes:

Design for axial force, bi-axial bending, and torsion:

Pipes

Regular Polygonal Tubes

Structural Tubing

The documentation for the ASCE4805 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and “ASCE4805" in the GT STRUDL Output Window.

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

CALCULATE ERROR (ESTIMATE) (BASED ON) -

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

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

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

The L2 norm is given by:

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

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

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

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:

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

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

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

Normalized Percent Maximum Difference

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

Normalized Percent Difference from Average Method

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

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:

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

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 iR/'aR', the ROTATE LOADING command performs the following operations and copies the results into the destination loading condition:

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

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

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

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

Examples:

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

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

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

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

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

Error Messages:

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

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

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

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

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

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

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

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

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

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

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

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

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

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

5.4.2 REFERENCE COORDINATE SYSTEM Command

General form:

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

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

Explanation:

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

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

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

(1) Define the origin and rotation of coordinate axes of the reference system with respect to the global coordinate system, and
(2) define three joints or the coordinates of three points in space.

In either case, i_1 or 'a_1' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x, v_y, and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x, v_y, and v_z, are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive directions of these axes; v_1, v_2, and v_3 are the rotation angles 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 POINT COORDINATE and LINE INCIDENCES Commands

GTMenu can now write construction geometry commands to an input file, which can be read later into GT STRUDL in order to initialize the construction geometry of GTMenu. The two commands written are "GTMenu POINT COORDINATES" and "GTMenu LINE INCIDENCES".

(1) **GTMenu POINT COORDINATES**

General Form:

GTMenu POINT COORDINATES

$\{'a_1'\}$ coordinate-specs$_1$

•

•

•

$\{'a_n'\}$ coordinate-specs$_n$

Elements:

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

where,

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

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

(2) **GTMenu LINE INCIDENCES**

General Form:

GTMenu LINE INCIDENCES

$\{'a_1'\}$ type$_1$ incidence-specs$_1$

•
•
•

$\{'a_n'\}$ type$_n$ incidence -specs$_n$

Elements:

$$\text{type} = \left\{ \begin{array}{l} \rightarrow \underline{\text{LIN}}\text{E} \\ \underline{\text{POL}}\text{YNOMIAL (}\underline{\text{CUR}}\text{VE)} \\ \underline{\text{ARC}}\text{ (}\underline{\text{TEM}}\text{PLATE)} \\ \underline{\text{CEN}}\text{TERED (}\underline{\text{ARC}}\text{) }\underline{\text{PER}}\text{CENT } v_1 \\ \underline{\text{BEZ}}\text{IER (}\underline{\text{CUR}}\text{VE)} \\ \underline{\text{SPL}}\text{INE (}\underline{\text{CUR}}\text{VE) (}\underline{\text{ORD}}\text{ER } k_2\text{)} \end{array} \right\}$$

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

where,

$'a_1', 'a_2', ..., 'a_n'$ = 1 to 8 character alphanumeric Line/Curve identifiers beginning with C (i.e. C1, C2).

$'\text{point}_1', ..., '\text{point}_p'$ = 1 to 8 character alphanumeric Point identifiers beginning with P (i.e. P1, P2).

v_1 = positive number (integer or real).

k_2 = integer between 2 and the number of incidences.

1, 2, ...,p = Point subscripts for a Line/Curve. The following table gives the number of Points used to specify different types of Line/Curve:

type	number of incidences
LINE	2 - 500
POLYNOMIAL CURVE	2 - 10
ARC TEMPLATE	3
CENTERED ARC	3
BEZIER CURVE	2 - 10
SPLINE CURVE	2 - 10

5.4.4 GTMenu SURFACE DEFINITION Command

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

General Form:

$\underline{\text{GTMenu}}$ $\underline{\text{SUR}}\text{FACE}$ $\underline{\text{DEF}}\text{INITION}$

$\{'a_1'\}$ surface-specs$_1$

•

•

•

$\{'a_n'\}$ surface-specs$_n$

Elements:

$$\text{surface-specs} = \left\{ \begin{array}{l} (\underline{\text{PAT}}\text{CH } \underline{\text{SUR}}\text{FACE } \underline{\text{SPA}}\text{CING}) \text{ iu iv patch-specs} \\ (\underline{\text{SUR}}\text{FACE } \underline{\text{OF}}) \ \underline{\text{REV}}\text{OLUTION } (\underline{\text{SPA}}\text{CING}) \text{ iu iv sor-specs} \end{array} \right\}$$

$$\text{patch-specs} = \text{U } (\underline{\text{CUR}}\text{VES}) \ 'b_1' \ldots 'b_n' \text{ V } (\underline{\text{CUR}}\text{VES}) \ 'c_1' \ldots 'c_m'$$

$$\text{sor-specs} = (\underline{\text{REV}}\text{OLUTION } \underline{\text{ANG}}\text{LE}) \text{ v axis-specs U } (\underline{\text{CUR}}\text{VE}) \ 'b_1'$$

$$\text{axis-specs} = (\underline{\text{AXI}}\text{S}) \left\{ \begin{array}{l} \underline{\text{POI}}\text{NTS } 'd_1' \ 'd_2' \\ \underline{\text{COO}}\text{RDINATES } \underline{\text{STA}}\text{RT } x_1\ y_1\ z_1 \ \underline{\text{END}} \ x_2\ y_2\ z_2 \end{array} \right\}$$

where,

$'a_1', 'a_2', \ldots, 'a_n'$	=	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.

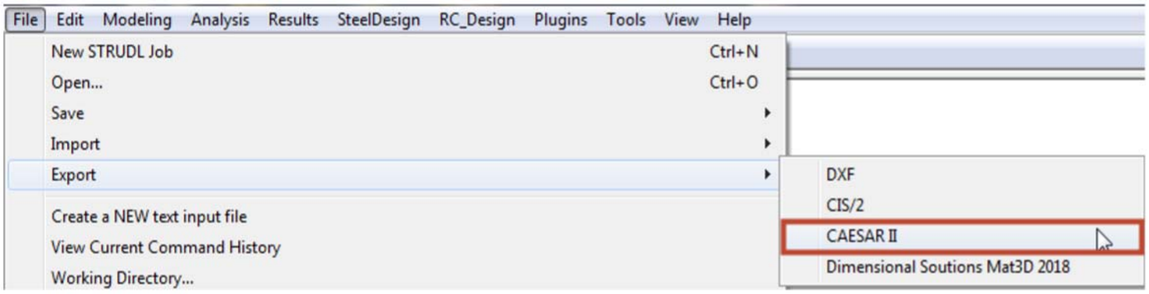
'b_1', 'b_2', ..., 'b_n'	=	1 to 8 character alphanumeric Line/Curve IDs for U direction. n must be greater than or equal to 1 and less than or equal to 10. Line/Curve IDs begin with C (i.e. C1,C2).
'c_1', 'c_2', ..., 'c_m'	=	1 to 8 character alphanumeric Line/Curve IDs for V direction. m must be greater 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:

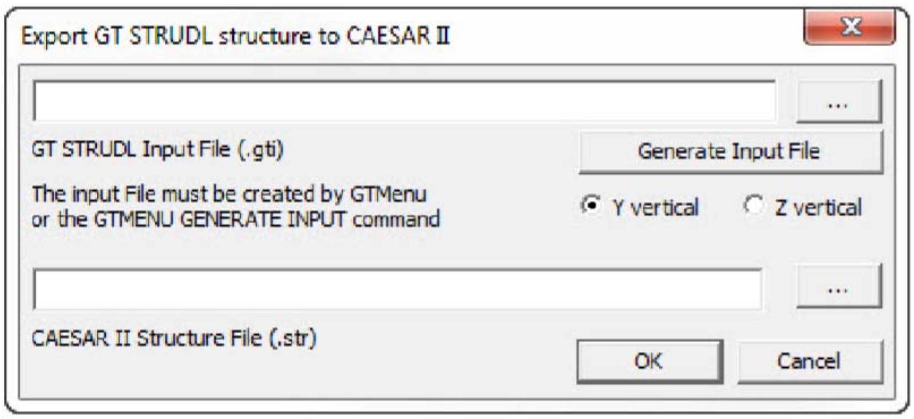
```
GTMenu SURFACE DEFINITION
    'S1' PATCH SURFACE SPACING 10 20 -
        U CURVES 'C1' -
        V CURVES 'C2'

    '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'
```

5.4.5 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:

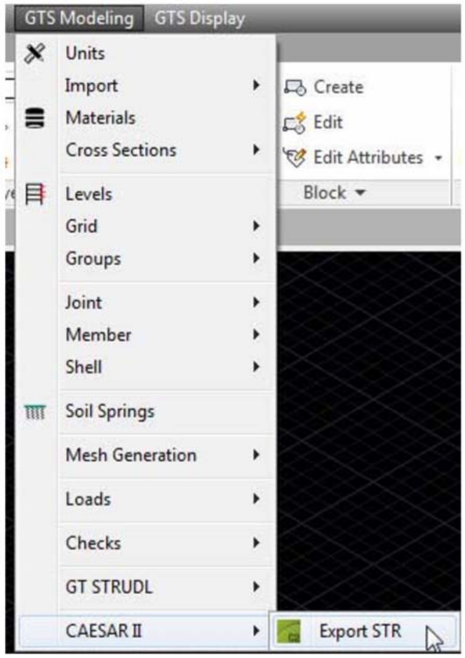


After selecting CAESAR II, the following dialog will pop-up:



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:

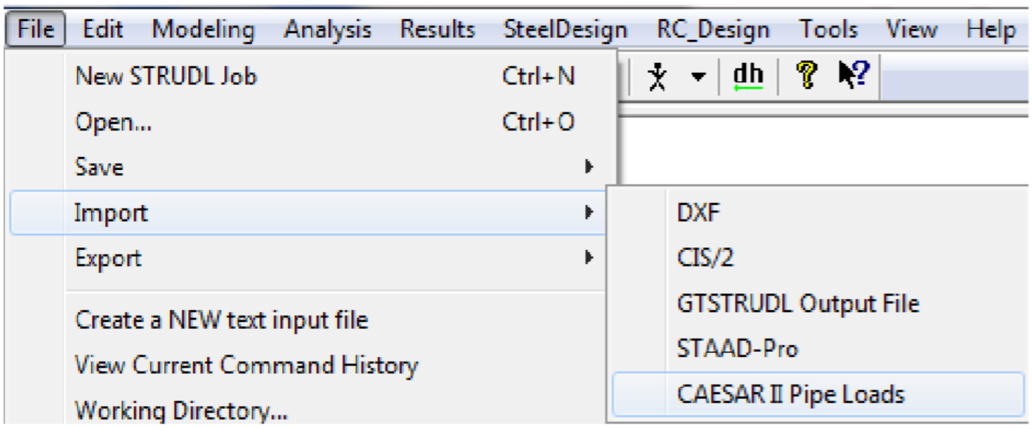
```
WARNING: Section L1x1x1/4 is not available in CII, please use another one
or edit
F:\\HexagonPPM\CaesarII\PlantStructure\\PStructure_0708_01.str file
manually
```

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).

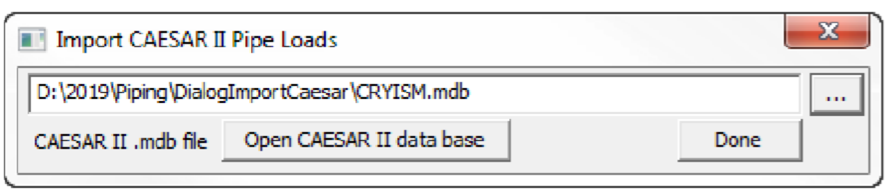
5.4.6 Import CAESAR II Pipe Loads

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:



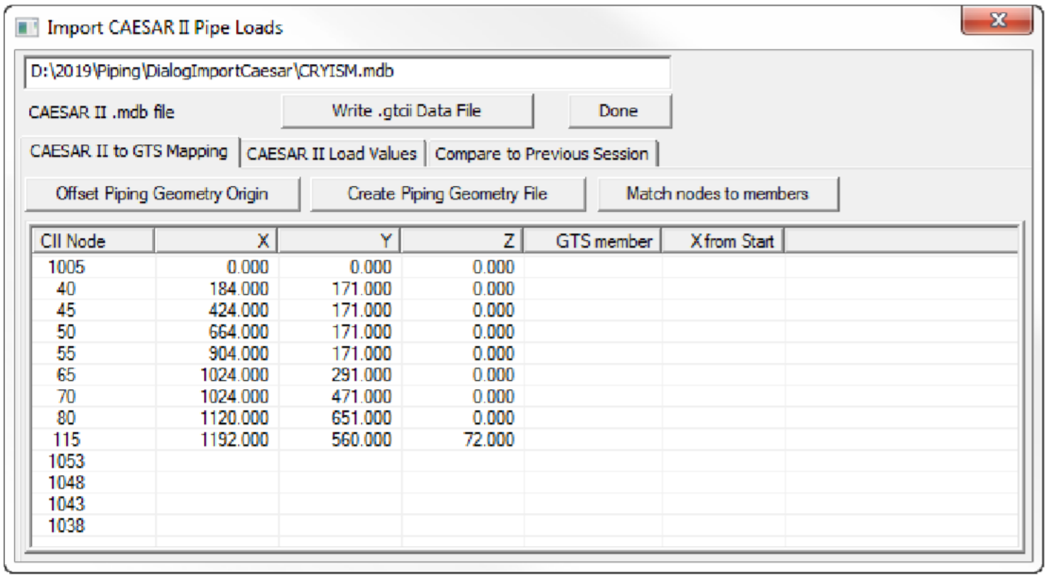
Clicking the menu item will launch the opening dialog, and if there is structural data in the GT STRUDL data base, write files so the dialog can get the structural model data. Choose a .mdb file by typing or browsing with the button and then click the 'Open CAESAR II data base' button.



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



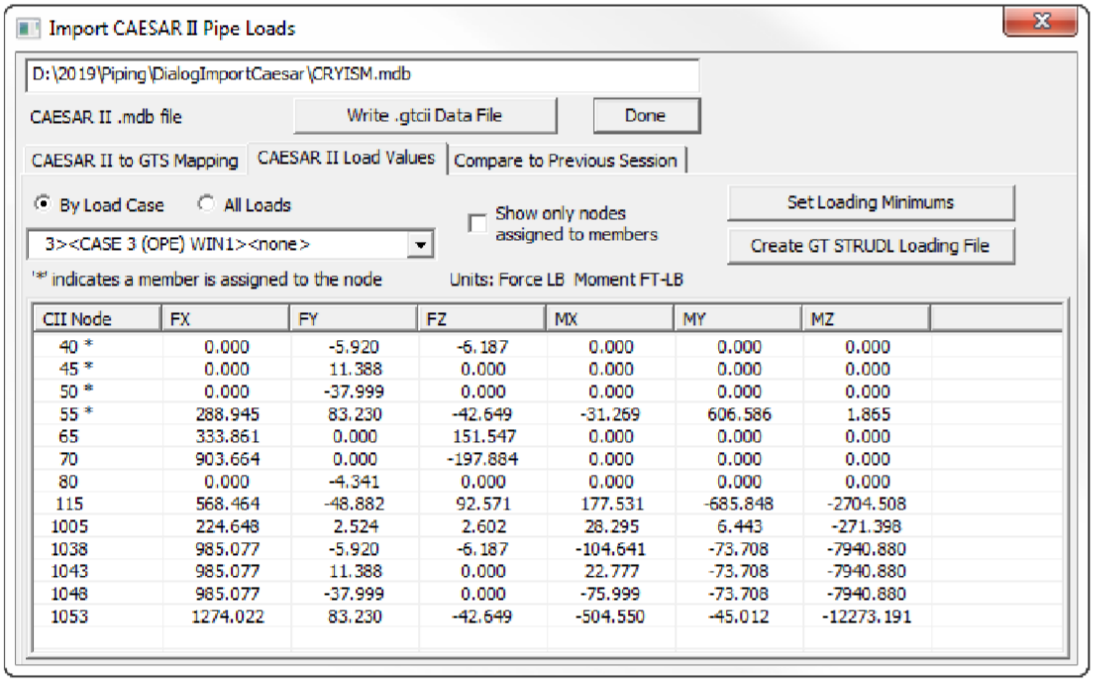
This tab is used to assign the CAESAR II nodes to a location along a GT STRUDL member. Only reactions for nodes with a 'GTS member' assignment can be exported to a loading file. You can type member names and fractional distance along the local X axis into the grid by double-clicking the cell. Press Enter 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



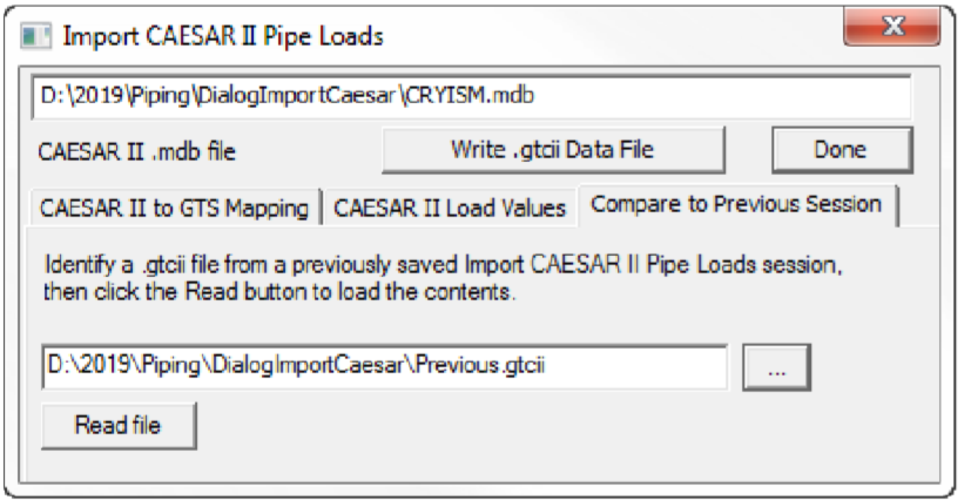
Reaction values can be displayed with only one load case in the grid, or all loads in the grid. A dropdown menu is provided to shift between load cases when in single load case mode. Only nodes with assigned GT STRUDL members, those with an entry in the “GTS member” column, can be exported into a loading 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.

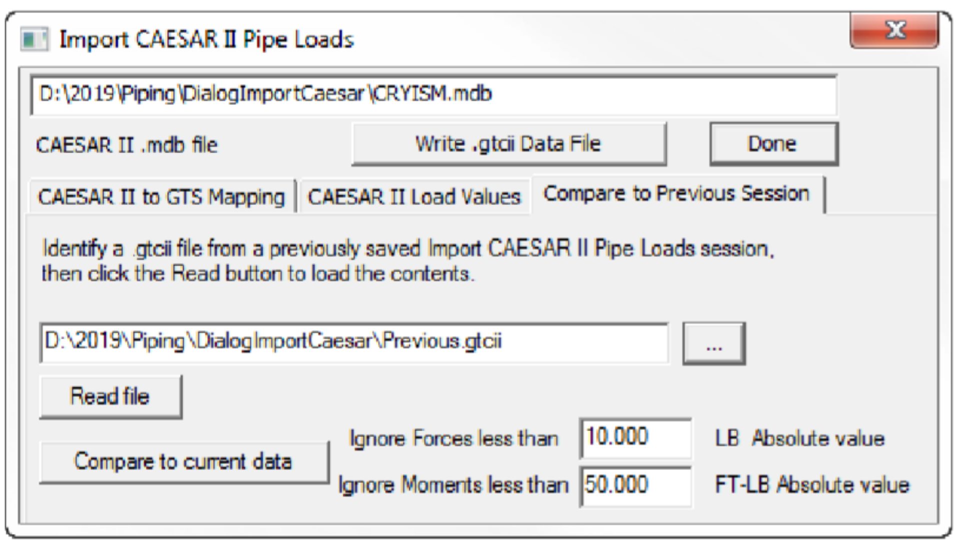
If you want to be able to compare this load data with a subsequent CAESAR II pipe stress analysis, use the “Write .gtcii Data File” button to save the current data to used with the next tab.

The "Compare to Previous Session" Tab

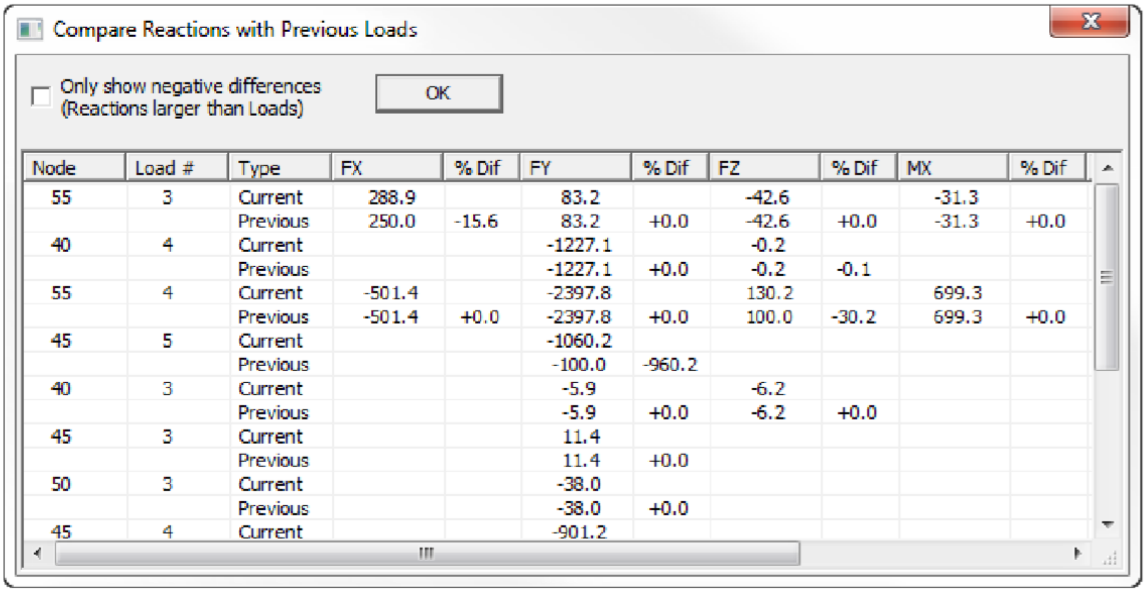
If you have saved a previous data file (.gtcii extension), you can use that to compare with a new CAESAR II analysis to verify that the new reaction values do not exceed the loading values sent to GT STRUDL. Enter or browse for the .gtcii file and then click the 'Read file' button.



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					