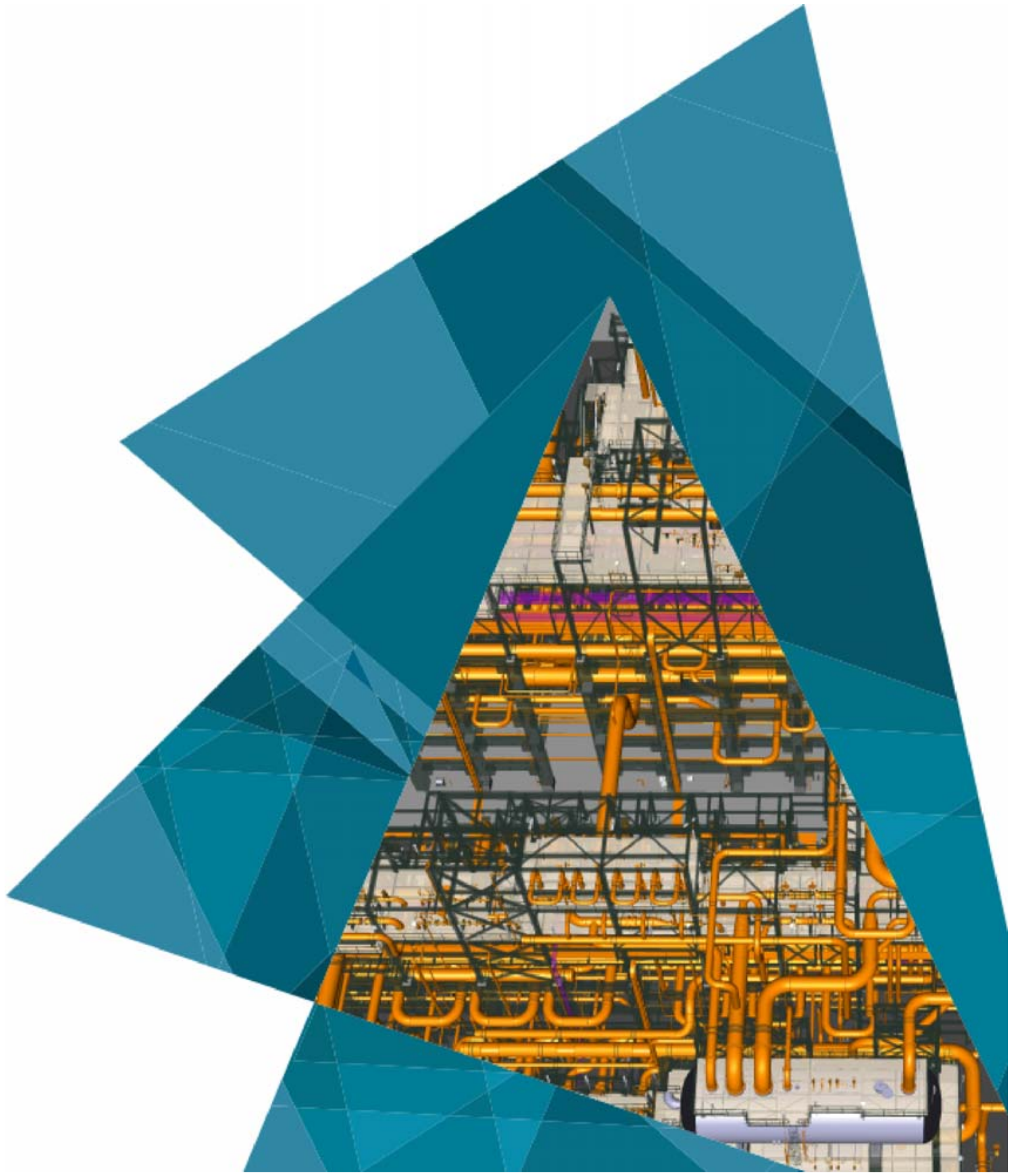
GT STRUDL® Version 2019 Release Guide



Release Date: April 2019

HEXAGON PPM

Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 2019 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

Copyright © 2019 Hexagon AB and/or its subsidiaries and affiliates. All rights reserved

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

U.S. Government Restricted Rights Legend

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- Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States.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

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

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

Other Documentation

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

Terms of Use

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

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

Disclaimer of Warranties

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

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

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

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

Limitation of Damages

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

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

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

Export Controls

Intergraph Corporation's commercial-off-the-shelf software products, customized software and/or third-party software, including any technical data related thereto ("Technical Data"), obtained from Intergraph Corporation, its subsidiaries or distributors, is subject to the export control laws and regulations of the United States of America. Diversion contrary to U.S. law is prohibited. To the extent prohibited by United States or other applicable laws, Intergraph Corporation software products, customized software, Technical Data, and/or third-party software, or any derivatives thereof, obtained from Intergraph Corporation, its subsidiaries or distributors must 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 (https://build.export.gov/main/ecr/eg_main_023148).

c. To any entity when Customer knows, or has reason to know, the end use of the software product, customized software, Technical Data and/or third-party software obtained from Intergraph Corporation, its subsidiaries or distributors 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.

Any questions regarding export/re-export of relevant Intergraph Corporation software product, customized software, Technical Data and/or third-party software obtained from Intergraph Corporation, its subsidiaries or distributors, should be addressed to PPM's Export Compliance Department, 305 Intergraph Way, Madison, Alabama 35758 USA or at exportcompliance@intergraph.com. Customer shall hold harmless and indemnify PPM and Hexagon Group Company for any causes of action, claims, costs, expenses and/or damages resulting to PPM or Hexagon Group Company from a breach by Customer.

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 4 Known Deficiencies

Chapter 5 Prerelease Features

This page intentionally left blank.

Chapter 1

Introduction

Version 2019 covers GT STRUDL operating on PC's under the Windows 10 and 7 operating systems. For users who are accustomed to our older version numbering system, the version is internally known as Version 38.0.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2018R1. Chapter 2 briefly describes an extensive list of new features of which a few are highlighted below:

- AISC 15th Steel Design code including ANSI/AISC 341-16 Seismic Provisions, ANSI/AISC N690-18 Nuclear Provisions and AISC 15th Edition steel tables.

- Numerous improvements to CAD Modeler including supporting BricsCAD version 19 and AutoCAD versions 2019 and 2020 and animation of the deformed structure and finite element stress contours.

- Numerous enhancements to GTMenu including the ability to now execute nonlinear analysis and eigenvalue analysis, specify steel design parameters and perform steel code checks and design all without leaving GTMenu.

- Importing piping loading information from CAESAR II.

Chapter 3 provides you with details regarding error corrections that have been made since the Version 2018R1 release. Chapter 4 describes known problems with Version 2019. Chapter 5 describes prerelease features -- new features which have been developed and subjected to limited testing, or features for which the user documentation has not been added to the GT STRUDL User Reference Manual. The command formats and functionality of the prerelease features may change before they become supported features based on additional testing and feedback from users.

The Prerelease features are subdivided into Design, Analysis, and General categories. The features in these categories and their section numbers in Chapter 5 are shown below:

5.2 Design Prerelease Features
 5.2.1 A new national annex parameter for EC3-2005 steel design code
 5.2.2 Design of Flat Plates Based on the Results of Finite Element Analysis

(The DESIGN SLAB Command)
5.2.3 ASCE4805 Steel Design Code. This code is for the ultimate strength design of steel transmission pole structures.

5.3 Analysis Prerelease Features
5.3.1 Calculate Error Estimate Command
5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

5.4 General Prerelease Features
5.4.1 Rotate Load Command
5.4.2 Reference Coordinate System Command
5.4.3 GTMenu Point Coordinates and Line Incidences Commands
5.4.4 GTMenu Surface Definition Command
5.4.5 Export model to CAESAR II
5.4.6 Import CAESAR II Pipe Loads

We encourage you to experiment with these prerelease features and provide us with suggestions to improve these features as well as other GT STRUDL capabilities.

Chapter 2

New Features in Version 2019

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 2019. This release guide is also available online upon execution of GT STRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 Steel Design

1. A new AISC 15th Edition (adopted on July 7, 2016) design code has been implemented as a released feature. Both LRFD (load and resistance factor design) and the ASD (allowable strength design) methods of the AISC Fifteenth Edition are implemented. This new code, AISC15, may be used to select or check any of the following shapes:

    Design for bi-axial bending and axial forces:

I shapes	Channel
Single Angles	Tees
Double Angles	Solid Round Bars
Solid Rectangular and Square Bars	

    Design for bi-axial bending, axial, and torsional forces:
    - Round HSS (Pipes)
    - Rectangular and Square HSS (Structural Tubes)

    The documentation for the AISC15 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and AISC15 in the GT STRUDL Output Windows.

2. Seismic provisions from ANSI/AISC 341–16 have been included in AISC15 code. The seismic provisions are applicable to all rolled sections documented in the steel design AISC15 User Reference Manual. Steel design parameters are documented in the Table 1.5-1 (Section 1.5) and the design provisions are documented in the Section 4.7 of the AISC15 User Reference Manual.

3. The ANSI/AISC N690–18, specification for safety-related steel structures for nuclear facilities has been included in AISC15 code. This code is based on the AISC steel construction manual, Fifteenth Edition specification with a few modifications. Both LRFD (load and resistance factor design) and the ASD (allowable strength design)

methods of the AISC Fifteenth Edition are applicable. Applicable cross-sections are I shapes, Channel, Single Angles, Tees, Double Angles, Round HSS (Pipes), Rectangular and Square HSS (Structural Tubes), Solid Round Bars, Solid Rectangular and Square Bars. Additional documentation is available in the Sections 1.6 and 4.8 of the AISC15 User Reference Manual. The documentation for the N690-18 or AISC15 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and AISC15 in the GT STRUDL Output Window.

4. The transmission tower provisions, slenderness ratio and computation of the number of bolts, are now implemented in AISC15 code. When a value of YES has been specified for parameter TowerCK, transmission tower provisions are checked in addition to the provisions of the AISC15 code. Applicable cross-sections are single and double angles. The documentation may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and AISC codes with Transmission Tower Provisions in the GT STRUDL Output Windows.

2.2 Steel Tables

1. A New User Guide “GT STRUDL Tables of Steel Profiles” is created to present information about major steel profile tables provided with GT STRUDL and stored on the subsystem data file. This guide provides steel section tables with a description of their contents and an explanation of the profile naming convention used in these tables. In addition to information about tables, steel profile labels are presented in the associated table lists.

2. New AISC steel tables have been added based on 15th Edition of the American Institute of Steel Construction. The following 18 new tables have been added:

W-AISC15	rolled W shapes
M/S/HP15	rolled M, S, and HP shapes
RecHSS15	rolled Rectangular and Square HSS
C-AISC15	rolled Channel shapes
WTAISC15	rolled Tee shapes
L-EQ-15	rolled Equal Leg Single Angle shapes
L-UN-15	rolled Unequal Leg Single Angle shapes
L-ALL-15	rolled Equal and Unequal Leg Single Angle shapes
2L-EQ-15	rolled Equal Legs Double Angle shapes
2L-LL-15	rolled Long Legs back-to-back Double Angle shapes
2L-SL-15	rolled Short Legs back-to-back Double Angle shapes
2L-ALL15	rolled Equal, Long, and Short Legs back-to-back Double Angle shapes

RndHSS15	rolled Round HSS
Pipes15	rolled Pipe shapes
WBEAM-15	rolled W shapes commonly used as beams
WCOL-15	rolled W shapes commonly used as columns
A85RCH15	rolled Rectangular and Square HSS(conform to ASTM A1085)
A85RDH15	rolled Round HSS(conform to ASTM A1085)

A summary of these tables and the profiles in the tables is available in the new GT STRUDL Tables of Steel Profiles User Reference Manual. You may also view the tables and profiles using the Profile Browser under the Steel Design pulldown or in GTMenu. The new tables and profiles are also available in CAD Modeler and the Model Wizard.

3. New metric AISC steel tables have been added based on 15th Edition of the American Institute of Steel Construction. The following 18 new tables have been added:

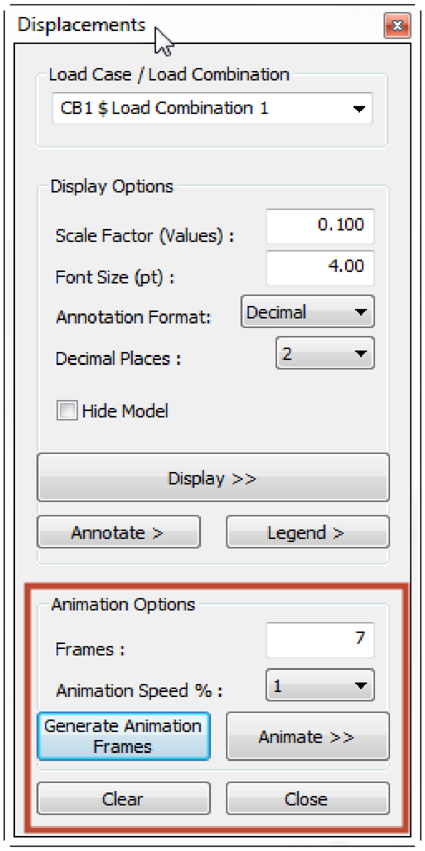
WAISC15M	rolled W shapes
MSHP-15M	rolled M, S, and HP shapes
ReHSS15M	rolled Rectangular and Square HSS
CAISC15M	rolled Channel shapes
WTAIS15M	rolled Tee shapes
L-EQ-15M	rolled Equal Leg Single Angle shapes
L-UN-15M	rolled Unequal Leg Single Angle shapes
L-ALL15M	rolled Equal and Unequal Leg Single Angle shapes
2L-EQ15M	rolled Equal Legs Double Angle shapes
2L-LL15M	rolled Long Legs back-to-back Double Angle shapes
2L-SL15M	rolled Short Legs back-to-back Double Angle shapes
2LALL15M	rolled Equal, Long, and Short Legs back-to-back Double Angle shapes
RdHSS15M	rolled Round HSS
Pipes15M	rolled Pipe shapes
WBEAM15M	rolled W shapes commonly used as beams
WCOL-15M	rolled W shapes commonly used as columns columns
A85RC15M	rolled Rectangular and Square HSS(conform to ASTM A1085)
A85RD15M	rolled Round HSS(conform to ASTM A1085)

A summary of these tables and the profiles in the tables is available in the new GT STRUDL Tables of Steel Profiles User Reference Manual. You may also view the tables and profiles using the Profile Browser under the Steel Design pulldown or in GTMenu. The new tables and profiles are also available in CAD Modeler and the Model Wizard.

2.3 CAD Modeler

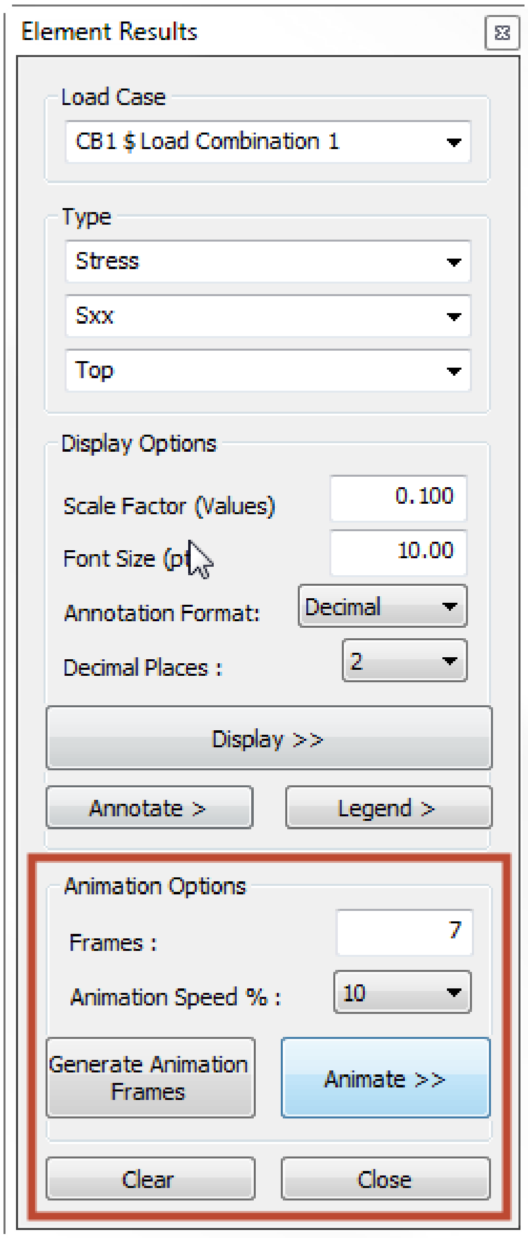
1. CAD Modeler now supports BricsCAD version 19 and AutoCAD versions 2019 and 2020.

2. You can now animate the deformed shape. By clicking on the Displacements option in the Display Results panel, the modified Displacements dialog shown below will appear with the new Animation Options panel highlighted:



In this dialog, you can select the number of animation Frames and animation Speed and then the Generate Animation Frames button. Once the animation frames are created, you can then click on the Animate button and the deformed shape will be animated.

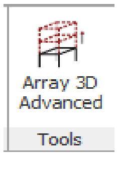
3. You can now also animate the finite element stress contours on the deformed shape of the model. The modified Element Results dialog is shown on the next page with the the Animation Options panel highlighted:



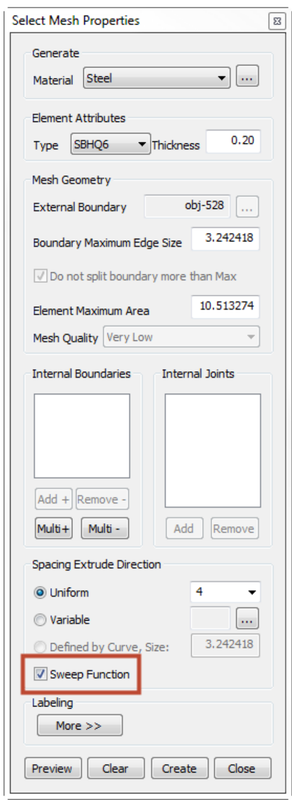
In this dialog, you can select the number of animation Frames and animation Speed and then the Generate Animation Frames button. Once the animation frames are created, you can then click on the Animate button and the stress contours on the deformed shape will be animated.

4. A new command called GTSARRAY3D has been implemented which has the same functionality as the CAD application ARRAY3D command but is up to 10 times faster when copying a model. This function is also available from the new Tools

panel on the GTS Modeling Ribbon Bar by selecting Array 3D Advanced as shown below:



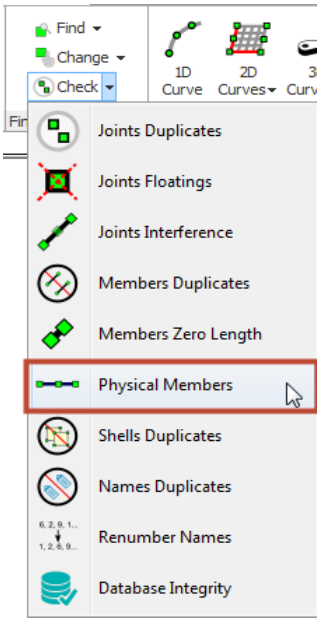
5. A Sweep option has been added to the 3D Extrude mesh generation option. This option will allow you to extrude along a curved polyline and perform a translation as well as a rotation during the extrusion. The modified Select Mesh Properties dialog is shown below with the Sweep option highlighted:



6. A new Check option has been added to Check Physical Members. This option will check any physical members in the model to be sure that all the analytical members in the physical member satisfy the following checks:

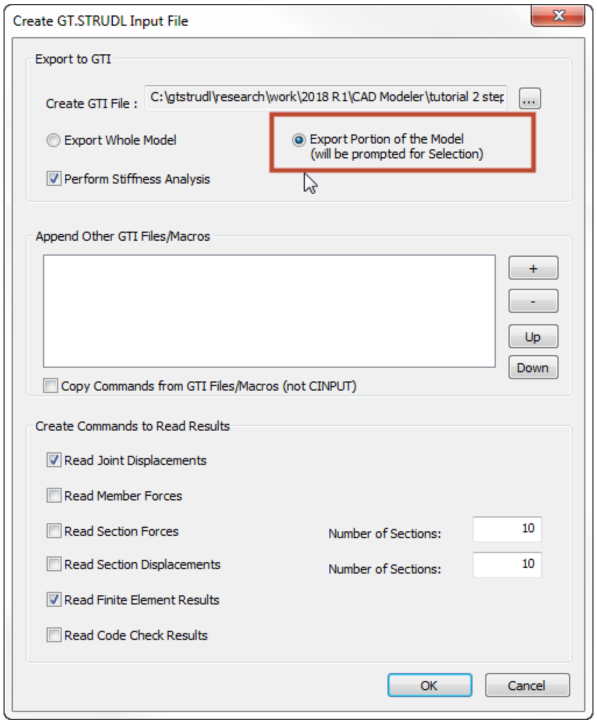
- Same Beta angle
- Same cross section
- No internal releases
- Sequential order of members within the physical member
- All members are in a straight line

The new Check pulldown is shown below with the new Physical Members check highlighted:



7. You are now able to use mixed units of length to specify levels in your model. For example, you can now specify a level's height as 20ft-5in or 20'-5". The Level Height is now highlighted in yellow in the Level Properties dialog to indicate that you can use mixed units of length.

8. When creating a gti file, you are now able to create the gti file for a portion of your model. By default, the whole model will be exported to the gti file. This new option has been added to the Create GT STRUDL Input File dialog as shown on the next page. If you have chosen to export only a portion of the model, you will be prompted to select the entities to export after clicking the OK button in the dialog. This option is very useful if you want to export only a portion of your model to CAESAR II for your piping analysis.



9. Selections chosen in the Read Results section in the above dialog will now be retained in the Read Results dialog.

10. When importing a gti file into CAD Modeler, a pop-up will appear now indicating the number of entities being imported (joints, members, elements, loads,...).

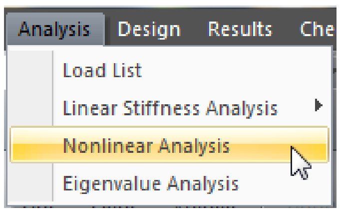
11. The Legend for results will now indicate the units for the results. An example of this for a finite element stress contour legend is shown below:

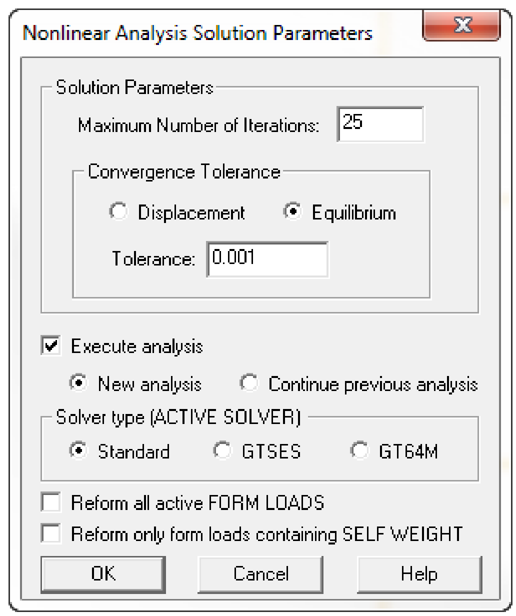
12. Element Loads and Average Element Results have been added to the available options in the Report Builder dialog. More information on Report Builder is presented in a Section 2.9.

2.4 GTMenu

1. The generated input file now contains ASSIGN GUID commands if Physical Member GUIDs exist in the data base.

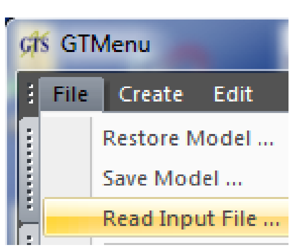
2. Nonlinear static analysis is now available under the Analysis menu. This will bring up the Nonlinear Analysis dialog where you can select the appropriate options as shown below. On selecting OK, nonlinear analysis will be initiated and the Output Monitor will appear.





At least one independent load must be active for Nonlinear Analysis. No dependent loads (LOAD COMBINATION) may be active for Nonlinear Analysis. If either of these requirements are not met, an error message box will be generated and Nonlinear Analysis will not proceed.

3. The File menu now includes a new entry ‘Read Input File...’. This feature allows you select and process a GT STRUDL input file, normally with a .gti extension.



This feature requires a commitment of any changes made while in GTMenu - you will be prompted so cancellation is possible. After the input commands have been processed, the GTMenu database is updated and a redraw is performed. Any new joints, members, elements, loads, results, etc. created by the input file will now be available in GTMenu.

Four commands are ignored when reading an input file from GTMenu: STRUDL, GTMENU, CINPUT STANDARD and RESTORE. You will see an message like this if one of these commands is found:

```
****INFO - STRUDL command ignored when reading from GTMenu.
```

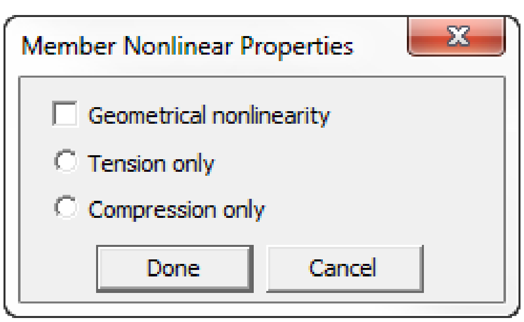
Note: CINPUT 'file_name' is acceptable, but CINPUT STANDARD (or just CINPUT with no option) expects input from the keyboard, and GTMenu does not have a Command Window for keyboard access like Command Mode.

After you select the file to be processed, the Output Monitor will open in modal status, meaning that no further GTMenu actions are allowed until the file is completely processed. When the Done button appears, the file has finished processing. Click the Done button when you are ready to return to interactive GTMenu. The redraw will occur after the Done button is clicked.

4. Nonlinear effects may now be reviewed and edited in GTMenu. When Inquire data is displayed for a member, a new "Nonlinear effects" entry has been added.

```
Member Name:     1            Length: 10.000
 Type:           SPACE FRAME
 Incidences -    Start:  1         End:  6
 No Eccentricities or End Joint Sizes
 No Member Releases
 Member Beta Angle:     0.0
 Member Section:    W14X53   TABLE WCOLUMN9 W14X53
 Member Material:   STEEL
 No Physical Member Group
 Nonlinear effects: Geometrical nonlinearity
```

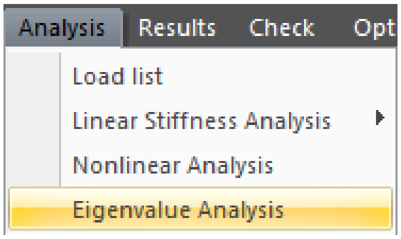
You can double-click the new entry to change the Nonlinear effects status with the Member Nonlinear Properties dialog:



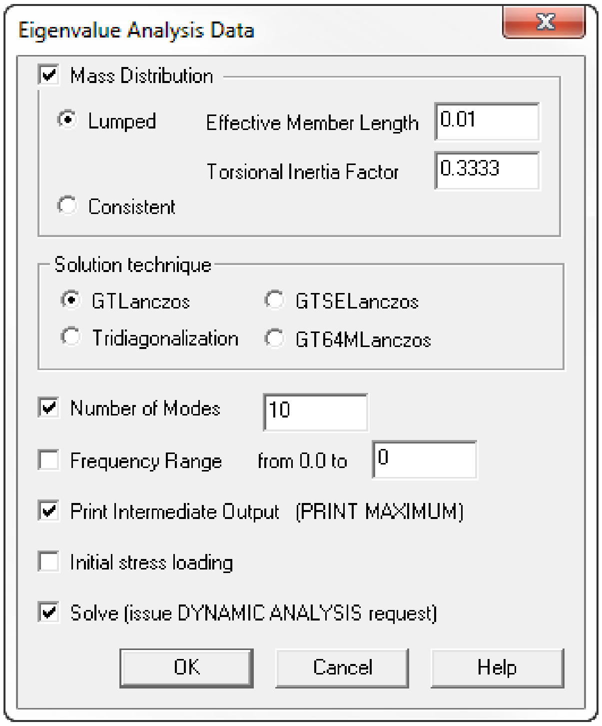
The Data Type panel in the Edit Member Data dialog also has a new button to edit Nonlinear effects:

<u>Documentation</u>:
"2.5.2.1 The NONLINEAR EFFECTS Command", Volume 3, GT STRUDL Reference Manual

5. Eigenvalue analysis (dynamic frequencies and mode shapes) has been added to the Analysis menu.



Selecting this option will open the Eigenvalue Analysis Data dialog shown on the next page, which is the same dialog available while in Text Mode. Note that any modeling changes made while in GTMenu will need to be committed before running a eigenvalue analysis.



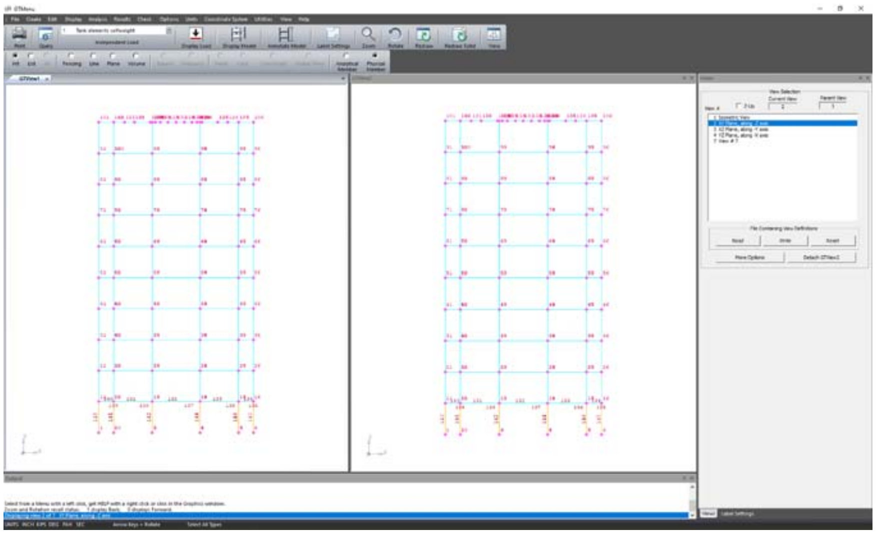
Click the Help button to get an overview of how to use the dialog.

<u>Documentation</u>:
"2.4.3.1 Inertia Specification Command", Volume 3, GT STRUDL Reference Manual
"2.4.5.2 Eigenproblem Solution Specifications", Volume 3, GT STRUDL Reference Manual
"2.4.5.4 The DYNAMIC ANALYSIS Command", Volume 3, GT STRUDL Reference Manual

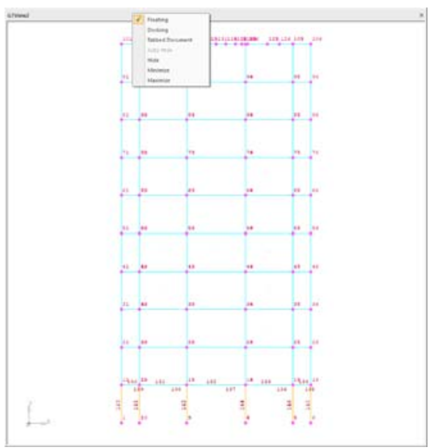
6. Detachable Second View (GTView2): GTMenu now allows the detachment of the second view to be moved to a second monitor.

By default, GTView2 starts attached to the main window into the working area. The user can use the contextual menu by right clicking on the tab to detach the view from the Tabbed control.

Once the view has been detached from the Tabbed control, the view behaves as a Dockable Pane, and can be docked at any available place in the Main Window.

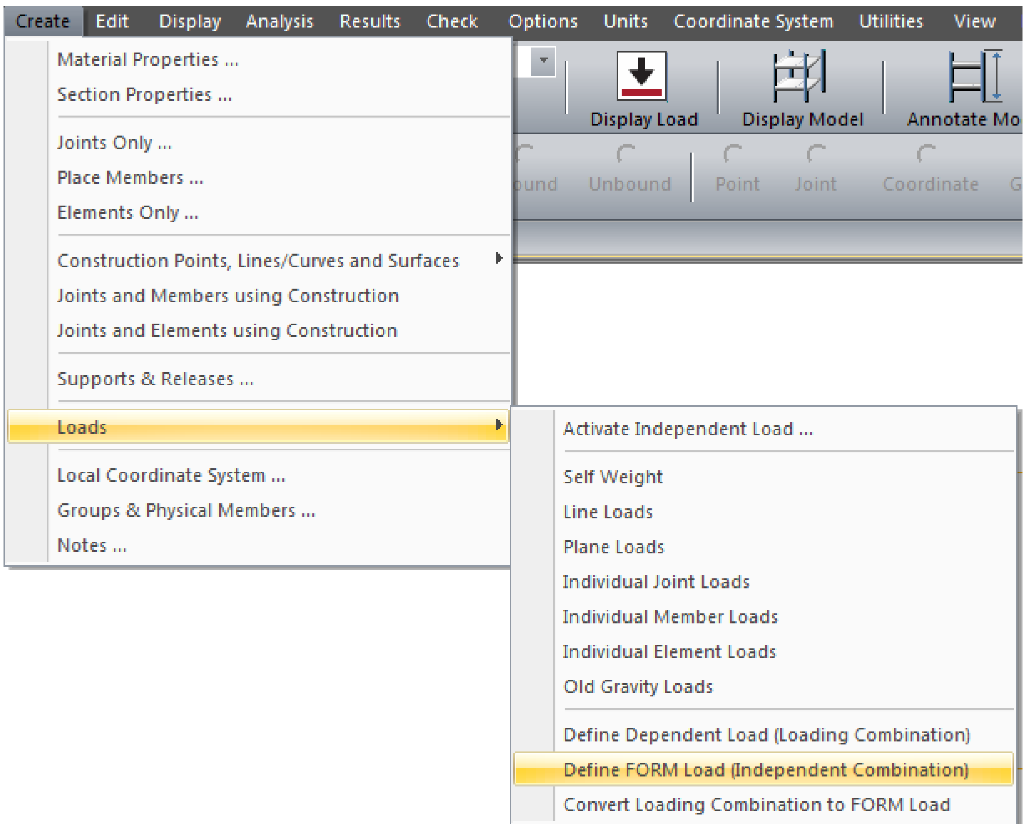


The view can also be set to float and moved to a second monitor. While in floating mode, the user will find two additional menu entries in the contextual menu. These can be used to Maximize/Minimize the floating view.



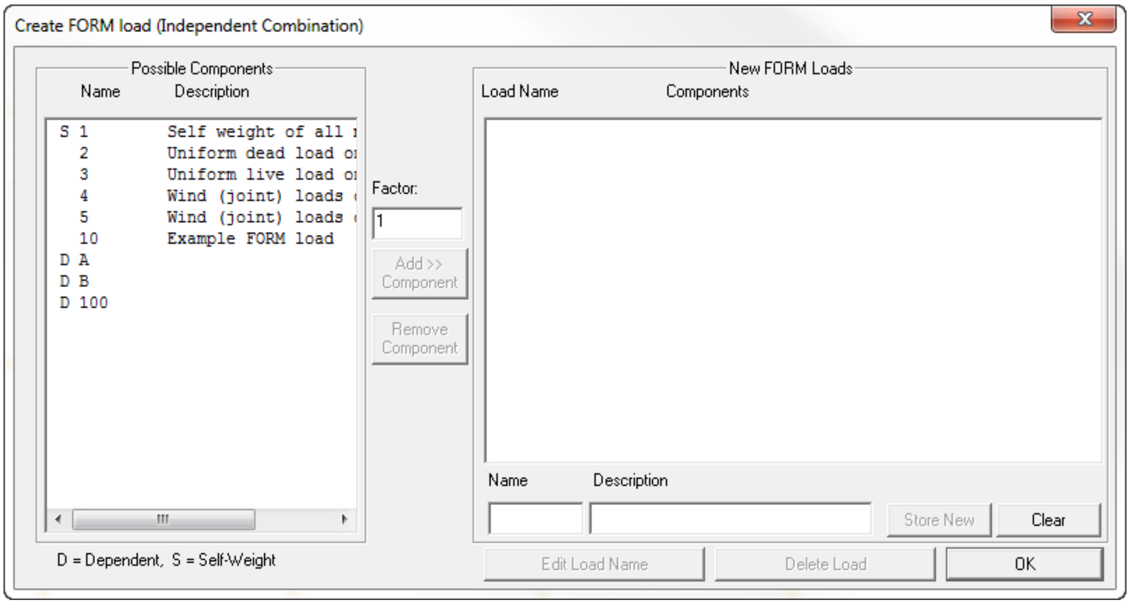
Additionally, when the view is minimized, a second GTMenu Icon is shown in the Windows Task Bar. Hovering the mouse over that icon will show a mini-window from which the user can show the view again.

7. A dialog to create FORM LOADS has been added to GTMenu. A FORM LOAD is an <u>independent</u> load created from a linear combination of specified load cases and factors, much as a Load Combination is specified, except the applied load values (joint, member, element, etc.) are factored and combined to create a new independent load case. A FORM LOAD is the only type of loading combination allowed for Nonlinear Analysis.

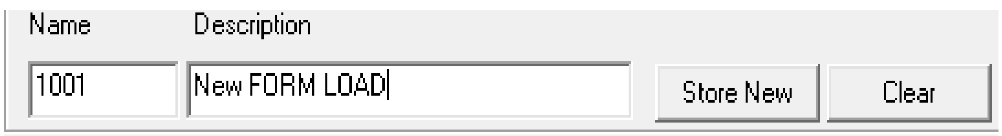


<u>Documentation</u>:
"2.1.11.3.2 Independent FORM LOADING Command", Volume 1, GTSTRUDL Reference Manual.
"2.1.11.5 Dependent Loading Identification", Volume 1, GTSTRUDL Reference Manual. (Loading Combination)

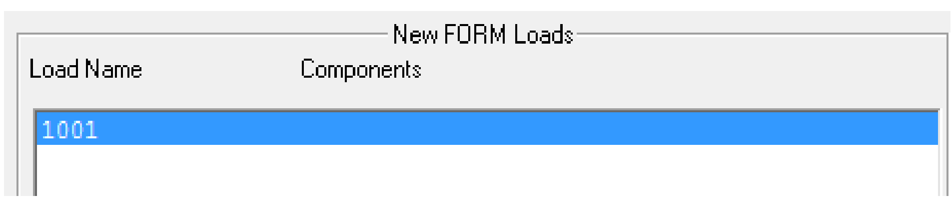
The new dialog has the same layout as the Load Combination dialog, with a list of possible components and list of the new FORM loads to be created when the OK button is clicked.



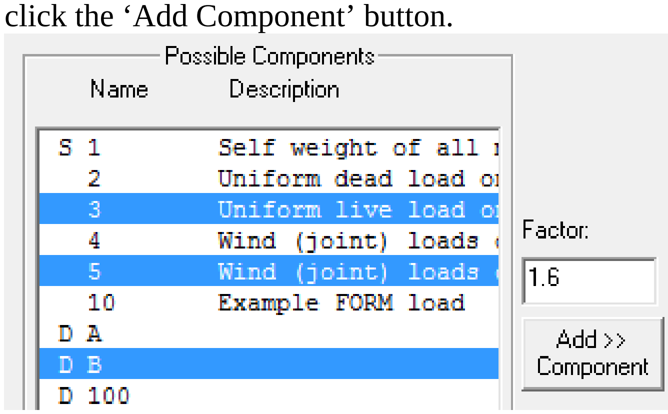
To create a FORM LOAD, enter a load name and optional description and then click the ‘Store New’ button.



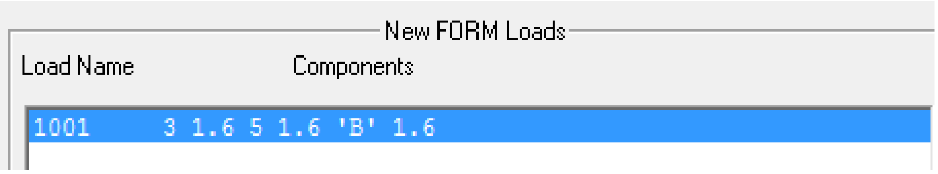
The load will now appear in the New FORM Loads list and be selected.



Select one or more loads from the ‘Possible Components’ list, enter a Factor and then click the ‘Add Component’ button.

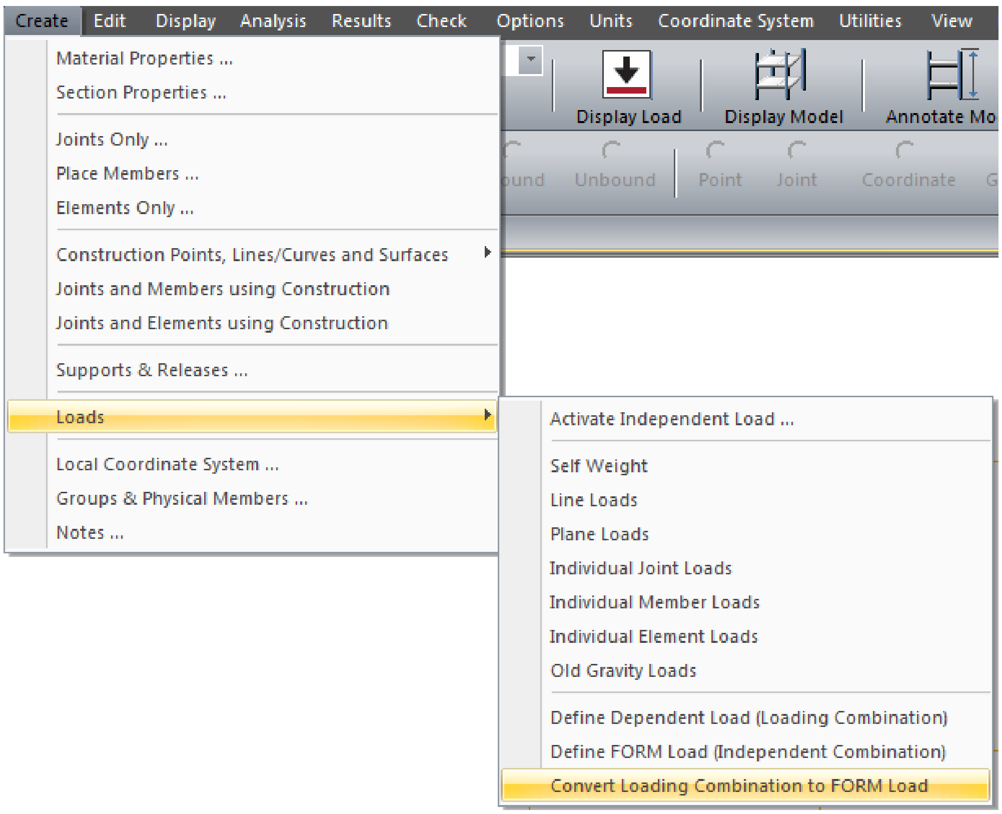


The selected components and factor will be added to the Form Load definition. Continue adding component/factor pairs until the Form Load definition is complete. If you add an incorrect component to the Form Load, select the component name in the 'Possible Components' window and click the 'Remove Component' button.

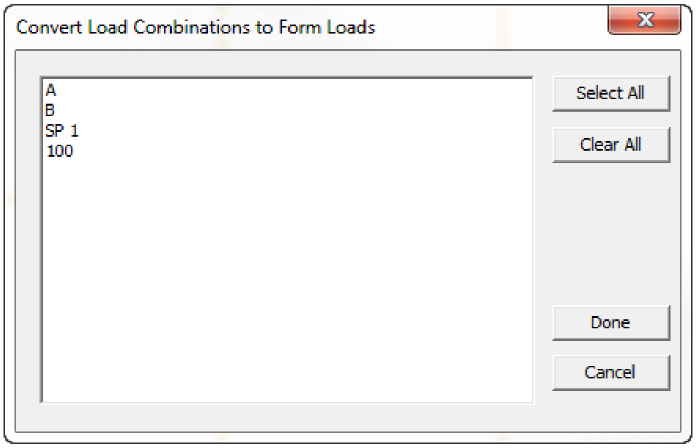


When all Form Loads have been completed, click the OK button to create the new loadings. This action requires a commitment of any modeling changes to continue. The Output Monitor will appear - click the 'Done' button to close the Output Monitor.

8. A dialog to convert Load Combinations (dependent loads) to Form Loads (independent loads) has been added to GTMenu.

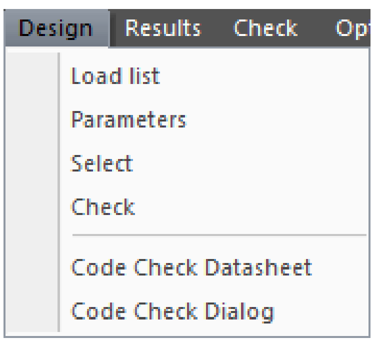


Select from the current Loading Combinations in the 'Convert Load Combinations to Form Loads' dialog using multi-select and click the 'Done' button to convert. This action requires a commitment of any modeling changes to continue. The Output Monitor will appear - click the 'Done' button to close the Output Monitor.



9. Element Loads and Average Element Results have been added to the available Report Builder options. Enhancements to Report Builder are discussed in a later section.

10. A new Design menu has been added to GTMenu, as shown below:



Load list is the same active load dialog as under the Analysis menu, which allows you to limit the number of load results to be used for Select or Check.

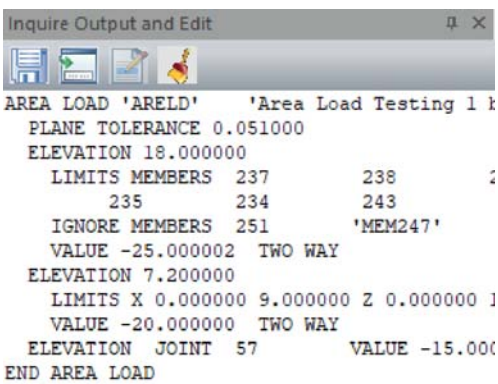
Parameters is a modal dialog that sets steel design criteria, such as the design standard like "AISC 15th Edition". Member selection is currently limited to List mode. At a minimum, the design standard must be set before Select or Check is used.

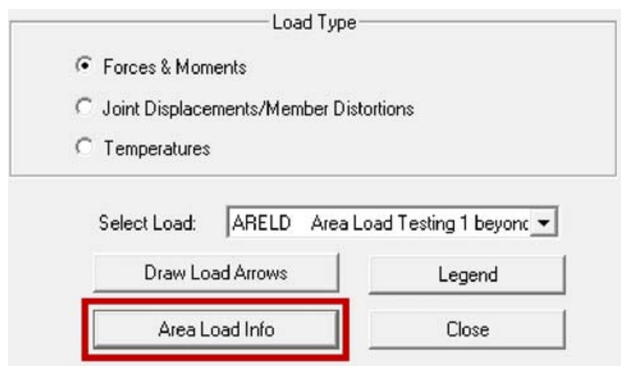
Select is a non-modal 'Pane' dialog that by default appears on the right with the View and Label Settings dialogs. All selection modes are available for this dialog. Select is used to find the most efficient profile to support the active loadings. Select will update the current member properties to the selected profile.

Check is a non-modal ‘Pane’ dialog that by default appears on the right with the View and Label Settings dialogs. All selection modes are available for this dialog. Check will compare the current profile to the specified design standard for the active loadings. No changes to the member are made.

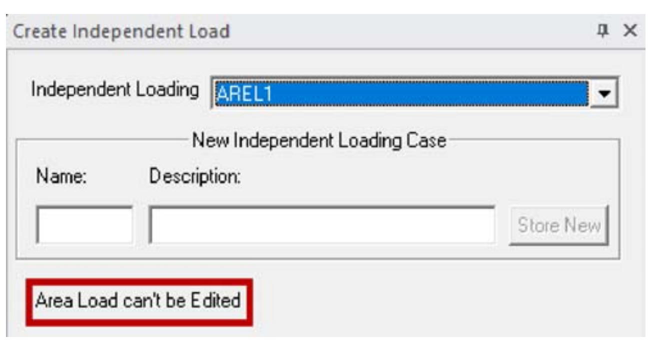
Code Check Datasheet and Code Check Dialog are the same functions as under the Results menu. They are only active if Select or Check has been used.

11. The Area Load definition may now be reviewed in GTMenu. The *Area Load Info* button has been added to the Display Load dialog and the button is activated upon the selection of an Area Load. Use the *Area Load Info* button to display the Area Load definition in the Inquire Output and Edit dialog.

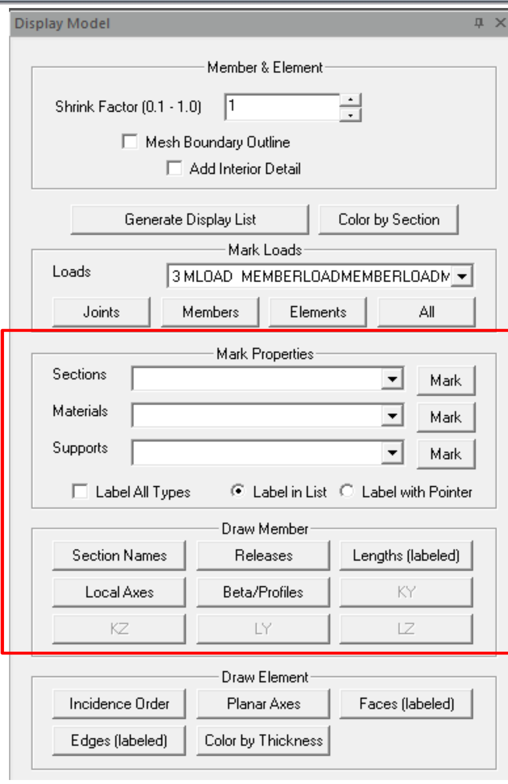


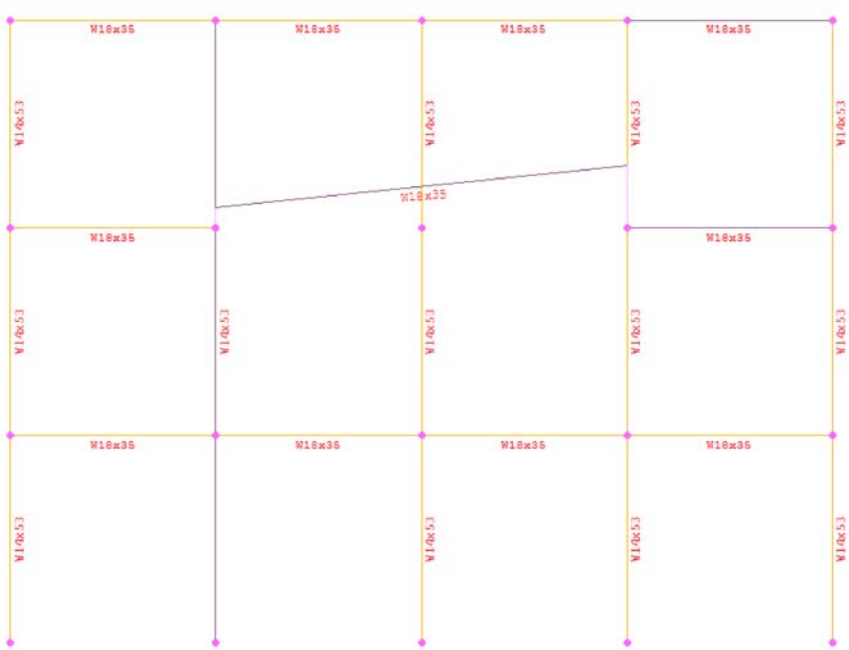


An important change related to the Area Loads in GTMenu is that the Create/Edit Independent Load dialog identifies the defined Area Loads and doesn’t allow you to modify the Area Loads. Upon the selection of an Area Load, no other loads can be added to the selected Area Load, similar to a Self-Weight loading.

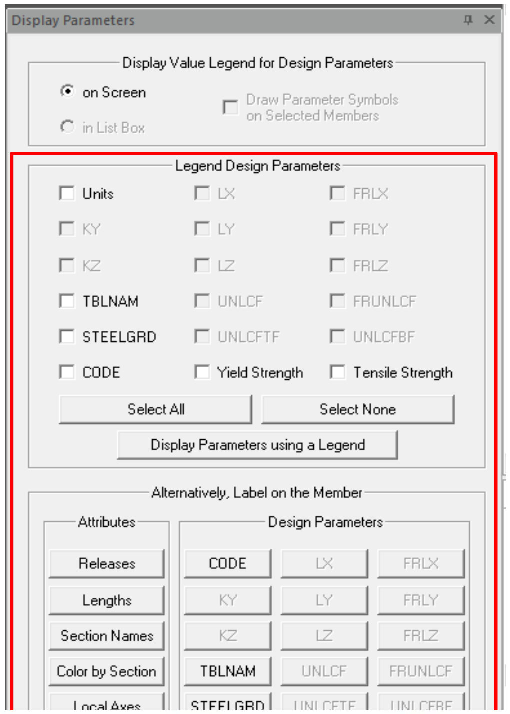


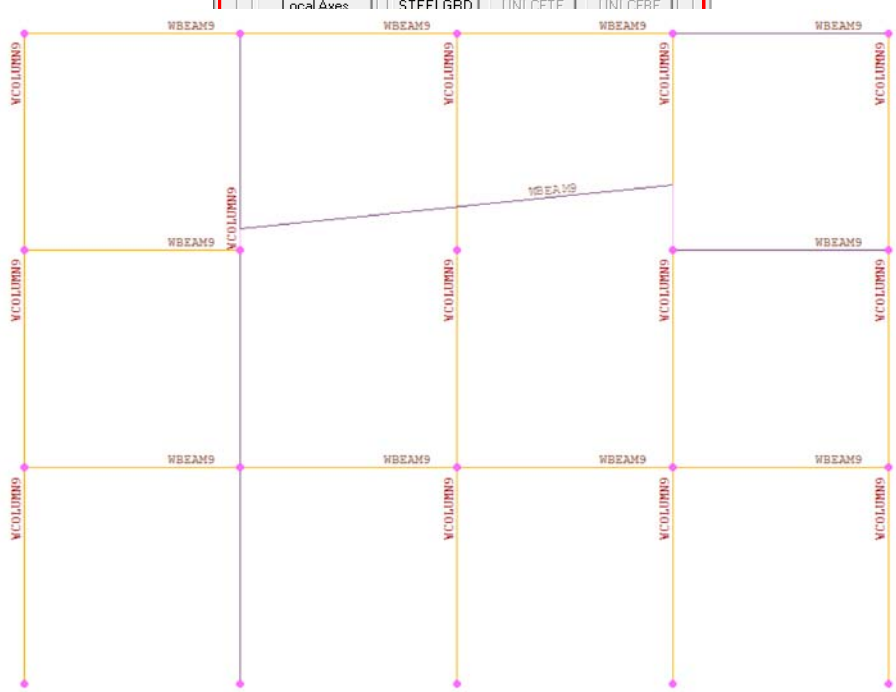
12. You are now able to mark member properties and draw member information on physical members from the Display Model dialog.





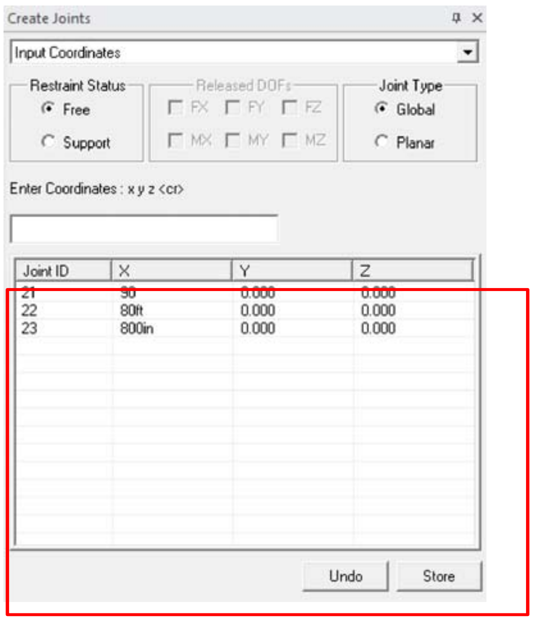
13. You are now able to display and label design parameters on physical members from the Display Parameters dialog.

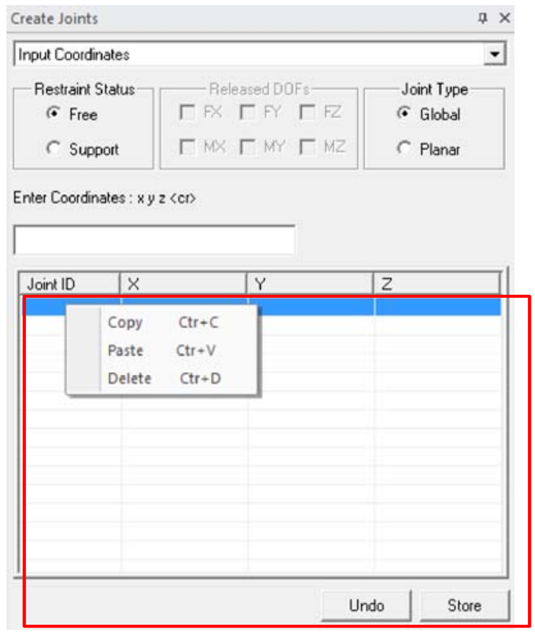




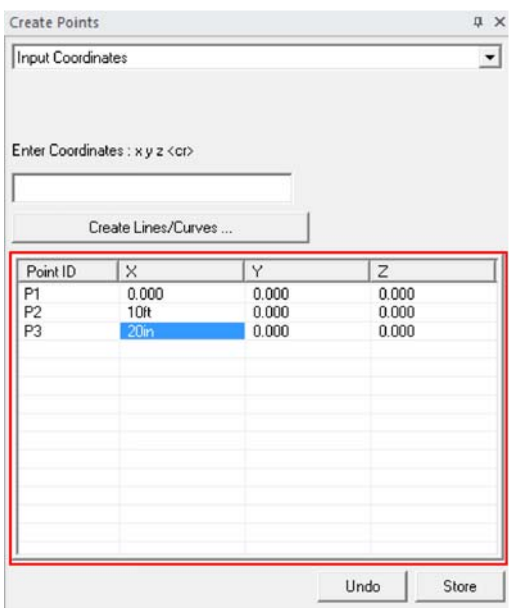
14. You are now able to create, edit, copy, and paste joint names(ID's) and coordinates from the Grid Sheet in the Create Joints dialog. After editing joint names and

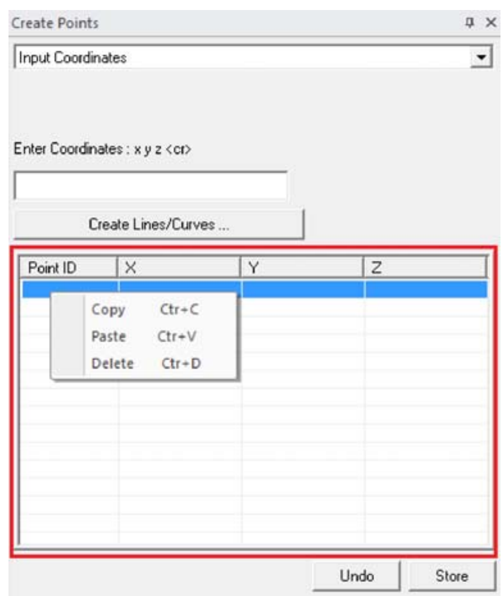
coordinates, the graphical display will automatically be updated to reflect new joint names and joint locations. In addition, SI and US length units are available in the coordinate grid cells.



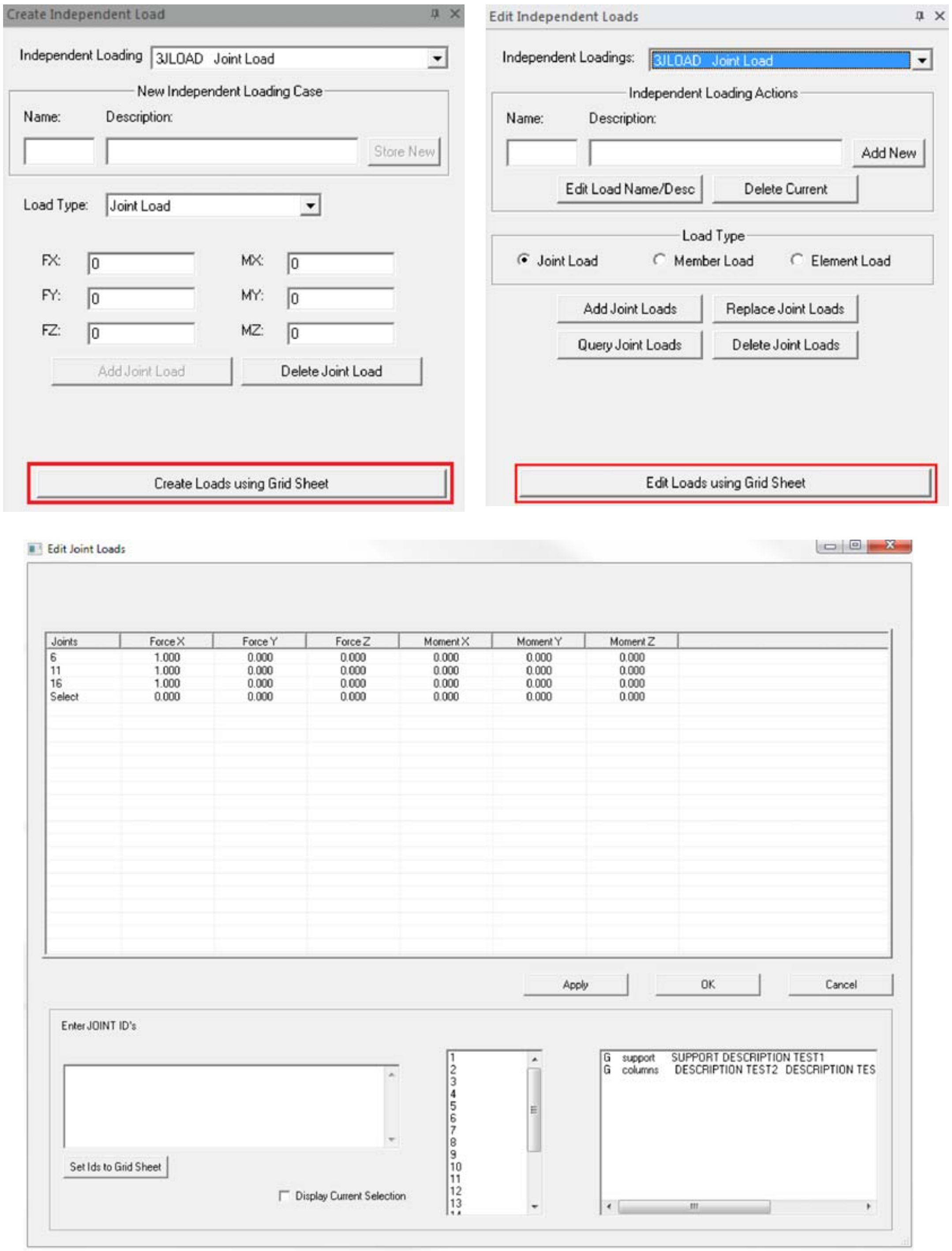


15. You are now able to edit, copy, and paste construction point names(ID's) and coordinates from the Grid Sheet in the Create Points dialog. After editing point names and coordinates, the graphical display will automatically be updated to reflect new point names and point locations.

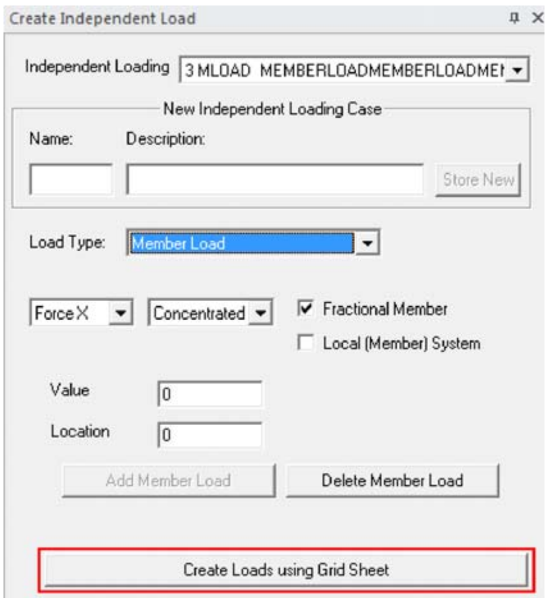


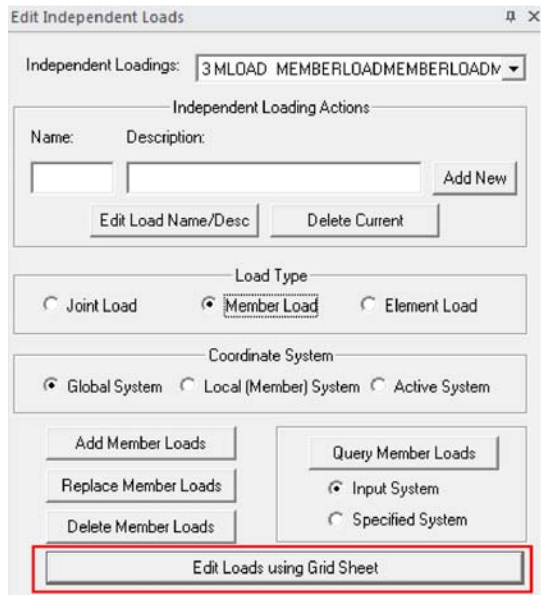


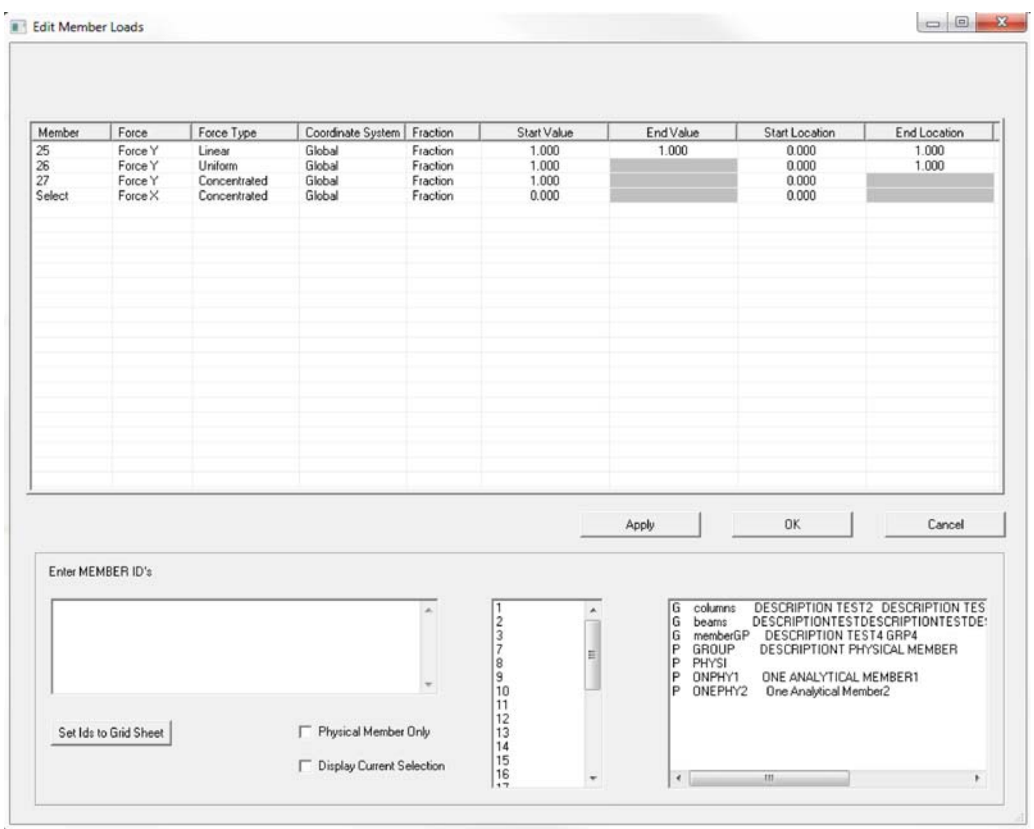
16. You are now able to edit, copy, and paste joint loads from the Grid Sheet in the Create Loads dialog and Edit Loads dialog. After creating or editing joint loads, the graphical display will automatically be updated to reflect joint loads by clicking the Apply button.



17. You are now able to edit, copy, and paste member loads from the Grid Sheet in the Create Loads dialog and Edit Loads dialog. After creating or editing member loads, the graphical display will automatically be updated to reflect member loads by clicking the Apply button.

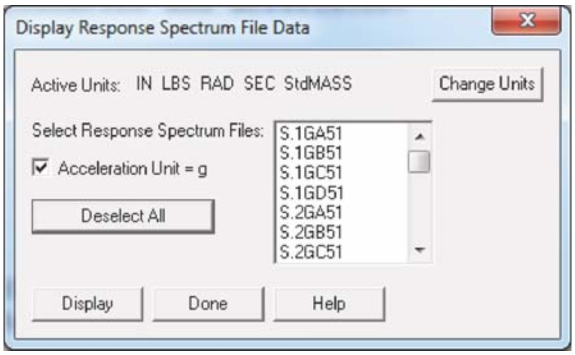




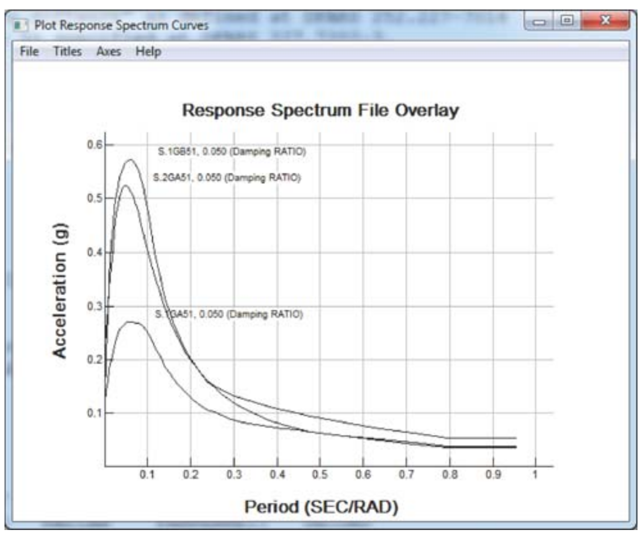


2.5 GT STRUDL Output Window (GTShell)

1. The *Display Response Spectrum File Data* dialog *(Results->Dynamic Analysis Results->Display Response Spectrum File Data*) has been enhanced to allow for the selection of multiple response spectrum files:



When multiple files are selected, the *Display* button produces an overlay plot of the data from all of the selected files:



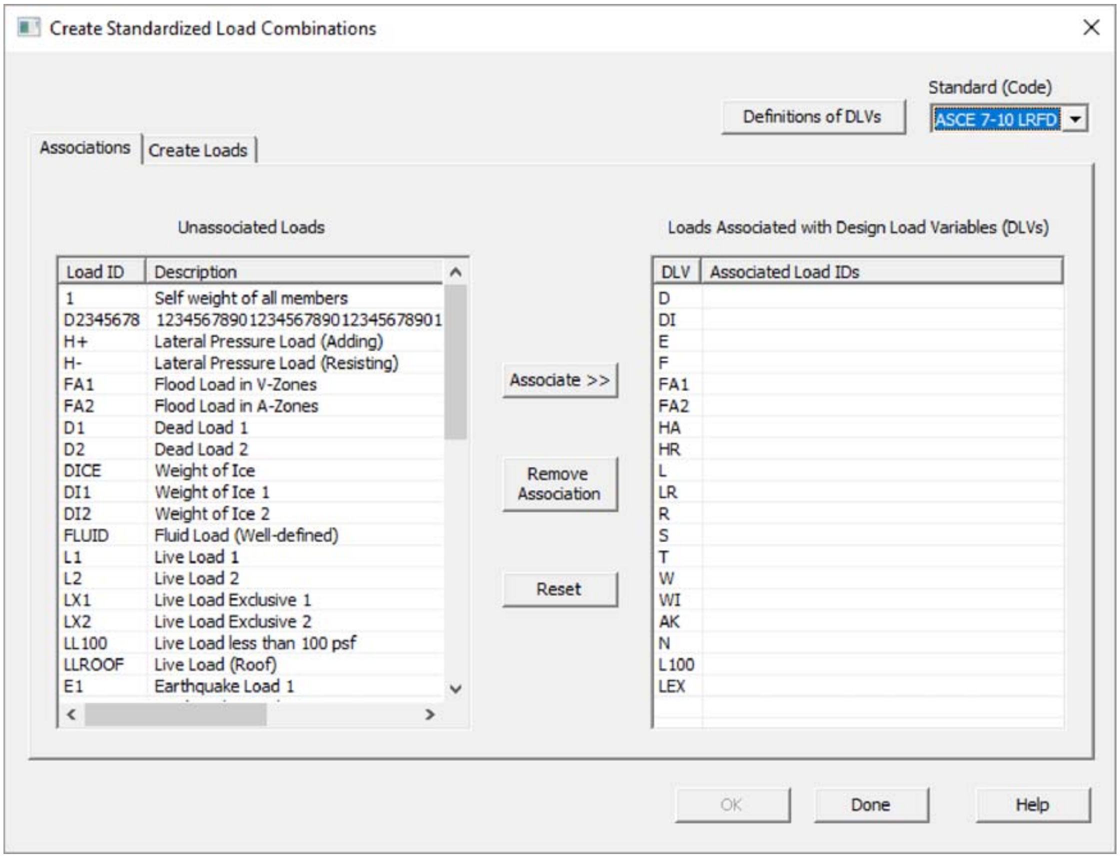
Note that the response spectrum curves from all of the selected files must have identical properties with respect to damping ratio, log or linear axis scaling, and horizontal axis units of frequency or period.

The dialog also includes a *Deselect All* button to conveniently clear all file selections from the *Select Response Spectrum Files* list box.

2. The Create Standardized Load Combinations dialog has been added to the GTShell to automatically create load combinations based on the combinations of the design

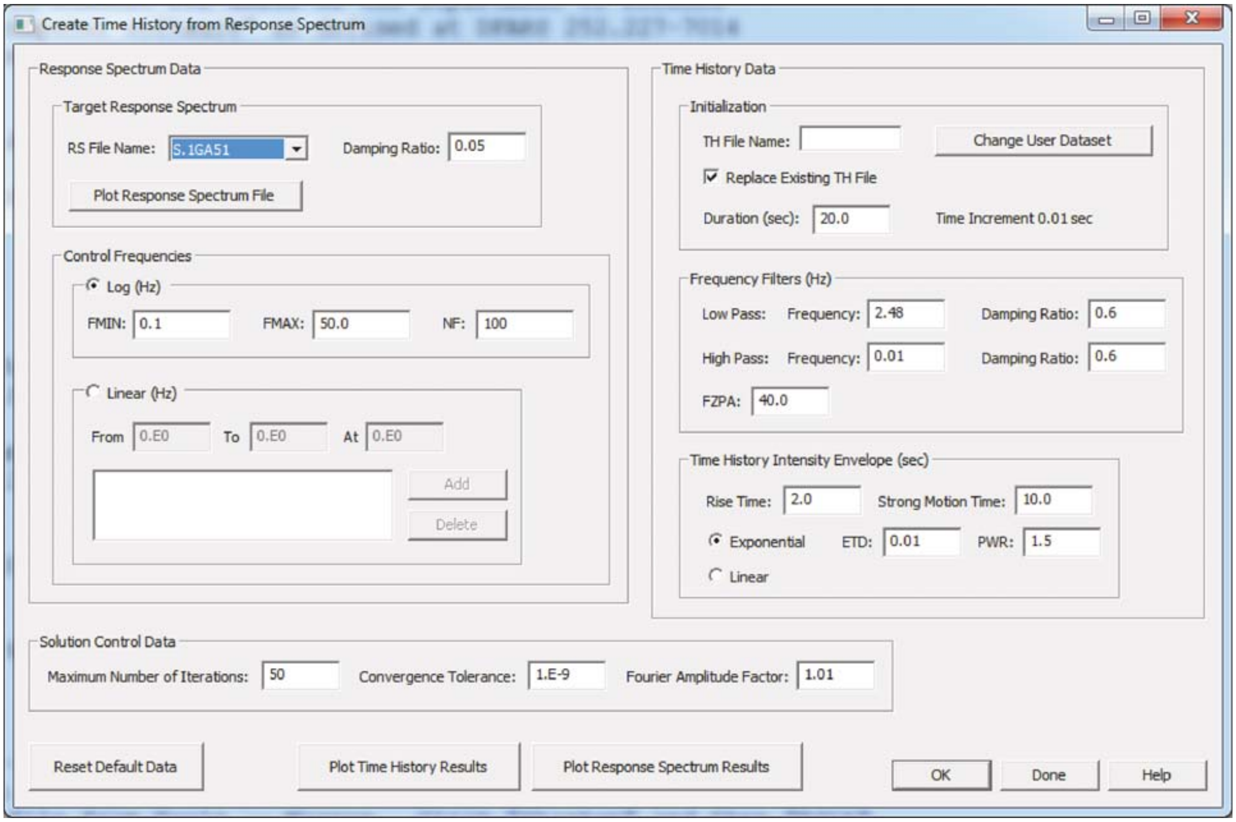
standards (codes) ASCE 7-10 and AISC 14 for both provisions of the strength design (LRFD) and the allowable stress design (ASD).

Because the creation of standardized load combinations can be done in two-command steps on the GTSolver (Command#1: STORE DESIGN LOAD VARIABLES; Command#2: CREATE AUTOMATIC LOAD COMBINATION DESIGN), the GTShell dialog interface has been designed to follow the similar steps with two tabs: “Associations” and “Create Loads”.



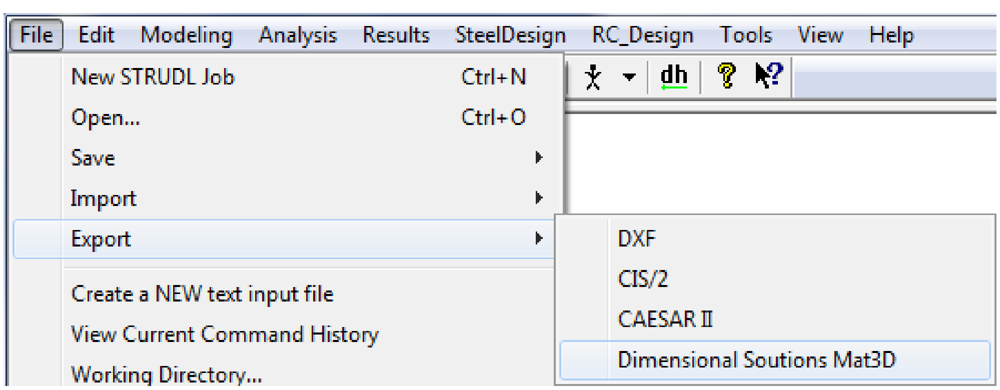
In the Associations tab, make associations between the defined loads and the Design Load Variables which are defined in the standard ASCE 7-10 Chapter 2. Combinations of Loads, as indicated by the standard (code) name highlighted in the “Standard (Code)” window at the upper right-hand corner of the dialog. Then move on to the Create Loads tab to select combinations and load creation options for creating automatic load combinations or form loads.

3. The Create Time History from Response Spectrum dialog has been added to facilitate the execution of the CREATE TIME HISTORY FROM RESPONSE SPECTRUM command presented later in Section 2.6, which creates a response-spectrum-compatible ground acceleration time history file from a specified target response spectrum file.

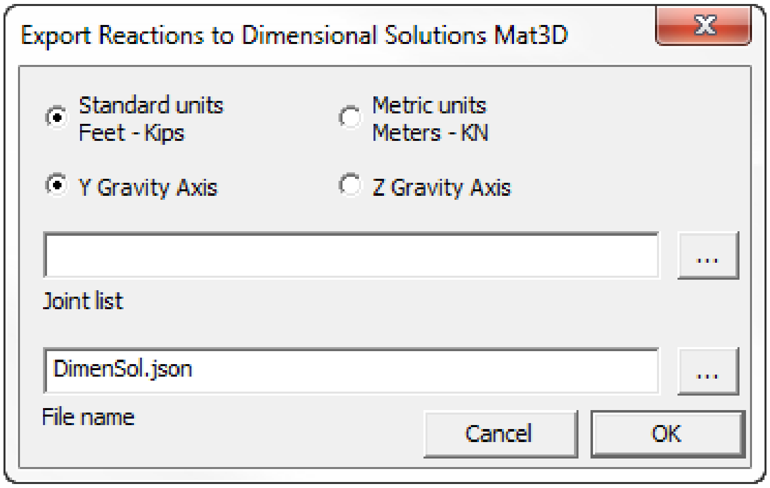


4. Element Loads and Average Element Results have been added to the available options in the Report Builder dialog. More information on Report Builder is presented in Section 2.9.

5. An option to write reactions to the foundation and footing design program Mat3D from Dimensional Solutions has been added to the File –> Export menu:



Use the "Export Reactions to Dimensional Solutions Mat3D" dialog to select units and the gravity axis. Enter a joint list, or use the Browse button [...] to bring up the joint selection dialog. Enter or browse for a file name.



When you click the OK button, reactions for the specified joints and the active load cases will be written to the specified .json file. This file can then be imported by MAT3D 2018 for foundation design.

2.6 Dynamic Analysis

A new CREATE TIME HISTORY FILE FROM RESPONSE SPECTRUM command has been added as follows:

CREATE TIME (HISTORY) FILE 'filename$_{TH}$' (FROM) RESPONSE (SPECTRUM)
Target Response Spectrum Specs
Time History Specs
Solution Control Specs
END (OF) (TIME) (HISTORY) (CREATION)

where

Target Response Spectrum Specs:

TARGET RESPONSE (SPECTRUM) (FILE) 'filename$_{TRS}$' DAMPING (RATIO) v_β

$$\text{(CONTROL (FREQUENCIES)} \left\{ \begin{array}{l} \to \text{LOG [FMIN]} \begin{Bmatrix} \to 0.1 \\ f_{min} \end{Bmatrix} \text{[FMAX]} \begin{Bmatrix} \to 50.0 \\ f_{max} \end{Bmatrix} \text{[NF]} \begin{Bmatrix} \to 100 \\ n_f \end{Bmatrix} \\ \text{LINEAR [FROM] } f_{F1} \text{ [TO] } f_{T1} \text{ [AT] } \Delta f_{A1} \cdots - \\ \qquad \text{[FROM] } f_{Fi} \text{ [TO] } f_{Ti} \text{ [AT] } \Delta f_{Ai} \end{array} \right\} \text{)}$$

Time History Specs:

$$(\underline{\text{TIME}}\ (\underline{\text{HIS}}\text{TORY})\ \underline{\text{DUR}}\text{ATION}\ \left\{\begin{matrix}\to 20.0\\ t_{DUR}\end{matrix}\right\}\ \underline{\text{FIL}}\text{TERS}\ \underline{\text{LOW}}\ [\underline{\text{FL}}]\ \left\{\begin{matrix}\to 2.48\\ f_{L}\end{matrix}\right\}\ [\underline{\text{BL}}]\ \left\{\begin{matrix}\to 0.6\\ \beta_{L}\end{matrix}\right\}\ -$$

$$\underline{\text{HIGH}}\ [\underline{\text{FH}}]\ \left\{\begin{matrix}\to 0.01\\ f_{H}\end{matrix}\right\}\ [\underline{\text{BH}}]\ \left\{\begin{matrix}\to 0.6\\ \beta_{H}\end{matrix}\right\}\ ([\underline{\text{FZPA}}]\ \left\{\begin{matrix}\to 40.0\\ f_{ZPA}\end{matrix}\right\}))$$

$$(\underline{\text{ENV}}\text{ELOPE}\ (\underline{\text{FUN}}\text{CTION})\ \left\{\begin{matrix}\to \underline{\text{EXPO}}\text{NENTIAL}\ [\underline{\text{TR}}]\ \left\{\begin{matrix}\to 2.0\\ t_{R}\end{matrix}\right\}\ [\underline{\text{TM}}]\ \left\{\begin{matrix}\to 10.0\\ t_{M}\end{matrix}\right\}\ -\\ ([\underline{\text{ETD}}]\ \left\{\begin{matrix}\to 0.01\\ v_{ETD}\end{matrix}\right\})\ ([\underline{\text{PWR}}]\ \left\{\begin{matrix}\to 1.5\\ v_{PWR}\end{matrix}\right\})\\ \underline{\text{LIN}}\text{EAR}\ [\underline{\text{TR}}]\ \left\{\begin{matrix}\to 2.0\\ t_{R}\end{matrix}\right\}\ [\underline{\text{TM}}]\ \left\{\begin{matrix}\to 10.0\\ t_{M}\end{matrix}\right\}\end{matrix}\right\})$$

Solution Control Specs:

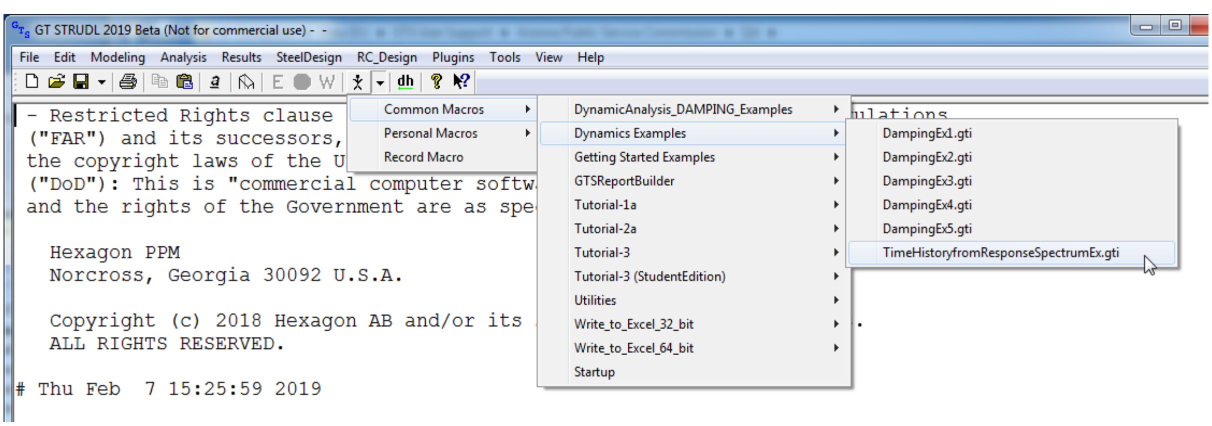
$$(\underline{\text{MAXI}}\text{MUM}\ (\underline{\text{NUM}}\text{BER})\ (\underline{\text{OF}})\ \underline{\text{ITER}}\text{ATIONS}\ \left\{\begin{matrix}\to 50\\ n_{iter}\end{matrix}\right\})$$

$$(\underline{\text{CON}}\text{VERGENCE}\ (\underline{\text{TOL}}\text{ERANCE})\ \left\{\begin{matrix}\to 1.E\text{-}9\\ v_{CTOL}\end{matrix}\right\})$$

$$(\underline{\text{FOU}}\text{RIER}\ (\underline{\text{AMP}}\text{LITUDE})\ (\underline{\text{FAC}}\text{TOR})\ \left\{\begin{matrix}\to 1.01\\ f_{facf}\end{matrix}\right\})$$

The purpose of this command is to create a synthetic ground acceleration time history that is compatible to a specified target response spectrum. The created time history is suitable to be used in a GT STRUDL linear or nonlinear dynamic time history analysis for which a transient loading is defined by a support acceleration that makes reference to the created time history.

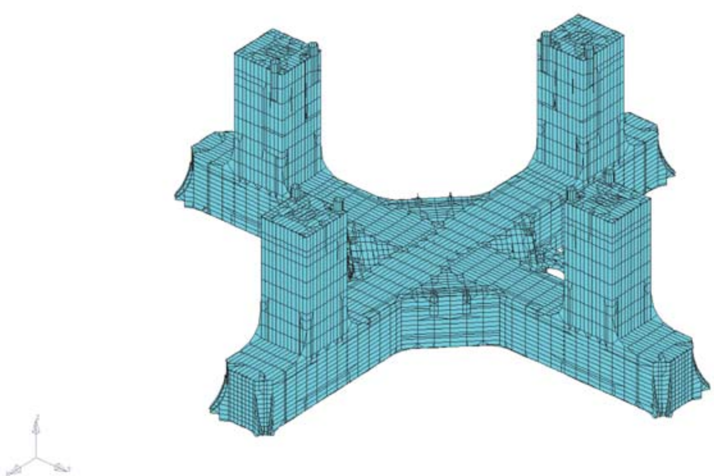
An example of this command is found in the standard installation macros path ***C:\Program Files (x86)\GTStrudl\Macros\Common\Dynamics Examples\TimeHistoryfromResponseSpectrumEx.gti***, which also can be accessed from the GTShell menu illustrated as shown below:



Documentation:
"2.4.8.2 Creation of a Time History File from a Response Spectrum File," Volume 3, GT STRUDL Reference Manual

2.7 GTSES/GT64M High-Performance Solvers

1. The GTSES and GT64M High-performance static analysis and dynamic eigenvalue analysis solvers, activated by the ACTIVE SOLVER command, have been enhanced to take advantage of scalable local and distributed multi-core processors such as the Intel® Xeon® family of processors. For example, the following figure shows a large model having 55,000 joints and 56,020 plate finite elements totalling 330,000 degrees of freedom, for which the GT64M Lanczos eigenvalue analysis solver was used to compute 50 modes.



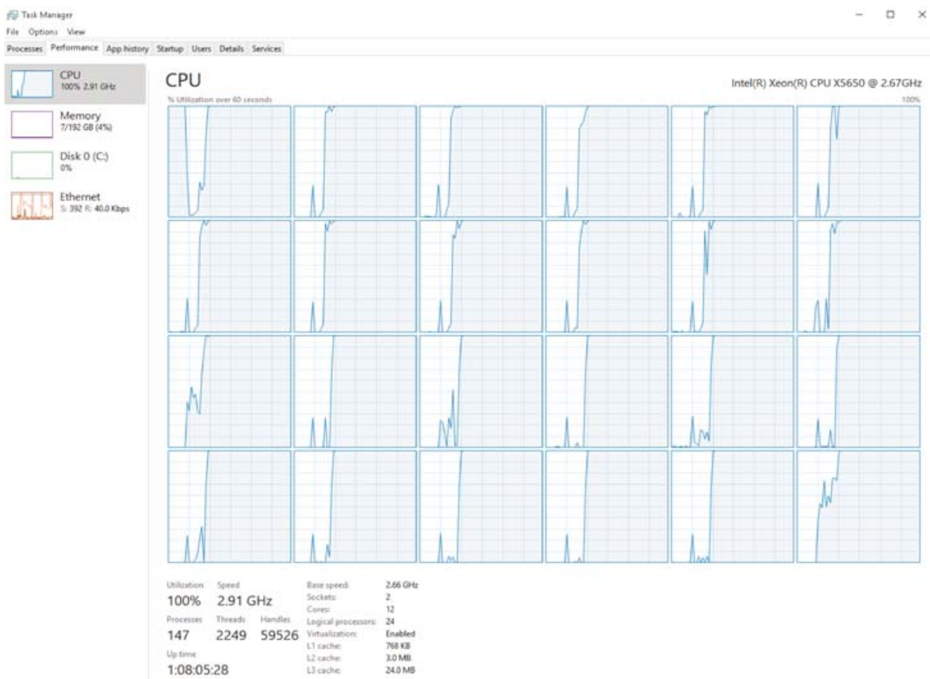
55,000 joints, 56,020 SBHQ6, SBHT6 elements, 330,000 DOFs, 50 Modes

The analysis was executed on a computer equipped with an Intel® Xeon® processor having the following specifications:

Intel® Xeon® CPU X5650 @ 2.67 GHz

Base Speed:	2.66 GHz
Sockets:	2
Cores:	12
Logical Processors:	24
L1 Cache:	760 KB
L2 Cache:	3.0 MB
L3 Cache:	24 MB

The next figure is taken from the Windows® 10 Task Manager Performance window showing the 24 logical processors in transition to 100% active during the execution of the analysis.



The total CPU time for the execution was approximately10 minutes.

2.8 DBX

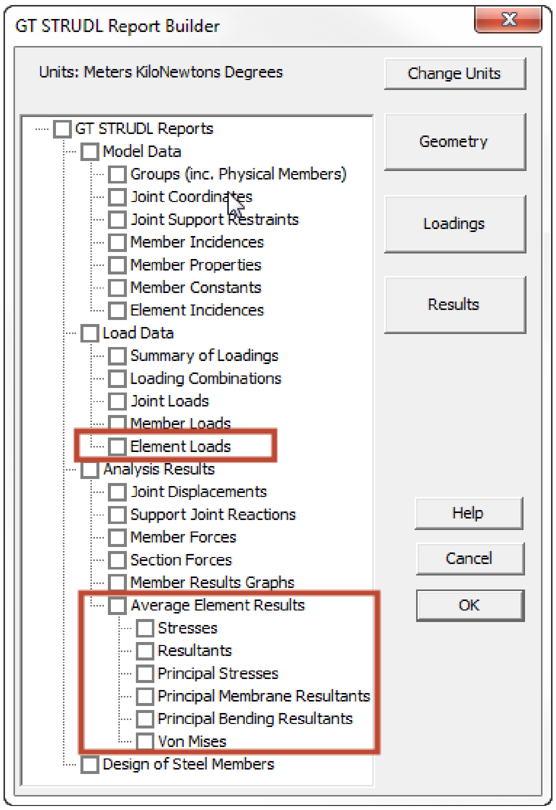
1. A new DBX data format for Applied Element Loads has been added and the format can be used to write the loads to files in the DBX format. Regular finite elements and the IPCABLE element are covered for the new load DBX data format. For example, you can use the WRITE command with the new command option APPLIED ELEMENT LOADS for the new DBX data as follows:

   WRITE APPLIED ELEMENT LOADS 'filename' ALL ELEMENTS

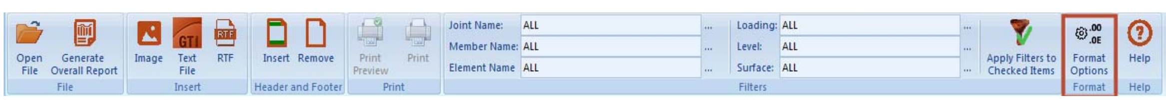
   Note that the DBX data for UNIFORM element loads are supported for any element load cases while those for VARIABLE element loads are supported only for the element load cases where the loads are applied to 8 or less joints of an edge, a face, or a volume for edge forces, surface forces, or body forces, respectively. If the element loads are applied to more than 8 joints, the WRITE command will not export the loads to the DBX file and will issue a warning message.

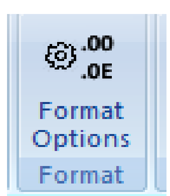
2.9 Report Builder

1. Element loads and Average Element Results are now included in the Report Builder dialogs in CAD Modeler, GTMenu and GTShell. The modified dialog is shown below:

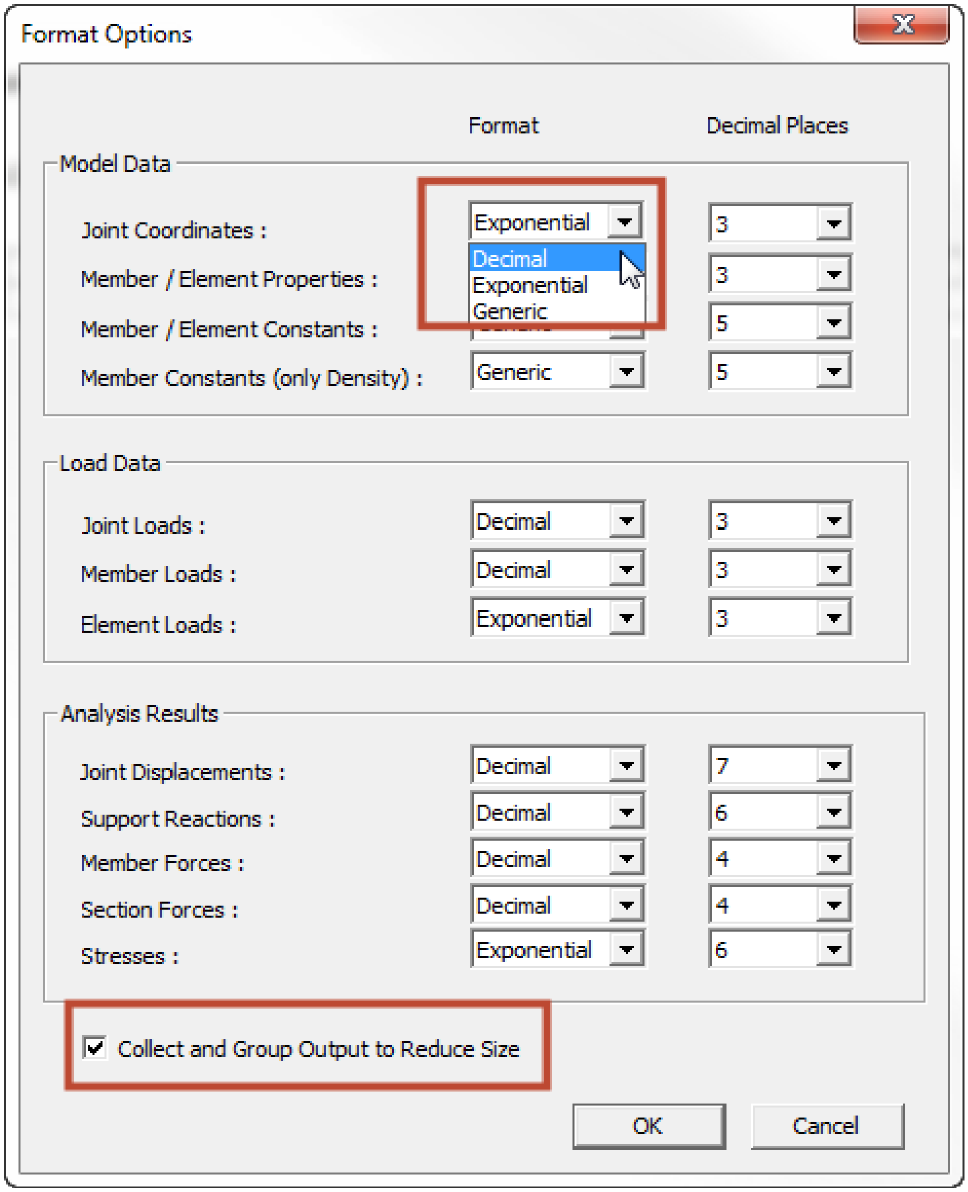


2. A new Format panel has been added to the Ribbon Bar as highlighted and shown below:





The Format Options dialog is shown on the next page:



In this dialog, you can select the format for the numerical data in the report as shown above (decimal, exponential, generic) and the number of decimal places. In addition, there is an option highlighted near the bottom of the dialog to "Collect and Group Output to Reduce Size". By selecting this option, data in the report that is the same for a group of members or elements is collected and a list is used to report the data rather than output the same data for each member or element. An example illustrating the grouping of data associated with multiple members is shown below:

Model Data

Member / Element Constants

Length: M, Force: KN, Angle: RAD, Temperature: DEGF , Time: SEC

Member		E	G	DENSITY	POISSON	CTE	BETA
1 TO 27		1.9995e+08	7.5842e+07	76.901	0.3	6.5e-06	0

2.10 New Examples

1. Three new nonlinear static analysis example input files have been added to the Version 2019 installation *Examples* folder as follows:

   exnlpa01.gti -- nonlinear pushover analysis of a simple plane frame that includes nonlinear spring member end connections to model plastic hinge behavior

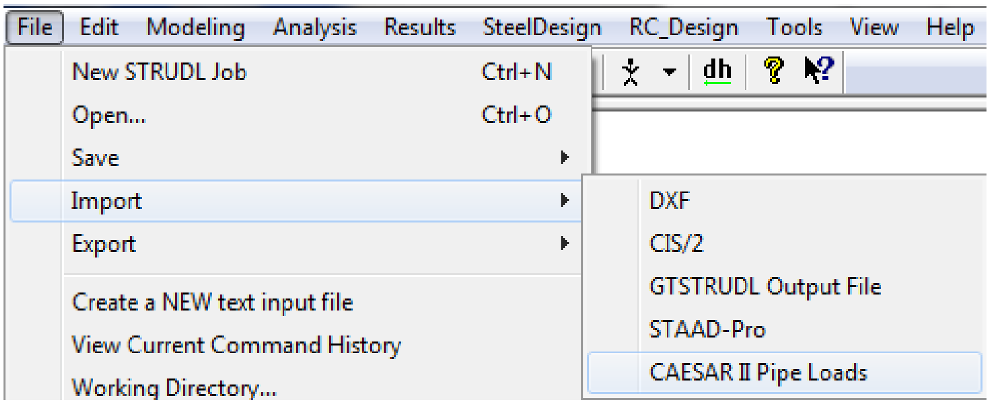
   exnlpa02.gti -- nonlinear pushover analysis of a simple plane frame that uses discrete member end plastic hinges modeled by cross-section fibers

   exnlpa03.gti -- nonlinear pushover analysis of a simple plane frame that uses member end plastic segments modeled by cross-section fibers

   When the standard Version 2019 GT STRUDL installation is used, the *Examples* folder path is *C:\Program Files (x86)\GTStrudl\2019\Examples*.

2.11 Import CAESAR II Pipe Loads

1. A new prerelease feature has been added to the File –> Import menu.



This feature allows you to create GT STRUDL loading commands based on the data in a CAESAR II .mdb file. Tools are supplied to help with assigning CAESAR II nodes to GT STRUDL members, editing the load values, such as rounding values up, and to compare the current CAESAR II results with a previous set of results. See “Section 5.4.6 Import CAESAR II Pipe Loads” in Chapter 5 for more information.

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 Analysis

1. In versions prior to Version 2019, the use of the standard solver form of the PERFORM NUMERICAL INSTABILITY ANALYSIS command following a GTSES/GT64M stiffness analysis that is used to initially detect a structural instability results in a subsequent LIST DISPLACEMENTS command that fails to report the joint displacements from the GTdebug load created by the PERFORM NUMERICAL INSTABILITY ANALYSIS command. The command sequence that produces this condition is given as follows:

   ```
   ACTIVE SOLVER GTSES
   STIFFNESS ANALYSIS
   PERFORM NUMERICAL INSTABILITY ANALYSIS
   LIST JOINT DISPLACEMENTS
   ```

   This problem has been corrected.

   (GPRF 2019.01)

3.2 Base Plate Wizard

1. The problem with the Add or Edit Loading dialog when the base plate has more than 9 Attachments has been corrected. The drop down menu to select the Attachment will be ordered properly.

   (GPRF 2019.03)

3.3 CAD Modeler

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

1. When meshing a region which contains holes using the 2D Area meshing function, missing elements near the holes no longer occur.

2. The steel code check datasheet no longer shows *'s for the KL/r provision when the critical provision isn't the governing KL/r ratio.

3. The Place Member dialog now appears correctly in the AutoCAD versions of CAD Modeler if you click on the corresponding ribbon item 3 or more times

.

4. Displacements/Deformations on models which contain finite elements are now displayed and can be annotated.

5. An erroneous ID for the 1st group in the Group List is no longer created after deleting the first group or in models which do not contain any groups.

6. An abort will no longer occur when you are in the process of creating a new group and then decide to delete the group before clicking on the OK button.

7. When importing a gti file, the orientation of the members will now be correct if the gti file has BETA=value for ALL. This error also occurred when exporting a model to CAESAR II and has been corrected there also.

8. The visibility of Levels is preserved now when using Colors/Group filters.

9. When specifying the Beta angle in the meshing dialogs, multiple instances of the "Enter Value" dialog no longer occur.

10. Finite element stress contours are now displayed for loading combinations. In addition, the datasheet for element stresses now contains results for loading combinations.

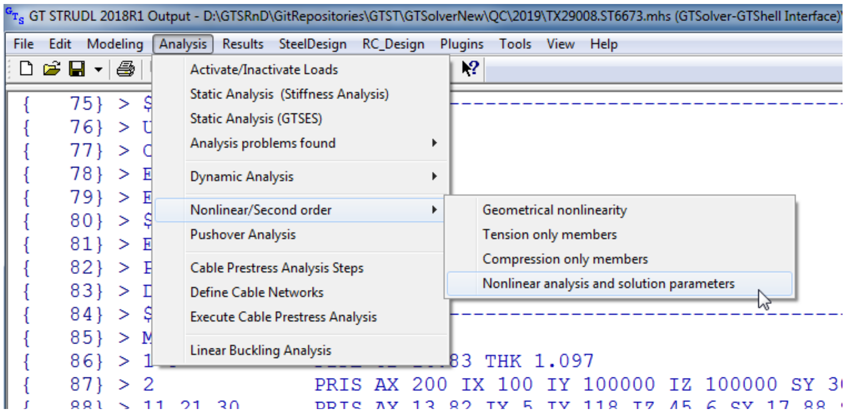
11. The finite element stress datasheet now reports VonMises stresses that can be compared with the yield stress. Previously, the VonMises stresses in the datasheet reported values that needed to be multiplied by $\sqrt{3}$ in order to compare with the yield stress.

12. Models with multiple material properties in an existing gti file can now be imported correctly.

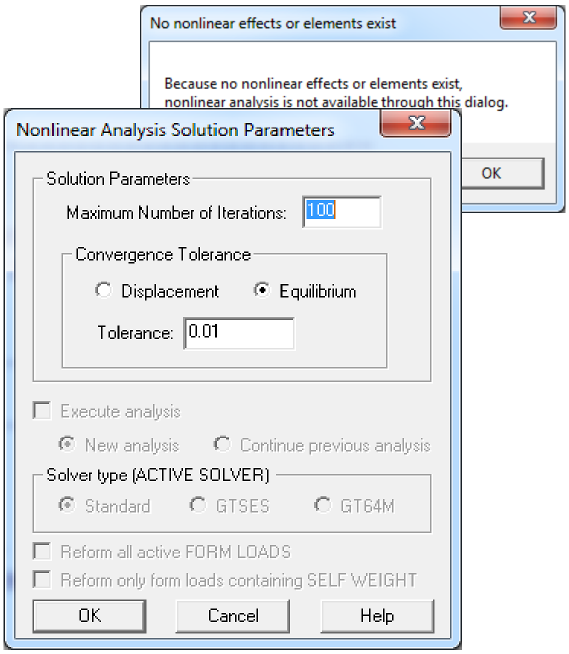
3.4 **GTShell** (GT STRUDL Output Window)

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

1. In versions prior to and including 2018R1, the execution of a nonlinear static analysis from the Nonlinear Analysis dialog issues a message box stating the absence of any nonlinearity if the structural model contains only plastic segment nonlinearity, as shown in the following figure:



This condition causes the nonlinear analysis execution options in the Nonlinear Analysis dialog to be disabled when the OK button in the message box is clicked:



Note that plastic segment nonlinearity can be defined only by command, an example of which is given as follows:

```
UNIT MM N
NONLINEAR EFFECTS
PLASTIC SEGMENT END -
 FIBER GEOMETRY NTH 32 NTWALL 1 LH 149 -
 STEEL FY 275 ESH 0.01 FSU 430 ESU 0.20 MEMBER 26
PLASTIC SEGMENT START -
 FIBER GEOMETRY NTH 32 NTWALL 1 LH 149 -
 STEEL FY 275 ESH 0.01 FSU 430 ESU 0.20 MEMBER 28
           .
           .
           .
```

As of Version 2019, plastic segment nonlinearity is now detected and this condition no longer occurs.

2. The FE Average Stresses datasheet (Results –> Datasheets) no longer reports incorrect stresses for loads after the first load case.

   (GPRF 2019.02)

3.5 GTMenu

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

1. In GT STRUDL versions prior to 2019, when nonlinear effects and nonlinear analysis parameters, including a displacement convergence tolerance value, had been specified in the structural model data, the command text input file produced by GTMenu, if requested, will contain a nonlinear analysis CONVERGENCE TOLERANCE DISPLACEMENT command in which a negative (-) displacement tolerance value is specified, for example:

   CONVERGENCE TOLERANCE DISPLACMENT -0.001

   This has been corrected in Version 2019. The command input file containing the specified nonlinear analysis data commands now includes a CONVERGENCE TOLERANCE DISPLACEMENT command that specifies a positive (+) displacement tolerance value.

2. The error In GT STRUDL 2018 R1 when splitting a physical member with end joint size has been corrected.

3. If you select a Cylindrical or Spherical coordinate system, the size of the text box for coordinate input in the Create Joint dialog is no longer extended.

4. If you click the All button under Mark Loads in the Display Model dialog, there are no longer error messages when no joints, members or elements are not loaded. The existing joint, member, and element loads are now marked.

5. If you click the Query Member Loads button from the Edit Member Load dialog, you can now select members to query after a previous query cancelled the selection from the Selection Filter dialog.

6. If you click the Query Member Loads button in the Edit Member Load dialog after displaying the member loads, the query output is now correctly displayed in the Inquire Output and Edit dialog.

7. If you delete loads using "Delete All with Same Value & Location" using the Selection Filter dialog, previous query information in the Inquire Output and Edit dialog is now updated.

8. If you click the line with the"Member ID" when querying member load in the Inquire Output and Edit dialog, an abort will no longer occur.

9. If you delete element loads using the Selection Filter dialog, the loads are now deleted.

10. If you delete a member load using "Delete Selected Load Only" in the Selection Filter dialog and then you delete another member load using "Delete All with Same Value & Location" in the Selection Filter dialog, there will be no longer be an abort.

11. In the Create/Edit Joint dialog, duplicate joint names(ID's) can no longer occur when creating or editing joint ID's.

12. In the Create/Edit Point dialog, duplicate point names(ID's) no longer occur when creating or editing point ID's.

13. If you define physical members and display the deformed shape under the Analytical Member selection from the Mode Bar and then select Physical Member in the Label Setting dialog, the deformed shape will now be displayed correctly.

14. If you define additional members after deleting members and then perform an analysis in GTMenu, the member releases will now be correct.

15. When labeling the values on a finite element contour display in LIST mode, two dialogs no longer appear asking for the joint to label.

16. After displaying joint reactions, you can now Exit GTMenu without being put into selection mode.

3.6 General

1. Problems related to the load combination and form load results produced by the CREATE AUTOMATIC LOAD COMBINATION DESIGN command have been corrected as follows:

   a. The generation of load combinations and form loads for ASCE 7-10 load combination 4 with ice now includes exclusive live loads that are associated with the LEX design load variable.

   b. Load combinations and form loads for ASCE 7-10 and AISC 14, LRFD load combination equations 4 and 6 and ASD load combination equations 5, 6a, 6b, and 7 when FA1 or FA2 flood zone loads are now generated including the complete set of required basic load combinations or form loads in addition to those that include the FA1 and FA2 loads.

   (GPRF 2018.08)

3.7 Export to CAESAR II

1. Models which contain multiple material properties can now be exported to CAESAR II correctly.

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 2019. 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:

5.2 Design Prerelease Features

- 5.2.1 A new national annex parameter for EC3-2005 steel design code
- 5.2.2 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)
- 5.2.3 ASCE4805 Steel Design Code. This code is for the ultimate strength design of steel transmission pole structures.

5.3 Analysis Prerelease Features

- 5.3.1 Calculate Error Estimate Command
- 5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

5.4 General Prerelease Features

- 5.4.1 Rotate Load Command
- 5.4.2 Reference Coordinate System Command

5.4.3 GTMenu Point Coordinates and Line Incidences Commands

5.4.4 GTMenu Surface Definition Command

5.4.5 Export to CAESAR II

5.4.6 Import CAESAR II Pipe Loads

We encourage you to experiment with these prerelease features and provide us with suggestions to improve these features as well as other GT STRUDL capabilities.

5.2 Design Prerelease Features

5.2.1 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 $(\overline{\lambda}_{LT,0})$). 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 = $0.9(f_u / f_y) \geq 1.1$

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	α_{LT}
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	α_{LT}
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:

$$
\begin{array}{l}
\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} \\ \quad \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 \text{ } \underline{\text{ELE}}\text{MENT list}_2 \text{ (}\underline{\text{TABLE}} \left\{ \begin{array}{l} \rightarrow \underline{\text{ASTM}} \\ \quad \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{S) (}\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)} \\ \quad \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{)}
\end{array}
$$

where,

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

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

Explanation:

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

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

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

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

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

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

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

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

Requirements:

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

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

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

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

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

Usage:

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

Usage Example: Description of Example Structure

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

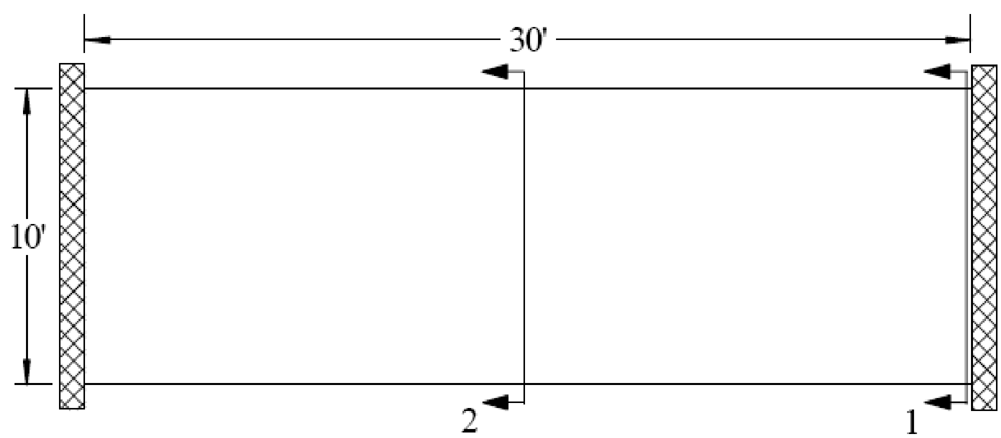


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

GT STRUDL Finite Element Model

The example structure was modeled in GT STRUDL using PLATE BENDING finite elements. The BPHQ element was utilized, and the configuration modeled corresponded to a mesh of ten elements by thirty elements. The model contained 300 finite elements and 341 nodes. The material properties were the default values associated with the MATERIAL REINFORCED CONCRETE command. All 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.

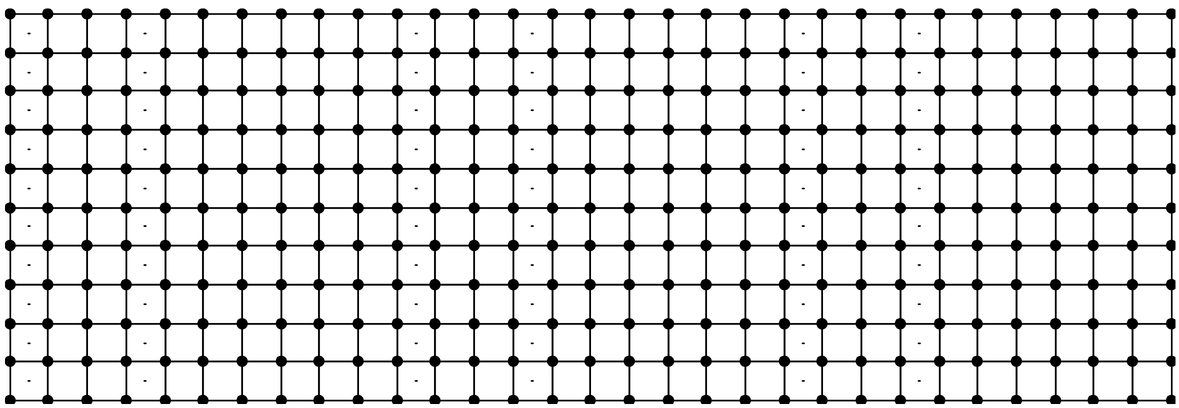


Figure 5.2.3-2 Example Finite Element Model

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) -

$$
* \left\{ \begin{array}{l}
\underline{\text{ENERGY}}\ (\underline{\text{NOR}}\text{M}) \\
\underline{\text{MAX}}\ \underline{\text{DIFF}}\text{ERENCE} \\
\underline{\text{DIFF}}\text{ERENCE}\ \underline{\text{FROM}}\ \underline{\text{AVER}}\text{AGE} \\
\underline{\text{PERCENT}}\ \underline{\text{MAX}}\ \underline{\text{DIFF}}\text{ERENCE} \\
\underline{\text{PERCENT}}\ \underline{\text{DIFF}}\text{ERENCE}\ \underline{\text{FROM}}\ \underline{\text{AVER}}\text{AGE} \\
\underline{\text{NORM}}\text{ALIZED}\ \underline{\text{PERCENT}}\ \underline{\text{MAX}}\ \underline{\text{DIFF}}\text{ERENCE} \\
\underline{\text{NORM}}\text{ALIZED}\ \underline{\text{PERCENT}}\ \underline{\text{DIFF}}\text{ERENCE}\ \underline{\text{FROM}}\ \underline{\text{AVER}}\text{AGE}
\end{array} \right\} -
$$

$$
(\underline{\text{AT}})^{*} \left\{ \begin{array}{l}
\underline{\text{TOP}} \\
\underline{\text{MID}}\text{DLE} \\
\underline{\text{BOT}}\text{TOM}
\end{array} \right\} (\underline{\text{SUR}}\text{FACES})\ (\underline{\text{FOR}}) \left\{ \begin{array}{l}
\rightarrow \underline{\text{ALL}} \\
\underline{\text{ELE}}\text{MENT list}
\end{array} \right\}
$$

The results from this command provide an estimate of the errors in the finite element discretization of the problem. Energy norm (L_2 norm) and nodal error estimates are available.

The L_2 norm is given by:

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

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

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

where N is the shape functions used for the assumed displacement field of the element.

The stress norm uses the average stresses and is given by:

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

The relative percentage error which is output for each element is given by:
The nodal error estimates estimate the accuracy of the data in a selected nodal output vector.

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

Six nodal error estimation methods are available:

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

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

Maximum Difference Method

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

Difference from Average Method

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

Percent Maximum Difference Method

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

Percent Difference from Average Method

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

Normalized Percent Maximum Difference

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

Normalized Percent Difference from Average Method

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

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

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

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

1. Enter GTMenu.

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

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

4. Select the Loading.

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

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

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

General form:

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

Explanation:

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

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

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


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

      PROBLEM - None

      ACTIVE UNITS  FEET KIP  RAD  DEGF SEC
```

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



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

CONSTANTS/BETA Commands for Adjusted Beta Angles
================================================

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

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

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

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

Elements:

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

r_1, r_2, r_3 = values in current angle units of the load component rotation angles θ_1, θ_2, θ_3 as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

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

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

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

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

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

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

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

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

Examples:

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

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

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

Previously defined loading 3 is specified in CHANGES mode, followed by a return to ADDITIONS mode. The ROTATE LOAD command is then given to add the components of load 4, including appropriate rotations, to loading 3.

Error Messages:

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

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

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

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

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

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

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

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

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

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

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

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

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

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

5.4.2 REFERENCE COORDINATE SYSTEM Command

General form:

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

$$\left\{ \begin{array}{l} (\underline{\text{ORI}}\text{GIN } [\,\underline{X}\,]\ v_x\ [\,\underline{Y}\,]\ v_y\ [\,\underline{Z}\,]\ v_z)\ (\underline{\text{ROT}}\text{ATION } [\,\underline{R1}\,]\ v_1\ [\,\underline{R2}\,]\ v_2\ [\,\underline{R3}\,]\ v_3) \\ \left\{ \begin{array}{l} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X}\ v_4\ \ \underline{Y}\ v_5\ \ \underline{Z}\ v_6 \end{array} \right\} \left\{ \begin{array}{l} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X}\ v_4\ \ \underline{Y}\ v_5\ \ \underline{Z}\ v_6 \end{array} \right\} \left\{ \begin{array}{l} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X}\ v_4\ \ \underline{Y}\ v_5\ \ \underline{Z}\ v_6 \end{array} \right\} \end{array} \right\}$$

Explanation:

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

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

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

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

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

In either case, i_1 or 'a_1' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x, v_y, and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x, v_y, and v_z, are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive directions of these axes; v_1, v_2, and v_3 are the rotation angles R_1, R_2, and R_3 in active angular units between the orthogonal axes of this system and the axes of the overall global coordinate system. The description of these angles is the same as given in Section 2.1.7.2 of Volume 1 of the GT STRUDL User Reference Manuals for rotated joint releases (θ_1, θ_2, and θ_3).

In the second case, three joints are required. Each of the three joints may be defined either by a joint identifier using the JOINT option of the command or by its global X, Y, and Z coordinates. If the joint identifier option is used, however, the coordinates of the joint must be specified previously by the JOINT COORDINATES command. The first time (i_2 or '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:

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

Explanation:

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

5.4.3 GTMenu POINT COORDINATE and LINE INCIDENCES Commands

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

(1) **GTMenu POINT COORDINATES**

General Form:

GTMenu POINT COORDINATES

$$\left\{ 'a_1' \right\} \quad \text{coordinate-specs}_1$$

.
.
.

$$\left\{ 'a_n' \right\} \quad \text{coordinate-specs}_n$$

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

$\left\{ \text{'a}_1\text{'} \right\}$ type$_1$ incidence-specs$_1$

.
.
.

$\left\{ \text{'a}_n\text{'} \right\}$ type$_n$ incidence-specs$_n$

Elements:

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

$$\text{incidence-specs} = \left\{ \text{point}_1\text{'} \right\} \left\{ \text{'point}_2\text{'} \right\} . . . \left\{ \text{'point}_p\text{'} \right.$$

where,

‘a_1’, ‘a_2’, ..., ‘a_n’	=	1 to 8 character alphanumeric Line/Curve identifiers beginning with C (i.e. C1, C2).
‘$point_1$’, ..., ‘$point_p$’	=	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:

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

$$\begin{array}{c} \left\{'a_1'\right\}\ surface-specs_1 \\ \vdots \\ \left\{'a_n'\right\}\ surface-specs_n \end{array}$$

Elements:

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

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

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

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

where,

'a_1', 'a_2', ..., '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:

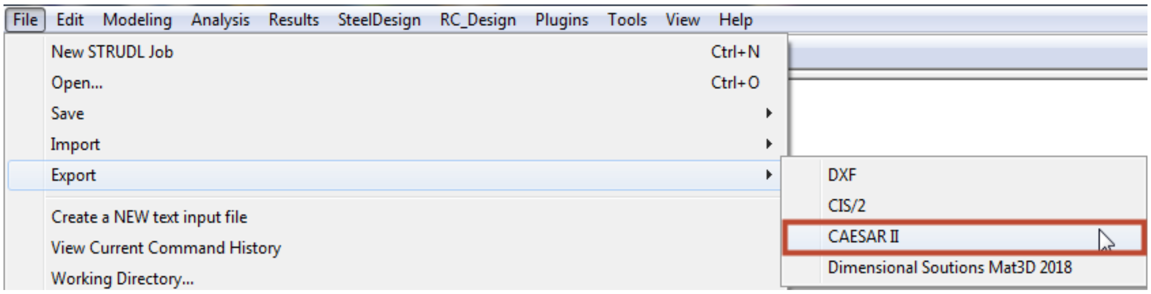
```
GTMenu SURFACE DEFINITION
    'S1' PATCH SURFACE SPACING 10 20 -
        U CURVES 'C1' -
        V CURVES 'C2'

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

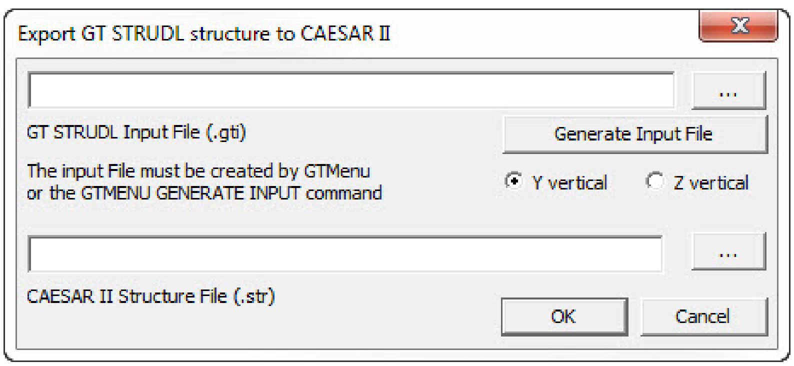
```
'S3' SURFACE OF REVOLUTION SPACING 10 20 -
    REVOLUTION ANGLE 360 -
    AXIS COORDINATES START  10.0 0.0 10.0 -
                     END    20.0 0.0  0.0 -
    U CURVE 'C2'
```

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:

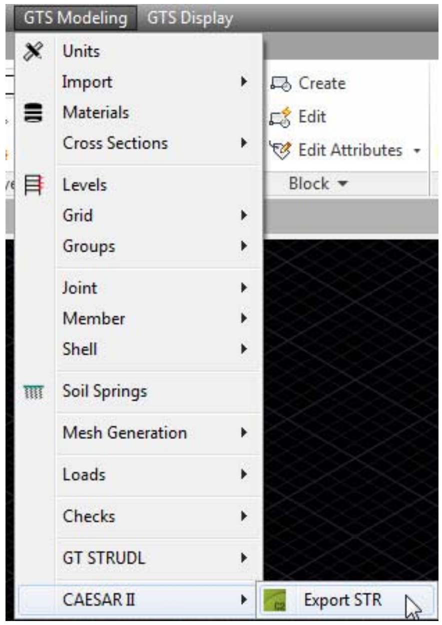


After selecting CAESAR II, the following dialog will pop-up:



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:

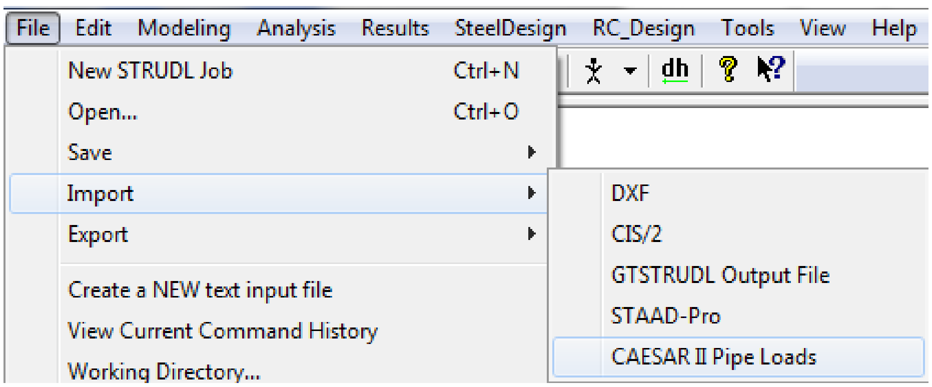
WARNING: Section L1x1x1/4 is not available in CII, please use another one or edit F:\\HexagonPPM\CaesarII\PlantStructure\\PStructure_0708_01.str file manually

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).

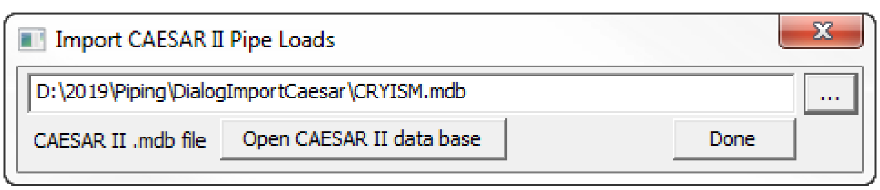
5.4.6 Import CAESAR II Pipe Loads

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:



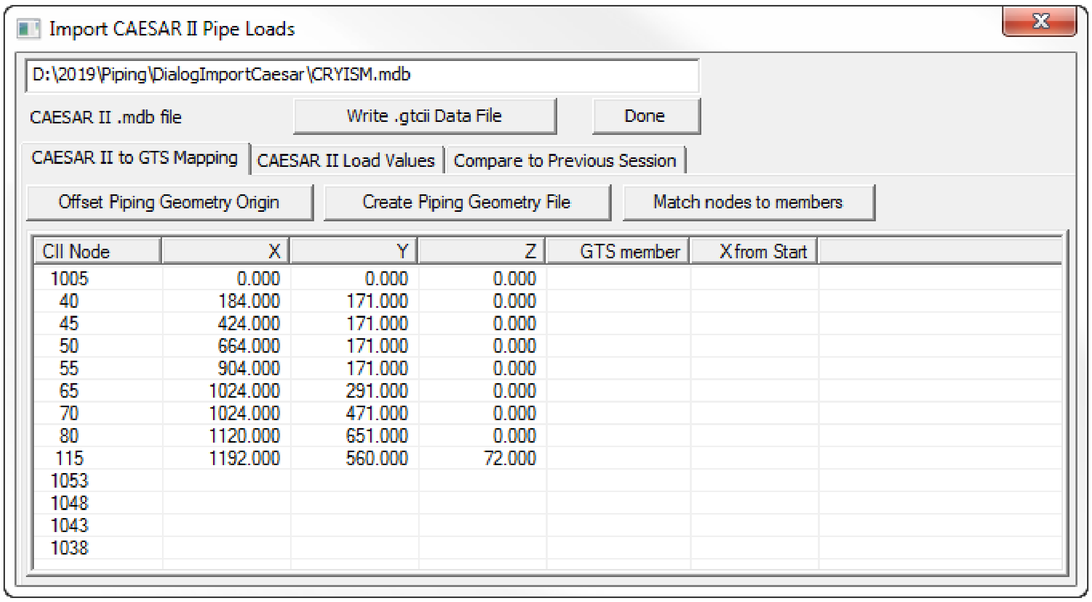
Clicking the menu item will launch the opening dialog, and if there is structural data in the GT STRUDL data base, write files so the dialog can get the structural model data. Choose a .mdb file by typing or browsing with the button and then click the 'Open CAESAR II data base' button.



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



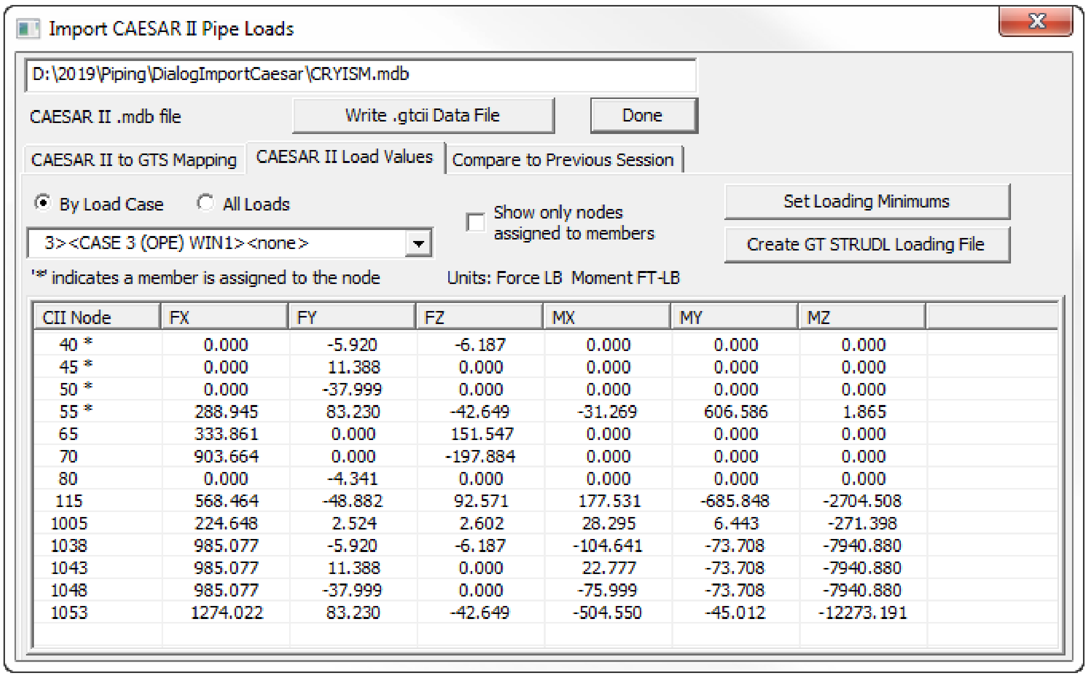
This tab is used to assign the CAESAR II nodes to a location along a GT STRUDL member. Only reactions for nodes with a 'GTS member' assignment can be exported to a loading file. You can type member names and fractional distance along the local X axis into the grid by double-clicking the cell. Press Enter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



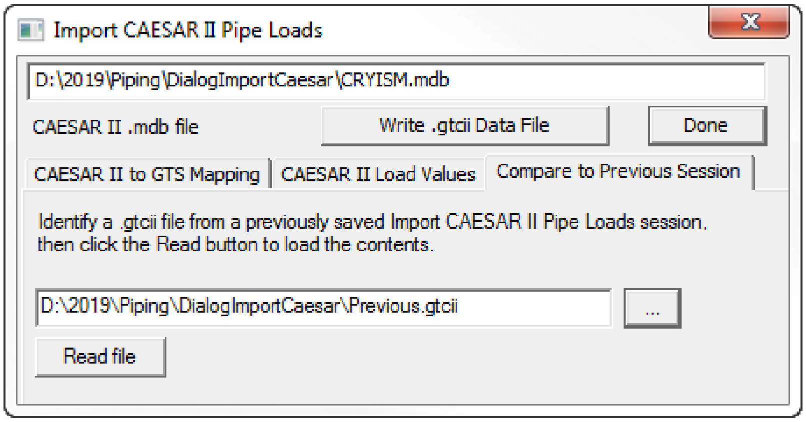
Reaction values can be displayed with only one load case in the grid, or all loads in the grid. A dropdown menu is provided to shift between load cases when in single load case mode. Only nodes with assigned GT STRUDL members, those with an entry in the “GTS member” column, can be exported into a loading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

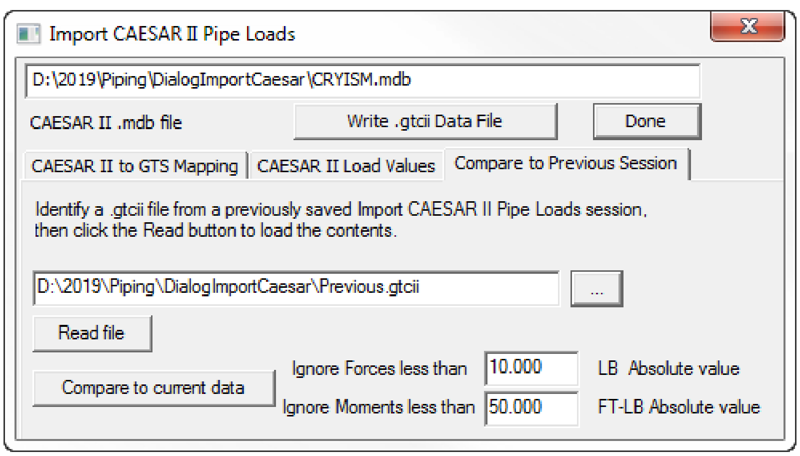
If you want to be able to compare this load data with a subsequent CAESAR II pipe stress analysis, use the “Write .gtcii Data File” button to save the current data to used with the next tab.

The “Compare to Previous Session” Tab

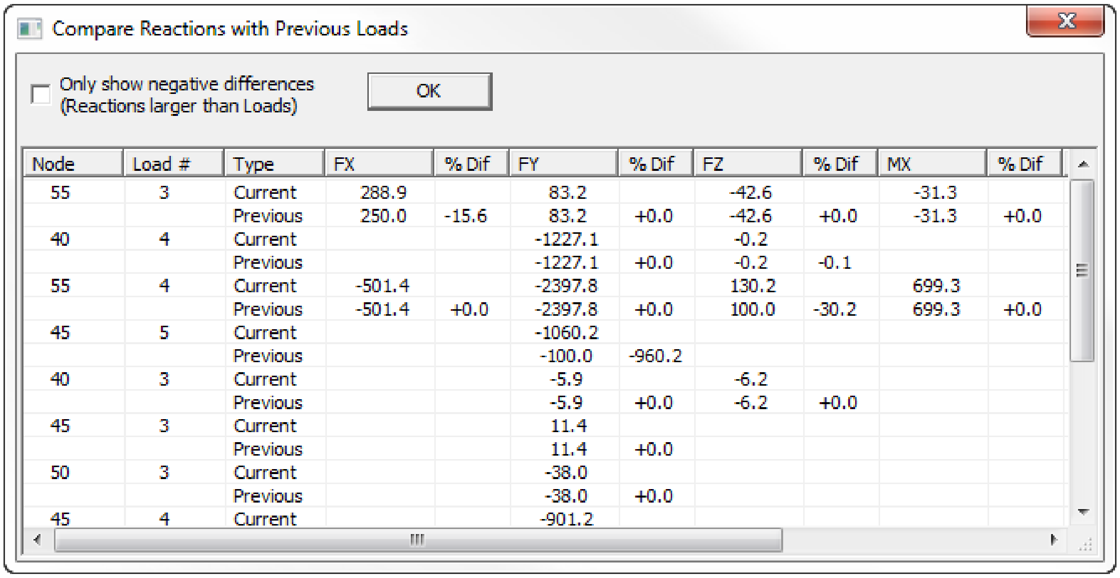
If you have saved a previous data file (.gtcii extension), you can use that to compare with a new CAESAR II analysis to verify that the new reaction values do not exceed the loading values sent to GT STRUDL. Enter or browse for the .gtcii file and then click the ‘Read file’ button.



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.


Compare Reactions with Previous Loads

Only show negative differences (Reactions larger than Loads)

OK

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					