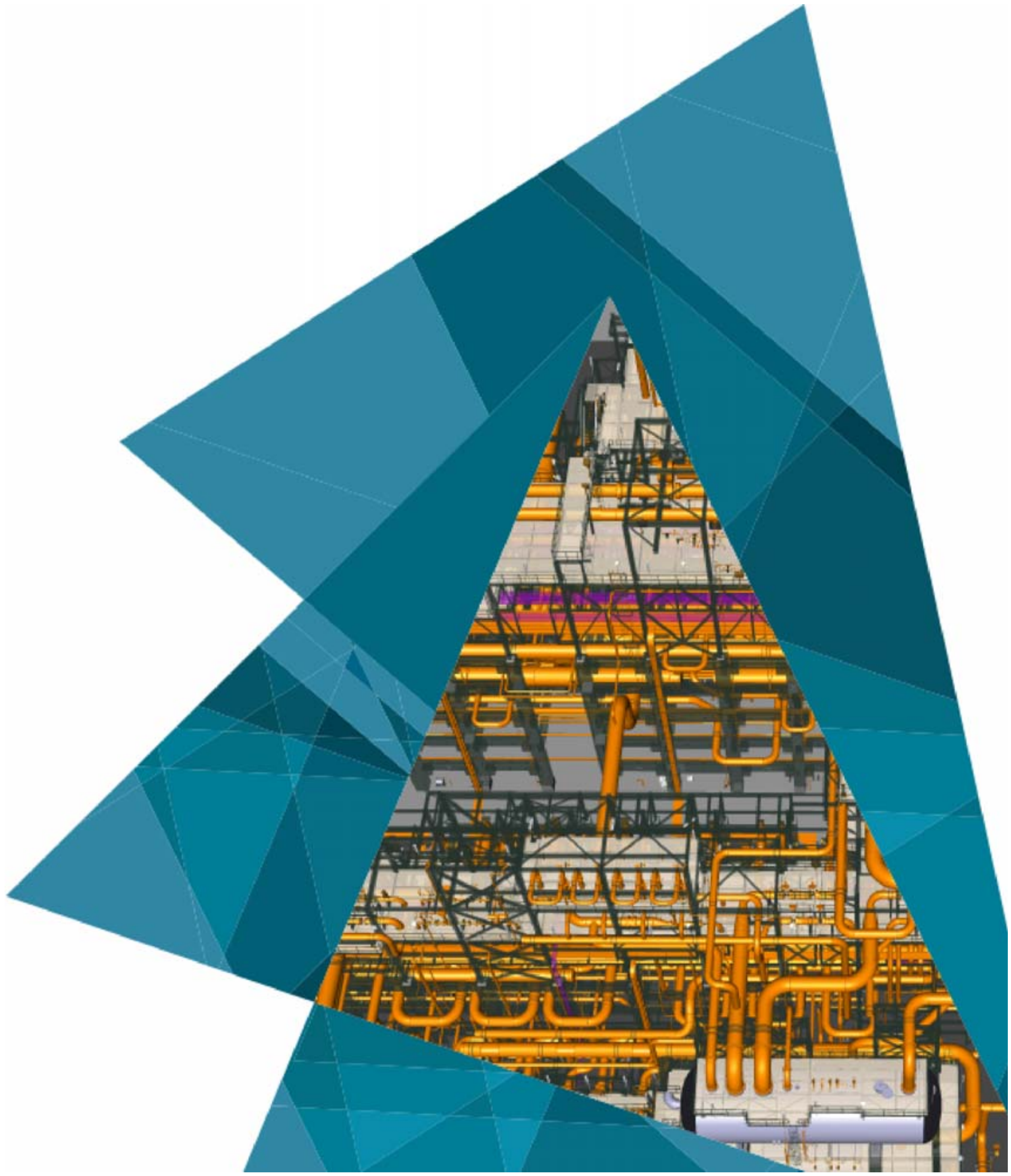
GT STRUDL® Version 2018 Release Guide



Release Date: December 2017

HEXAGON PPM

Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 2018 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

U.S. Government Restricted Rights Legend

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

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

Other Documentation

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

Terms of Use

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

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

Disclaimer of Warranties

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

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

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

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

Limitation of Damages

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

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

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

Export Controls

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**

NOTICES iii

Table of Contents v

Chapter 1

Introduction 1-1

Chapter 2 New Features in Version 2018

2.1 Steel Design 2-1
2.2 GTMenu 2-3
2.3 CAD Modeler 2-20
2.4 Nonlinear Analysis 2-29
2.5 GTSES/GT64M High Performance Solver 2-30
2.6 DBX 2-31
2.7 Interfaces 2-31
2.8 Loadings 2-32
2.9 GTShell (GT STRUDL Output Window) 2-33
2.10 Installation 2-37
2.11 Base Plate Wizard 2-37
2.12 Report Builder 2-38

Chapter 3 Error Corrections

3.1 Base Plate Wizard 3-1
3.2 CAD Modeler 3-1
3.3 Joint Constraints 3-2
3.4 DBX 3-2
3.5 Finite Elements 3-3
3.6 GTMenu 3-3
3.7 GTShell (GT STRUDL Output Window) 3-6

Chapter 4 Known Deficiencies

4.1 CAD Modeler 4-1

4.2 Finite Elements 4-1

4.3 General Input/Output 4-1

4.4 GTMenu 4-2

Chapter 5 Prerelease Features

5.1 Introduction 5.1-1

5.2 Design Prerelease Features 5.2-1

5.2.1 A new national annex parameter for EC3-2005
Steel design code 5.2-1

5.2.2 Design of Flat Plates Based on the Results of Finite Element
Analysis (The DESIGN SLAB Command) 5.2-7

5.2.3 ASCE4805 Code for the Design of Steel Transmission
Pole Structures 5.2-17

5.3 Analysis Prerelease Features 5.3-1

5.3.1 Calculate Error Estimate Command 5.3-1

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA
ANGLES Command 5.3-5

5.4 General Prerelease Features 5.4-1

5.4.1 Rotate Load Command 5.4-1

5.4.2 Reference Coordinate System Command 5.4-5

5.4.2-1 Printing Reference Coordinate System Command . . 5.4-8

5.4.3 GTMenu Point Coordinates and Line Incidences Commands . . 5.4-9

5.4.4 GTMenu Surface Definition Command 5.4-12

This page intentionally left blank.

Chapter 1

Introduction

Version 2018 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 37.0.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2017. In particular, Chapter 2 briefly describes an extensive list of new features including the following new features:

- Additional cross-sections have been added to EC3-2005 code. The new cross-sections are: Channels, Tees, and Solid Rectangular Bars. Also, Single and Double Angles now can be code checked for shear, bending, and combined axial and bending strength.
- Loadings and Nonlinear Analysis solver improvements
- Numerous improvements to CAD Modeler and GTMenu
- New Report Builder

Chapter 3 provides you with details regarding error corrections that have been made since the Version 2017 release. Chapter 4 describes known problems with Version 2018. 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

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 2018

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 2018. 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. Additional cross-sections have been added to EC3-2005 code. The new cross-sections are: Channels, Tees, and Solid Rectangular Bars. Also, Single and Double Angles now can also be code checked for shear, bending, and combined axial and bending strength. In previous versions, single angles could be code checked for only axial strength.

   The EC3-2005 code of GT STRUDL may be used to select or check any of the following shapes:

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

   The code is primarily based on the "Eurocode 3: Design of Steel Structures, Part 1-1 General rules and rules for buildings", DS/EN 1993-1-1: 2005(E) (104), as published by the Danish Standards Association (DS). The EC3-2005 code is based on the limit states design technique.

2. The Parameters for EC3-2005 have been reorganized to show National Annex parameters under the heading titled "National Annex Parameters". National Annex parameters may be modified to comply with National Standards. National Annex parameters are documented in Table 1.3-1, Section 1.3, Volume EC3-2005 and shown on the next page:

Table 1.3-1

EC3-2005 Code Parameters

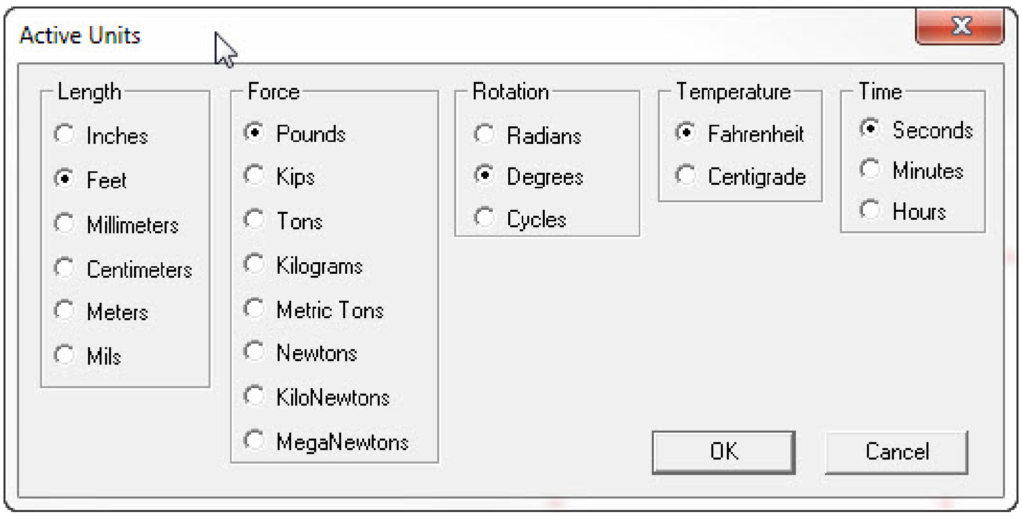
National Annex Parameters

Parameter Name	Default Value	Meaning
GM0	1.0	Partial safety factor used for resistance of Class 1, 2, and 3 cross-sections. This is γ_{M0} in the EN 1993-1-1: 2005 (E).
GM1	1.0	Partial safety factor used for resistance of members to instability assessed by member checks. This is γ_{M1} in the EN 1993-1-1: 2005 (E).
GM2	1.25	Partial safety factor used for resistance to fracture of cross-sections in tension. This is γ_{M2} in the EN 1993-1-1: 2005 (E).
Beta	0.75	Parameter to specify correction factor β for the lateral-torsional buckling curves for rolled sections. This parameter is used in the computation of Φ_{LT} and χ_{LT} factor which are used in the computation of the elastic critical moment M_{cr} (see provision 6.54 and Annex F of ENV 1993-1-1: 1992)
LamdaLT0	0.4	Parameter to specify $\bar{\lambda}_{LT,0}$, the plateau length of the lateral-torsional buckling curves for rolled sections. This parameter is used in the computation of Φ_{LT} and χ_{LT} factor which are used in the computation of the elastic critical moment M_{cr} (see provision 6.54 and Annex F of ENV 1993-1-1: 1992)

2.2 GTMenu

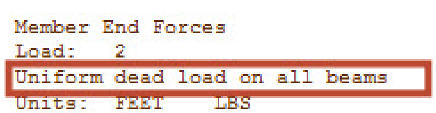
1. The Units dialogs have been replaced with the same Units dialog that is used in Command Mode (GTShell). This allows users to change all the units by opening only one dialog. Now when you click on Units you will see the following:

When you select Active Units, the dialog shown below will indicate the current units and allow you to change all of the units in one dialog:



2. The Display Load dialog now has a Legend button which will allow you to place a Legend on the display.

In addition, the Legend that is now placed when using the Legend button for Display Load or when displaying any Results includes the Load Description which makes it easier to identify the loading when results are viewed in the Scope Editor or in screen captures. An example of the new Legend illustrating the Load Description when displaying Member End Forces is shown below:



3. GTMenu will now remember the view orientation (Y up or Z up) last specified and default to that orientation the next time GTMenu is opened. This orientation setting will be remembered between GT STRUDL sessions on a per-user basis.

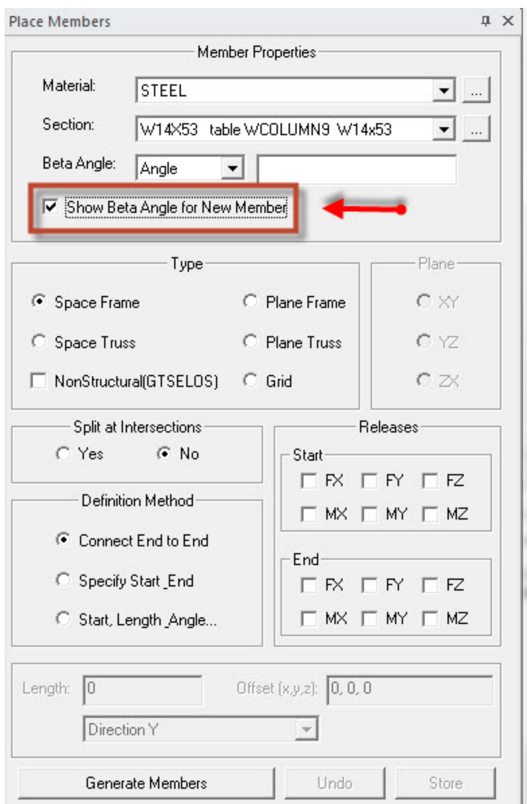
   In addition, a command may be used to indicate the desired orientation so when GTMenu is opened the orientation is correct.

   Syntax:

   GTMENU VIEW Y (UP)

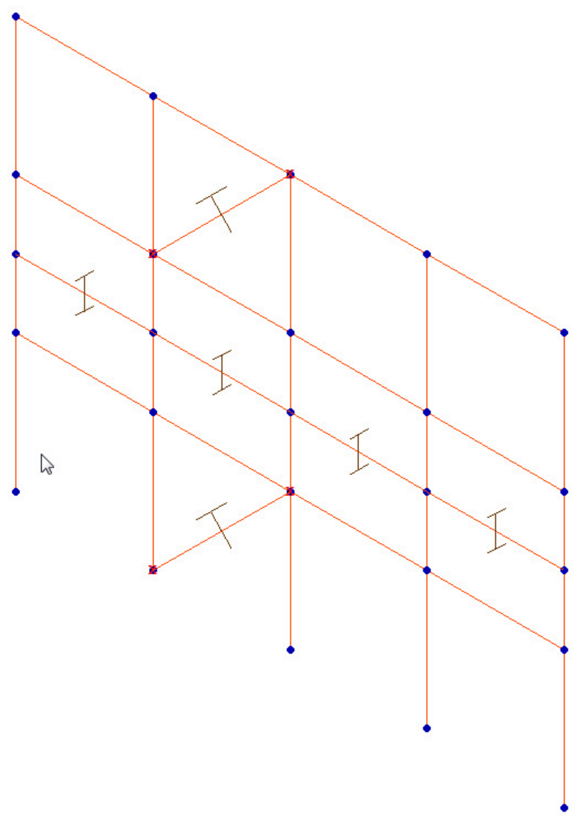
   or GTMENU VIEW Z (UP)

4. The Place Members dialog now has a new option which will display a member's profile orientation (Beta Angle) when the member is placed. This new option is shown in the Place Members dialog below:



By default, this new option is not checked but you may turn it on at any time and all newly created members which haven't been stored will show the members' profile orientation.

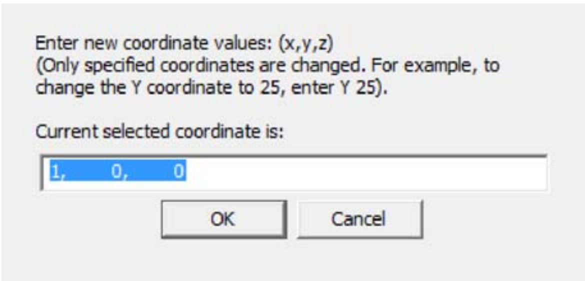
An example showing the placement of 6 new members with the members' profile orientation is shown in the figure below:



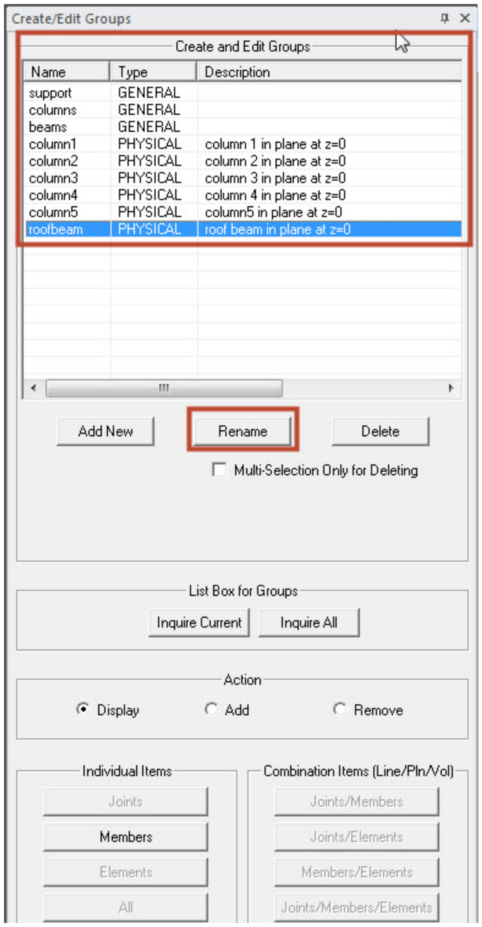
In the example above, 6 new members were created with 4 members having an I shape and 2 members having a Tee shape.

In some instances it may be difficult to visualize the orientation of the profile depending on the current view. The model may need to be rotated and resized to verify the members' orientation. The display of the profiles will be retained while rotating or zooming until the newly created members are stored.

5. The Done or OK and Cancel buttons are now consistently ordered in GTMenu dialogs. The Done or OK button is on the left and the Cancel button is on the right as shown below:

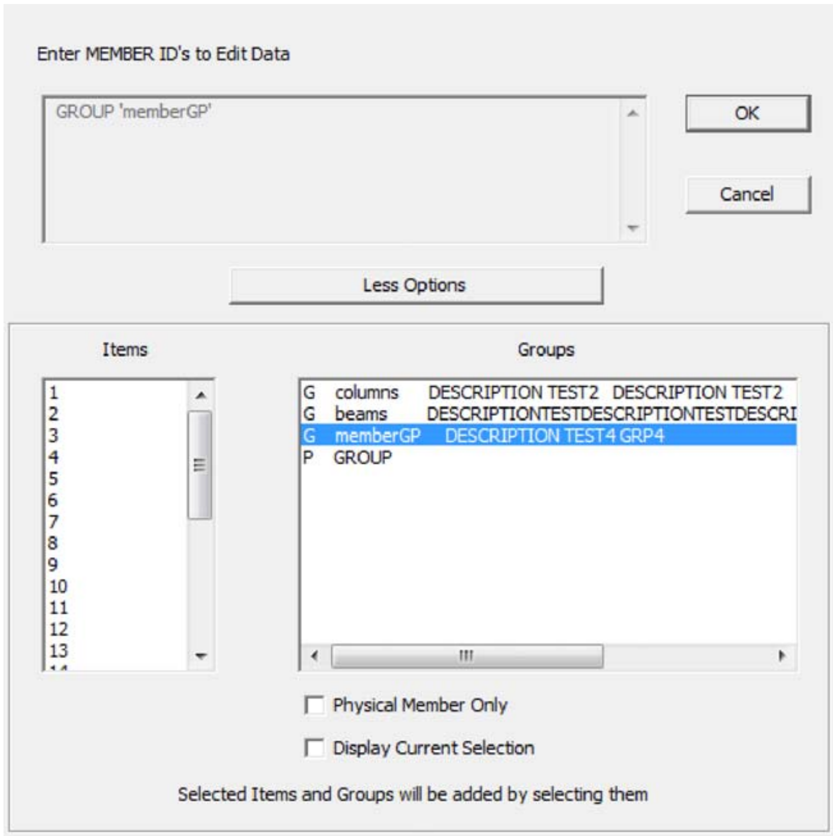


6. The Create/Edit Groups dialog has been improved to where you can now create a Group Description and also see the Description in the list of current groups. In addition, multiple groups can now be deleted. The modified Create/Edit dialog which shows the Group's Description is illustrated below:



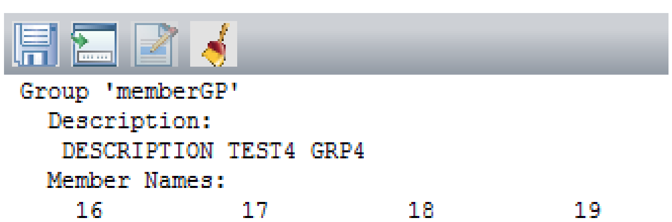
In addition, an existing Group Description may also be edited using the Rename button.

7. When in List mode, you are now able to see the Group Description which facilitates the selection of the correct Group. The following is an example of this feature:



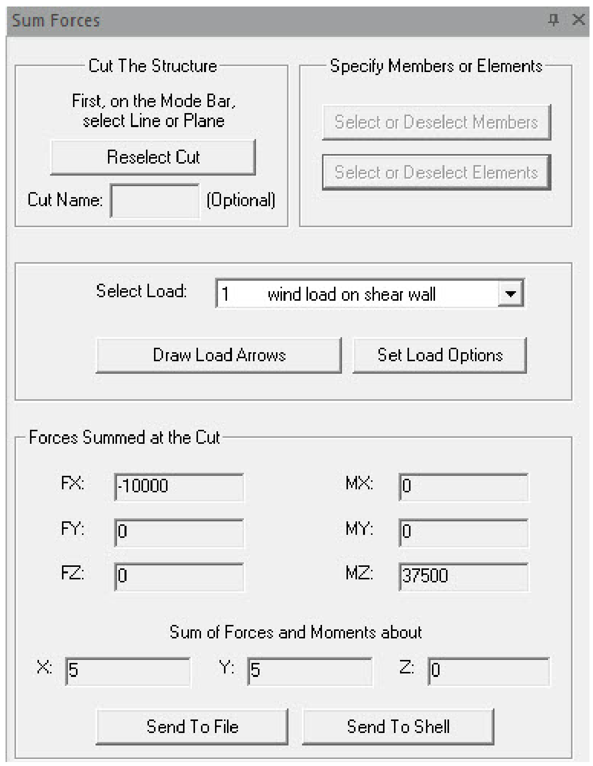
By selecting the option to Display Current Selection, the items in the list are marked in the graphical display area which allows you to more easily verify your selection.

8. You are now able to see the Group Description in the Inquire Output and Edit dialog when you select Inquire Current or Inquire All in the Edit Group Dialog:

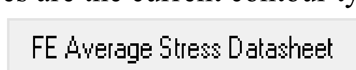


.

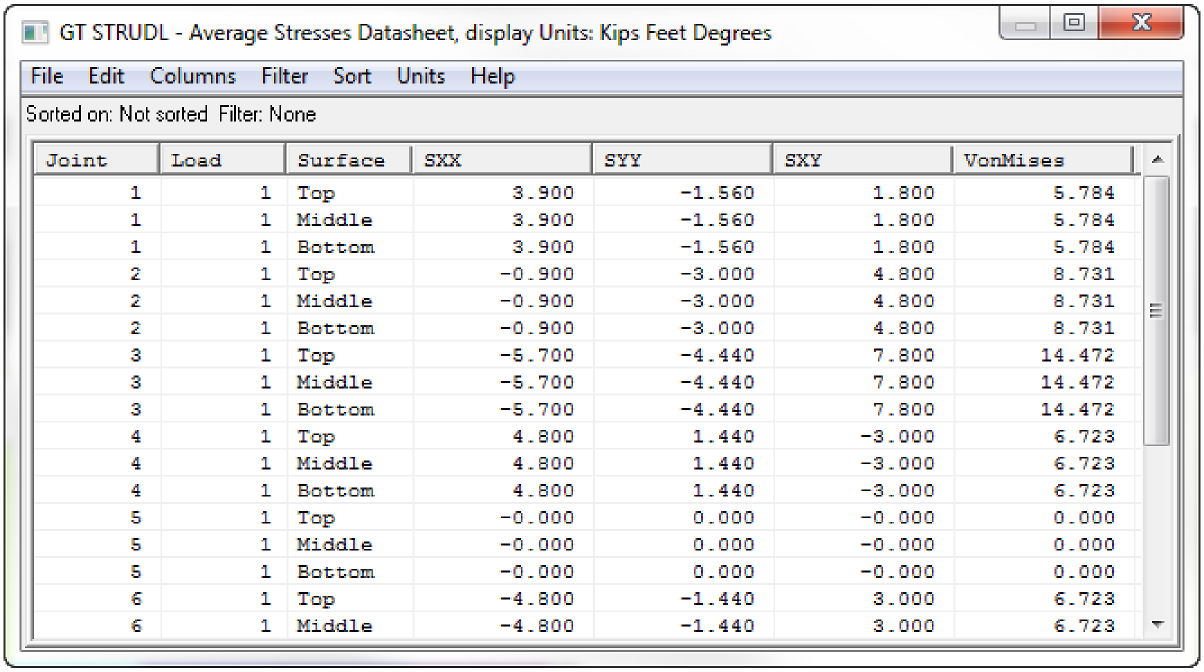
9. The Sum Forces dialog has been revised so that you are now able to retain selection information when you want to obtain results for different loadings using the same cut. Previously, the cut had to be respecified for each loading. In addition, the Forces Summed at the Cut are updated automatically when the loading is changed. The revised dialog is shown below:



10. A new datasheet has been added to the Element Contours dialog. When Stresses or Von Mises are the current contour type, the button shown below will appear



which when selected will then launch the datasheet shown on the next page:

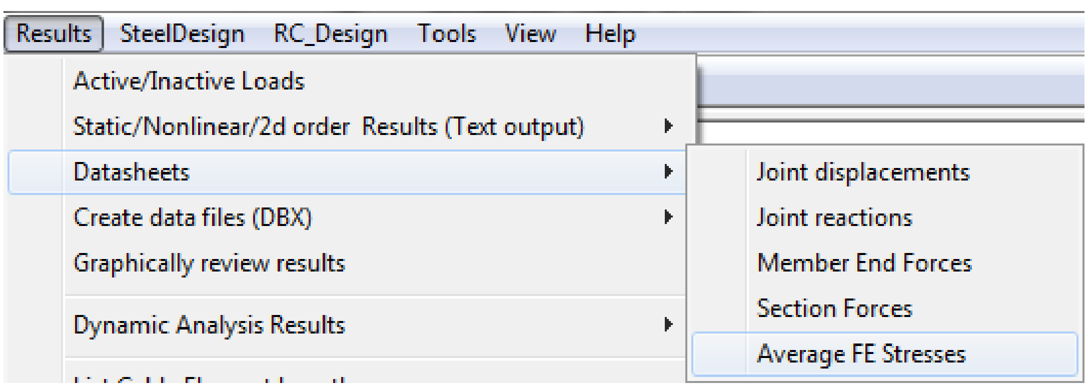

GT STRUDL - Average Stresses Datasheet, display Units: Kips Feet Degrees

File Edit Columns Filter Sort Units Help

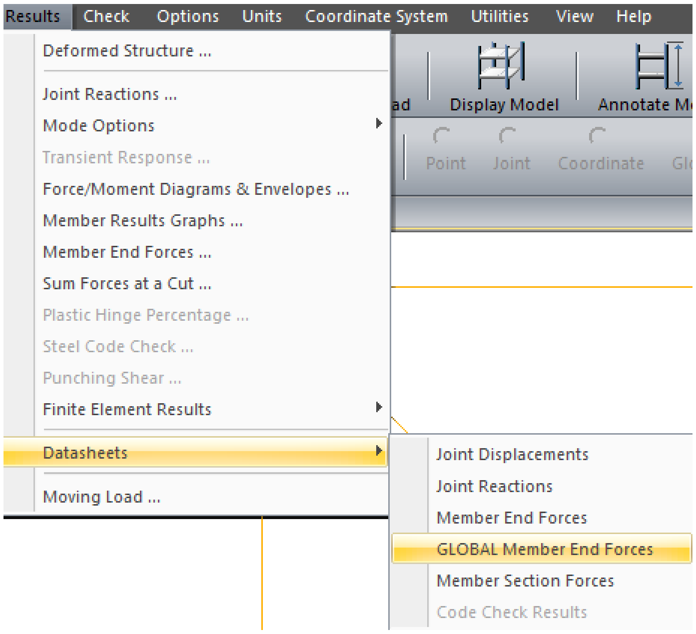
Sorted on: Not sorted Filter: None

Joint	Load	Surface	SXX	SYY	SXY	VonMises
1	1	Top	3.900	-1.560	1.800	5.784
1	1	Middle	3.900	-1.560	1.800	5.784
1	1	Bottom	3.900	-1.560	1.800	5.784
2	1	Top	-0.900	-3.000	4.800	8.731
2	1	Middle	-0.900	-3.000	4.800	8.731
2	1	Bottom	-0.900	-3.000	4.800	8.731
3	1	Top	-5.700	-4.440	7.800	14.472
3	1	Middle	-5.700	-4.440	7.800	14.472
3	1	Bottom	-5.700	-4.440	7.800	14.472
4	1	Top	4.800	1.440	-3.000	6.723
4	1	Middle	4.800	1.440	-3.000	6.723
4	1	Bottom	4.800	1.440	-3.000	6.723
5	1	Top	-0.000	0.000	-0.000	0.000
5	1	Middle	-0.000	0.000	-0.000	0.000
5	1	Bottom	-0.000	0.000	-0.000	0.000
6	1	Top	-4.800	-1.440	3.000	6.723
6	1	Middle	-4.800	-1.440	3.000	6.723

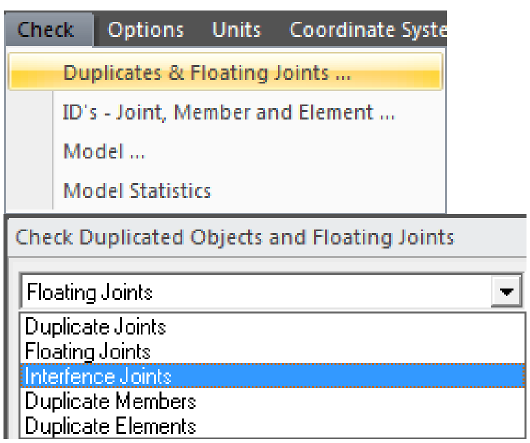
This datasheet is also now available in Command mode:



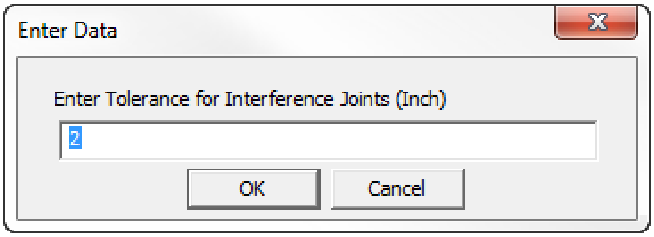
11. In addition, a new datasheet has been added to the Results –> Datasheets menu. This datasheet displays member forces (axial force, shear and moments at the start and end of the member) that have been rotated to the global reference frame. This will facilitate determining transfer forces and required connection strength. The original "Member End Forces" datasheet displays each member's forces in the local coordinate system for that member.



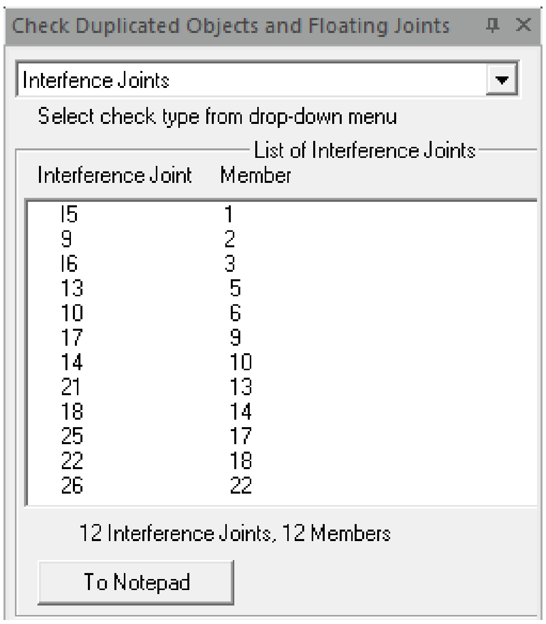
12. A new feature has been added to the Check –> Duplicates & Floating Joints... dialog: the Interference Joints check. This check is similar to the LOCATE INTERFERENCE JOINTS command (Section 2.1.12.11.1, Volume 1, GT STRUDL Reference Manual), using the same algorithm. The GTMenu version checks only the members in the current view and does not automatically create a GROUP as does the command. To access the new check, open the Duplicates & Floating Joints... dialog and from the drop-down menu choose Interference Joints:



Enter an appropriate tolerance in the current units. The tolerance sets the distance from a member's centroid to a given joint (not including the start or end joints) that will be considered an interfering joint.



If any Interference Joints are detected, the list box is filled with the pairs of the interfering joint and the member it interferes with as shown below:



The 'To Notepad' button will create a text file in your Working Directory named "InterferenceJoints_N.txt", where "N" is the view number of the current view. This file is then opened in Notepad. The file contents contain the tolerance, the view, the same joint/member list as in the dialog, plus separate lists of the joints and members, as shown below.

```
Interference Joints, tolerance 2.000 Inch
View:    2    XY Plane, along -Z axis

Joint               Member
I5                    1
9                     2
I6                    3
13                    5
10                    6
17                    9
14                    10
21                    13
18                    14
```

```
25                  17
22                  18
26                  22

Joint list:
 'I5' 9 'I6' 13 10 17 14 21 18 25 22 26

Member list:
 1 2 3 5 6 9 10 13 14 17 18 22
```

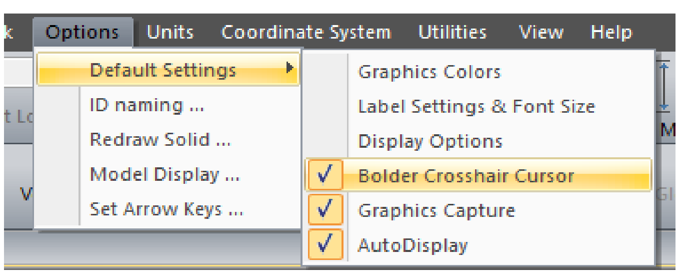
Note that the joint and member lists can be copied from this file and used to create a Group in the Create/Edit Groups dialog, using List mode.

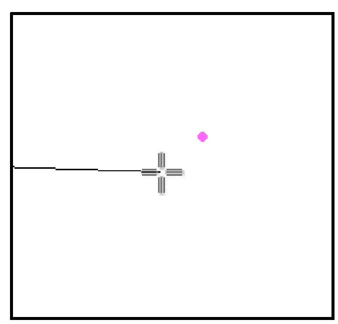
13. The GT STRUDL command text input file created by the GTMenu *File->Generate GT STRUDL/GTSELOS text input ...* menu selection now includes the SEISMIC LOAD commands for all defined seismic loads.

14. The GT STRUDL command text input file created by the GTMenu *File->Generate GT STRUDL/GTSELOS text input ...* menu selection now includes the WIND LOAD commands for all defined wind loads.

15. When creating a new load, other options are available to set applied loads to the current load. This may confuse the user into thinking that the applied loads will be added to the new load. Once a valid new load name is edited in the name Edit Box, other options are disabled until the user stores the new name.

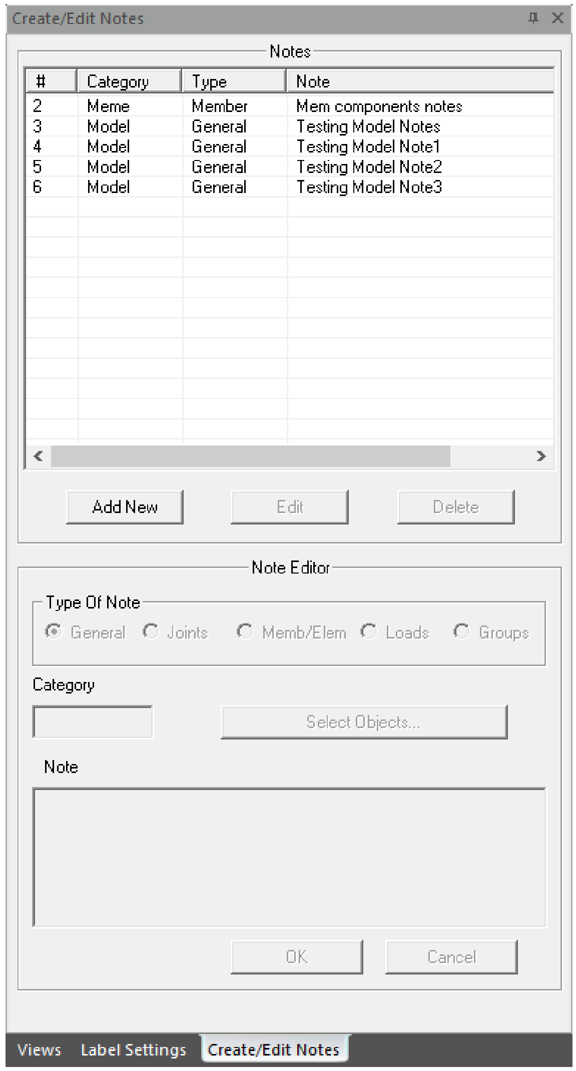
16. A new entry has been added to the *Options > Default Settings* menu to allow the use of a bolder crosshair cursor. This new entry is useful when the user has difficulty seeing the default cursor.



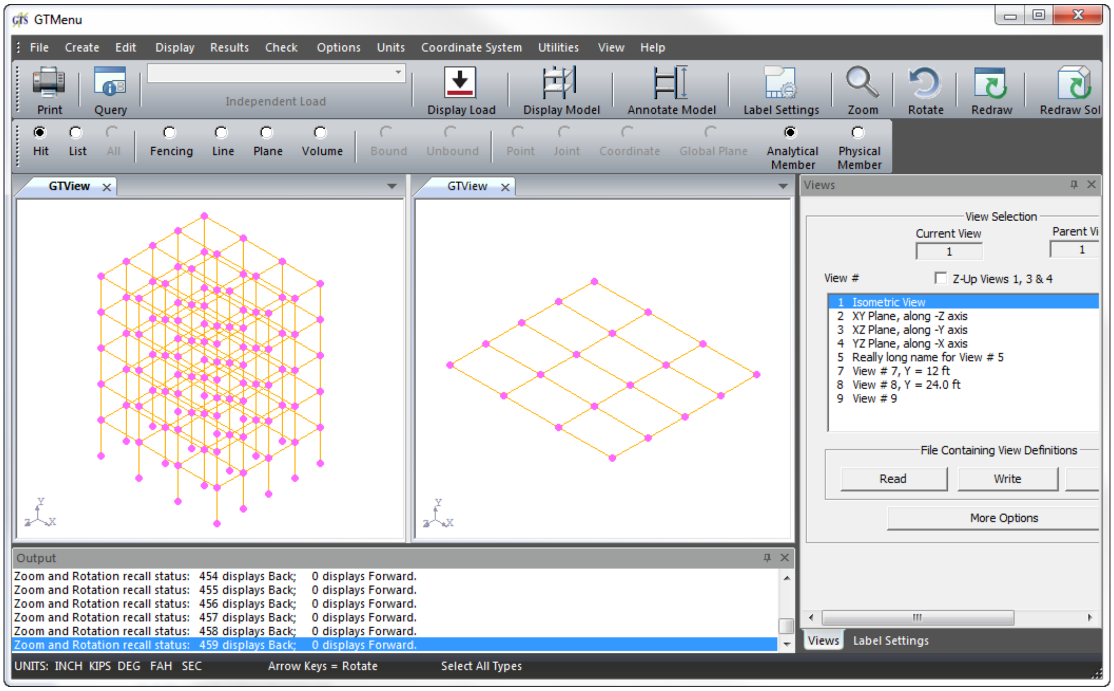


17. All of the Dialogs using Material and Member Properties have been improved to update the Materials/Properties lists, after the Create Members Properties or Create Materials Dialogs are used to create/edit Materials/Properties. Also, the last selection on the Materials and Properties lists is remembered when the Place Members, Create Elements, and Create Members dialogs are opened.

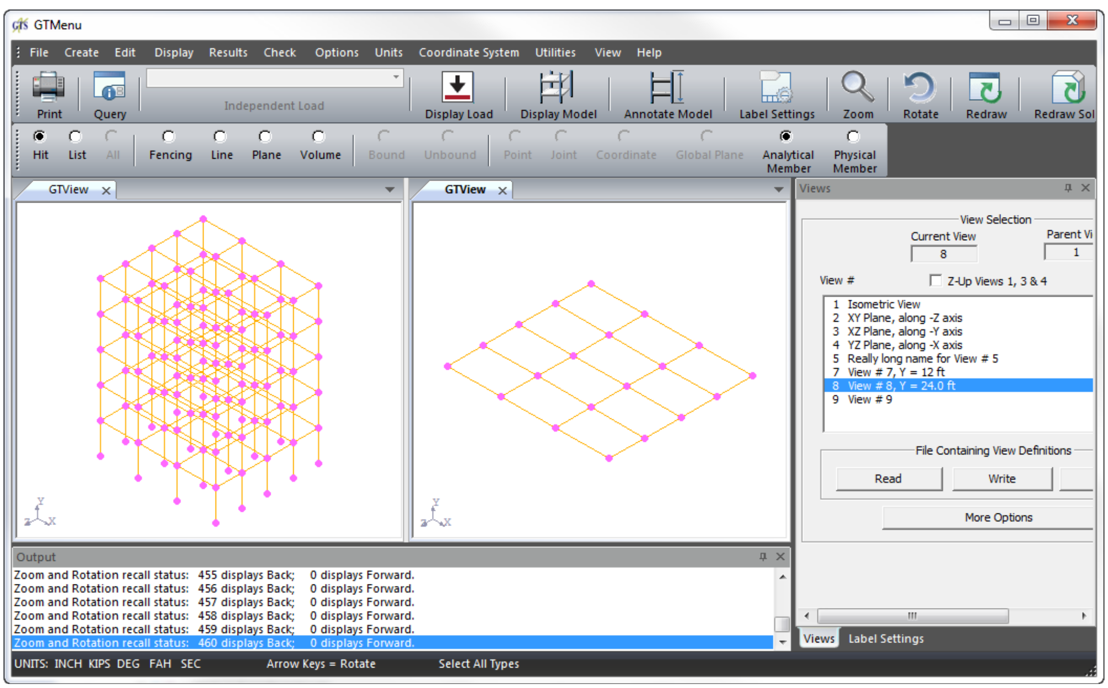
18. The Create/Edit Notes Dialog has been added. This can be opened from the correspondent menu entries in the Create and Edit menus as shown below:



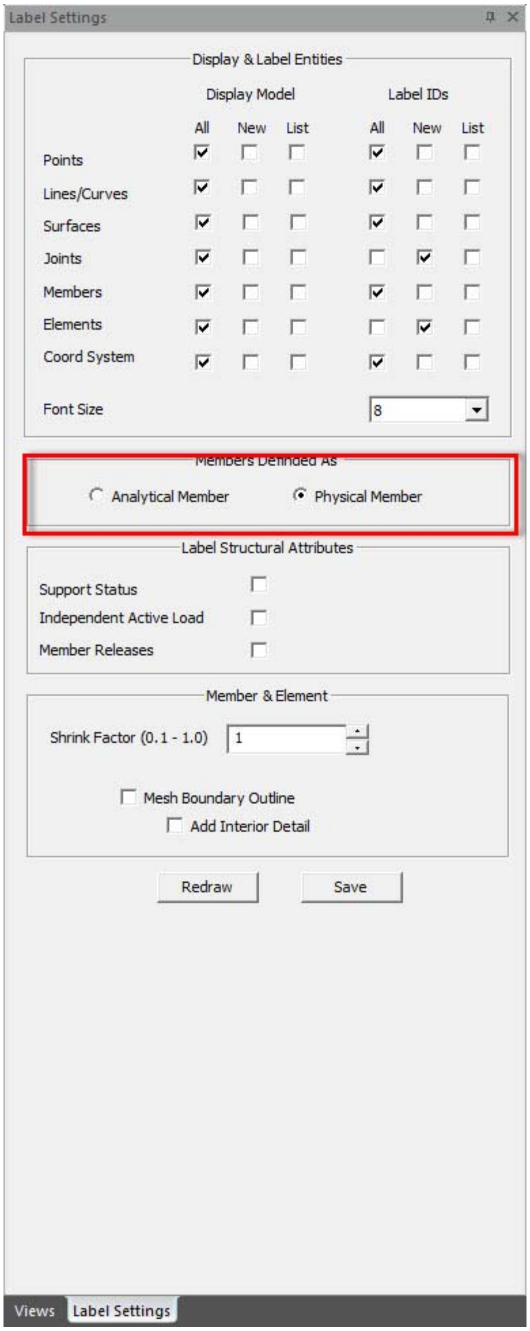
19. Changing between viewports in GTMenu will now highlight the view displayed in the current view in the Views dialog. In the figure shown on the next page, the viewport on the left is the active view and view #1 is highlighted in the Views dialog.



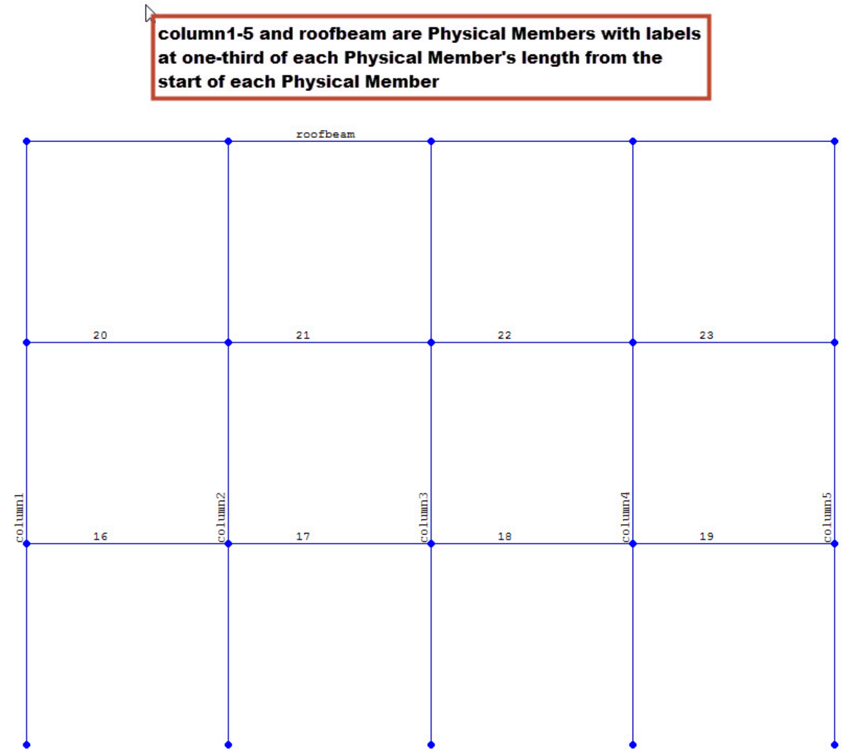
After changing the active view to the viewport on the right, view #8 is highlighted in the Views dialog as shown below:



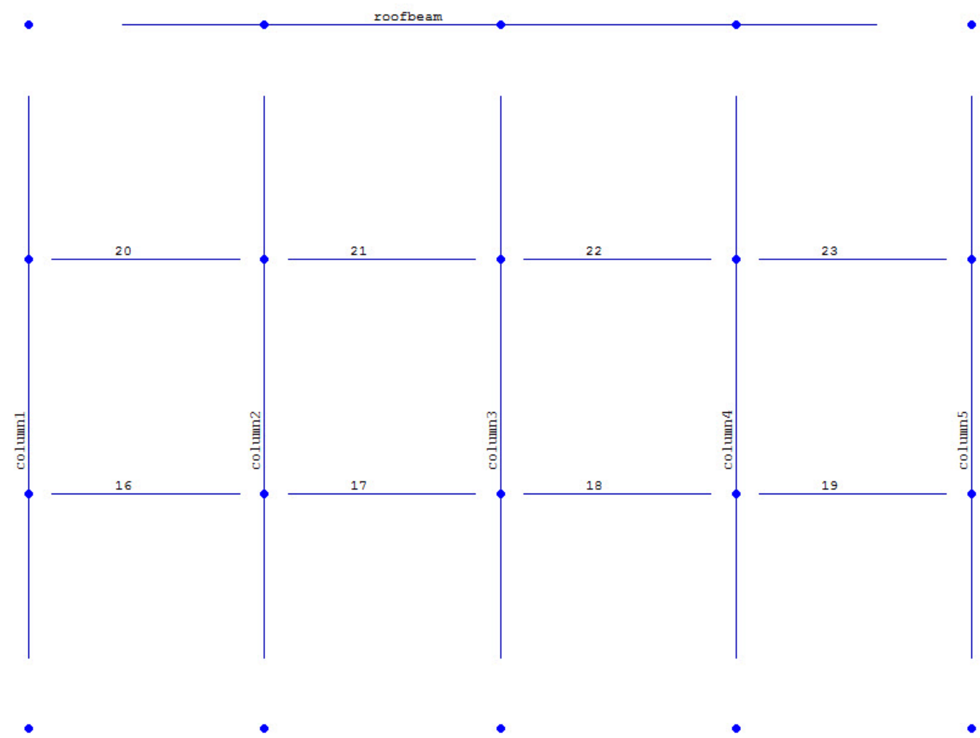
20. The Label Settings dialog now has an option where you can display and label Physical or Analytical members. If the label settings for members is set and you select the Physical Member option, the Physical members will be labeled at one third the length from the start of the Physical members. Any analytical members which are not part of a Physical member will be labeled as before. The revised Label Settings dialog is shown below:

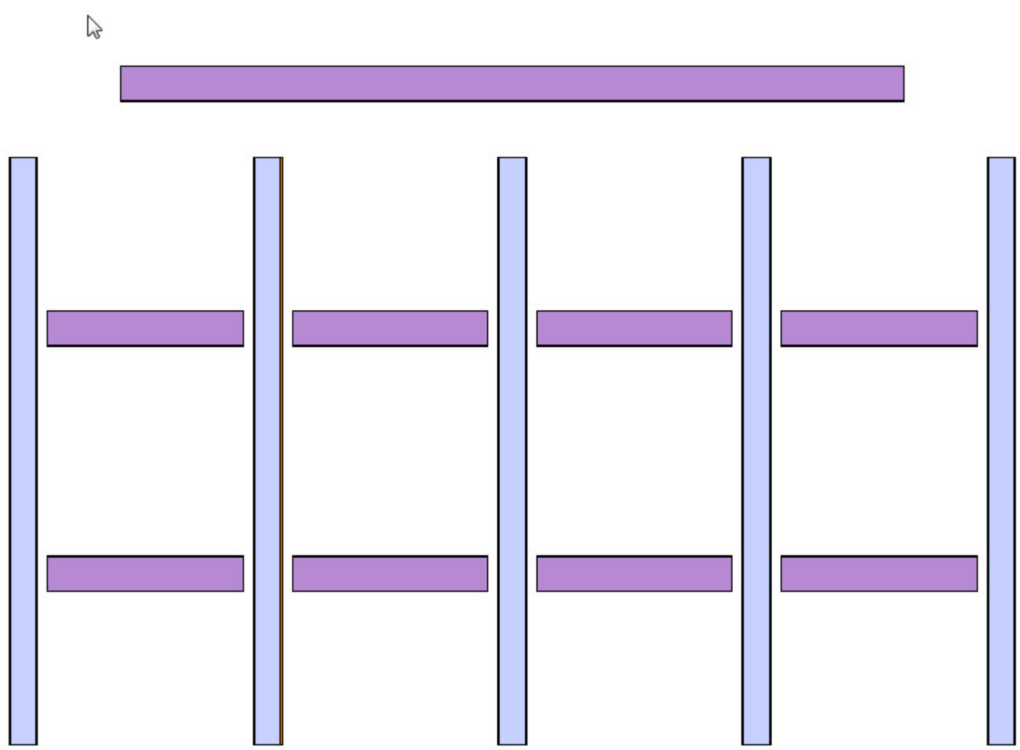


An example illustrating the labeling of Physical Members is shown below:



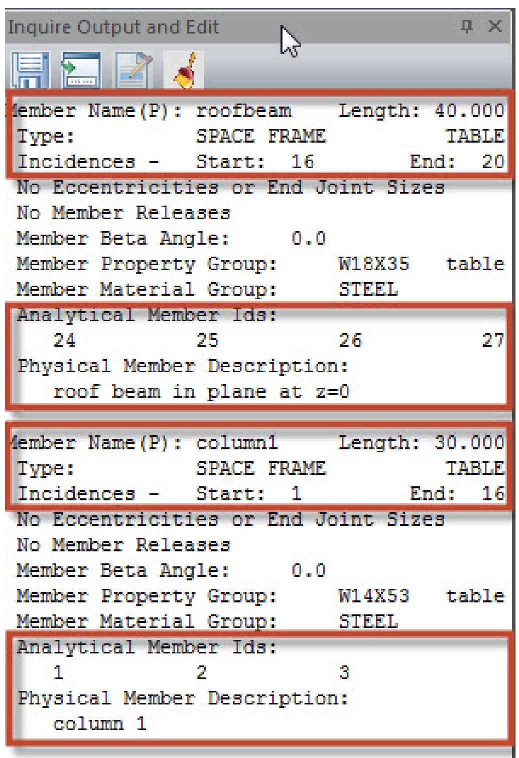
When the Physical Member option is set and you shrink the members, the Physical Members are shrunk from their start and end joints. In addition to the labels, this makes it easier to identify the Physical members. A wire frame display with shrink set to 0.8 for the above structure is shown on the next page as well as an example produced by Redraw Solid:


roofbeam
20
21
22
23
column1
column2
column3
column4
column5
16
17
18
19



21. The Mode Bar has also been enhanced to allow you to switch between Analytical and Physical Members as shown below:

22. If the Physical Member option is selected on either the Mode Bar or in the Label Settings dialog and you perform a Query on a Physical Member, the output in the Inquire Output and Edit dialog will be based on the Physical Member including the Physical Member's name, length, start and end joints, the Analytical Members included in the Physical Member, the Physical Member Description as well as other information. An example of a Query performed on the previous structure selecting the Physical Members roofbeam and column1 is shown below:

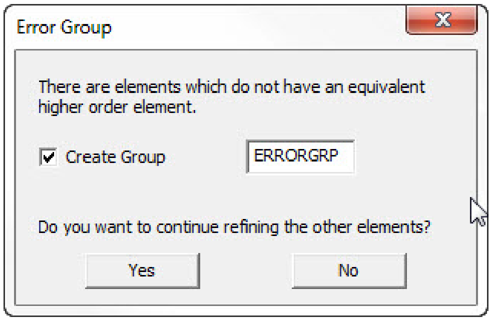


Note that there is a (P) after the Physical Member's name to make it easier to distinguish between the Query information for a Physical or Analytical member.

23. When a text input file (.gti) is created, the Group Description is now included in the Define Group commands.

24. A Redraw is now automatically performed when you delete Construction Points, Lines or Surfaces.

25. Previously when using the Refine Finite Element mesh dialog and you selected the option to Change to Higher Order Element and your model contained elements for which a higher order element was unavailable such as the six node LST triangle or the 20 node IPSQ solid, none of the elements would be changed to a higher order element. Now, the following pop-up dialog appears which allows you to create a group containing the elements for which a higher order element is unavailable. In addition, you are given the option to continue refining the mesh for elements for which a higher order element is available.



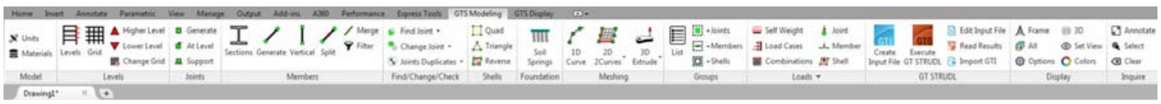
26. In all dialogs which have a List Control Box, the current selection now remains highlighted while the List Control Box is visible making it easier to see what you have selected.

27. When using the Draw Member options in the Display Model dialog shown below, the items displayed in the graphical display area are now retained as the model is manipulated by panning, zooming or rotating.

2.3 CAD Modeler

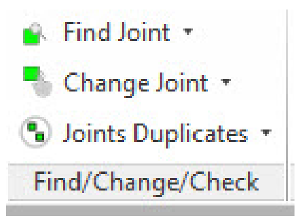
1. CAD Modeler now supports AutoCAD 2018 and BricsCAD V18 Pro and Platinum editions.

2. The GTS Modeling tab on the Ribbon Bar has been modified and includes several new features. The revised GTS Modeling Ribbon Bar is shown below:

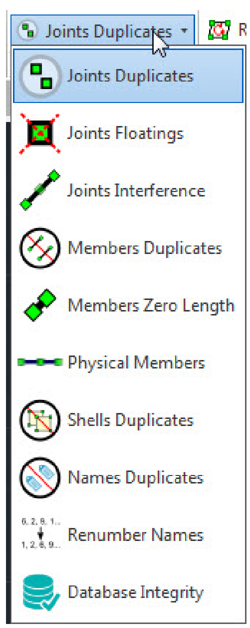


3. The following changes have been made to the GTS Modeling Ribbon Bar and to items in the Ribbon Bar:

    a. The new Find/Change/Check panel is shown below:



If you select the Joints Duplicates option, a pulldown appears as shown below where you will see the list of Checks that are available now:

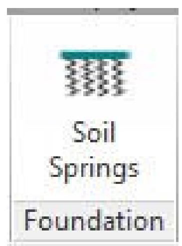


Previously, checks were only available for Duplicate and Floating Joints. Additional checks have been added which allow you to check for Interference Joints, Duplicate Members and Shells and Duplicate Names (IDs).

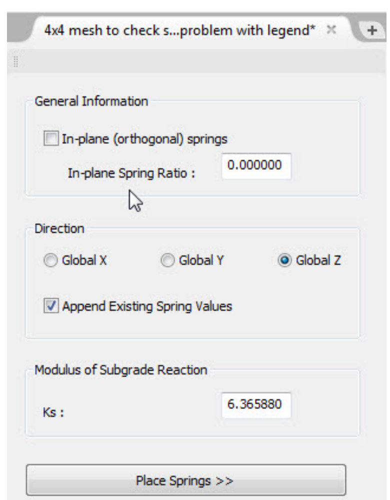
You can check for Zero Length Members and also check Physical Members to verify that they follow the rules for Physical Members as presented in Section 2.1.2.10.1 of Volume 1 of the Reference Manual. In addition, you can Renumber Names (IDs) and check the Database Integrity.

These checks are further described in Sections 2.5.20- 2.5.29 in the CAD Modeler Getting Started Guide.

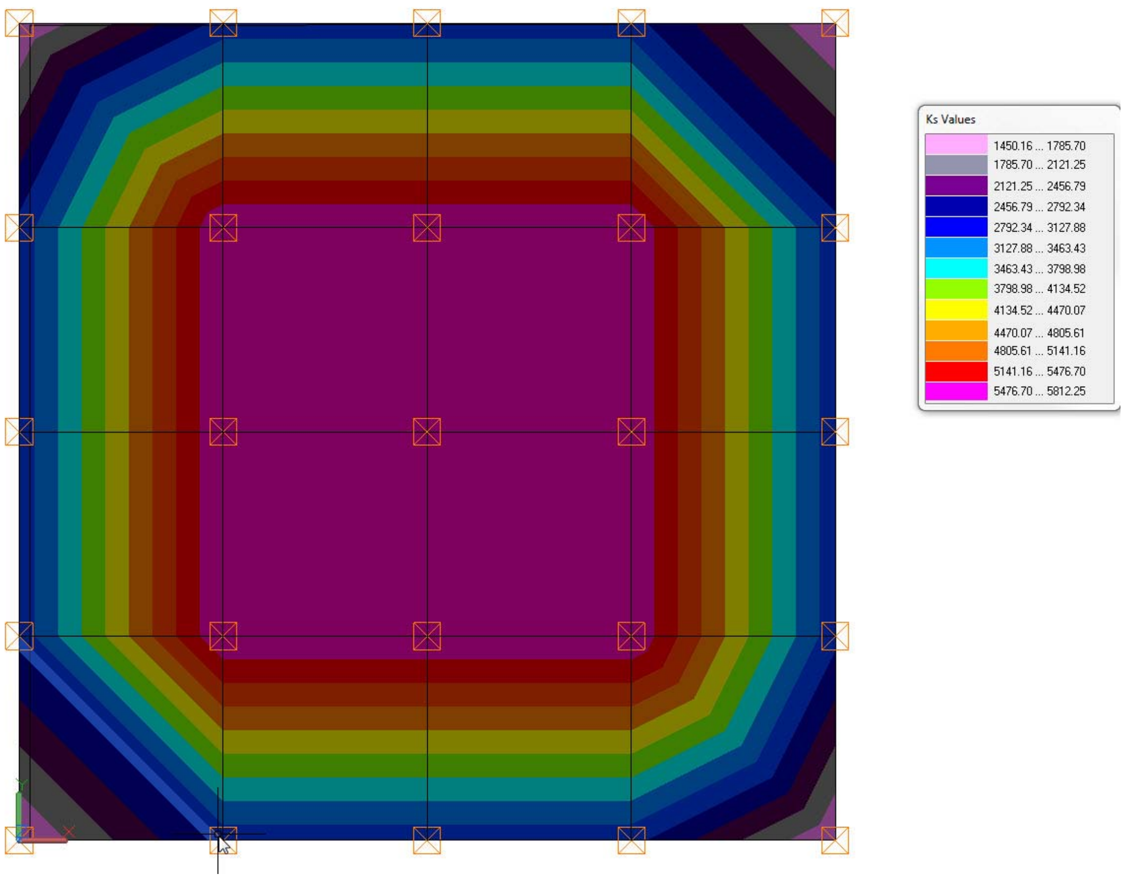
b. A new Foundation panel has been added which allows you to create elastic springs under a finite element mesh to model elastic soil behavior on structures such as slabs and mats. The new Foundation panel is shown below:



After selecting Soil Springs, the dialog shown below appears where you create the springs to represent the soil by specifying the global direction of the springs and the modulus of subgrade reaction. You may also specify in-plane springs which are a ratio of the normal springs to the elements. In addition, you have the option to append the soil springs to other springs that previously existed.



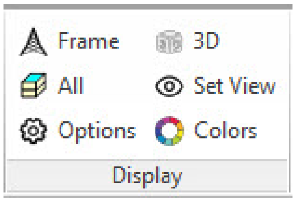
Upon selecting the Place Springs button in the dialog and select the elements, the spring values are computed and a contour plot with a legend is created to aid in the checking of the spring values assigned to each node in the direction of the soil support. The following figure illustrates the results of the application of soil springs to a simple 4x4 finite element mesh:



c. An Import GTI option has been added to the GT STRUDL panel as shown below:

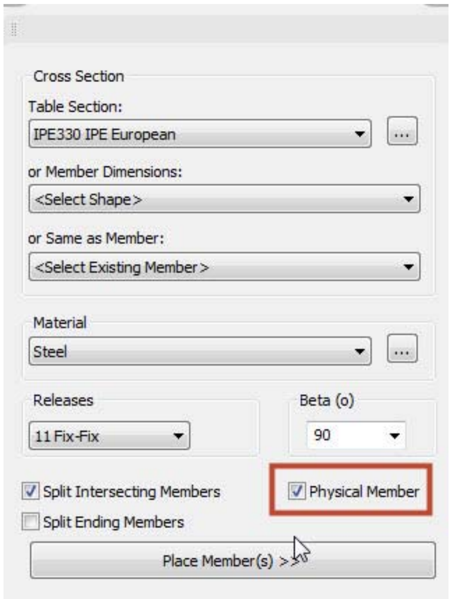
Previously, this option was only available from the GTS Modeling Menu bar pulldown.

d. The Display panel has been reorganized as shown below:

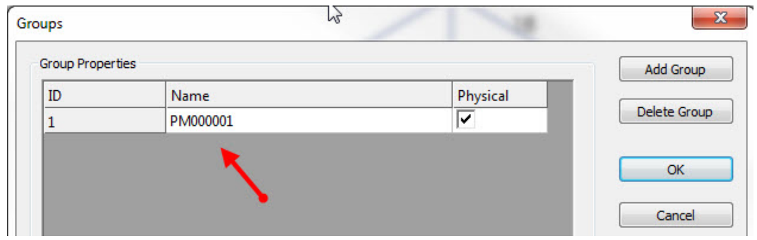


e. Other enhancements to items in the GTS Modeling tab which are not readily apparent from the Ribbon Bar are described below:

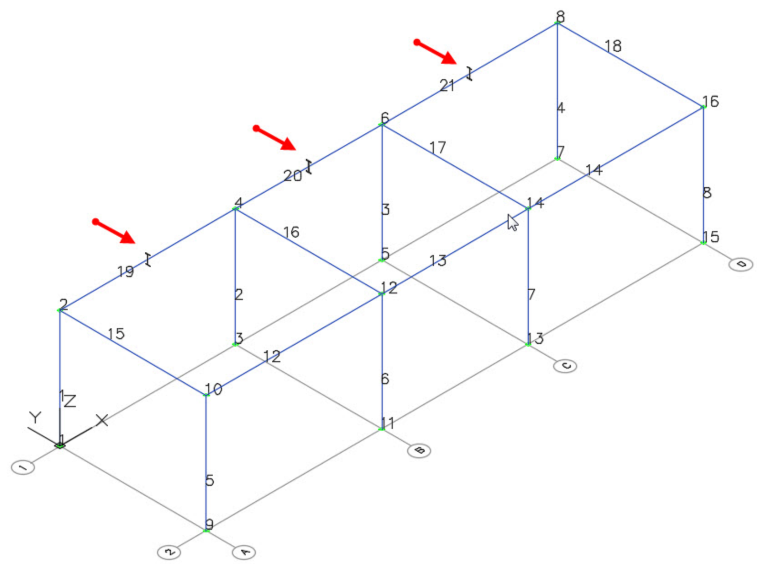
i. When Placing Members using the Generate option in the Members panel, a Physical Member group may now be automatically created. The new Physical Member group will have the name PMxxxxxx where xxxxxx is an integer beginning with 000001. This new option is highlighted in the Place Members dialog shown below:



After placing members with the Physical Member option checked, the Physical Member group names can be seen by using the List option in the Groups panel. An example of the listing after placing one Physical Member is shown on the next page:



ii. An icon showing the orientation of the cross section is now displayed when placing members. This new feature makes it easier to verify the orientation of the members. An example is shown below:

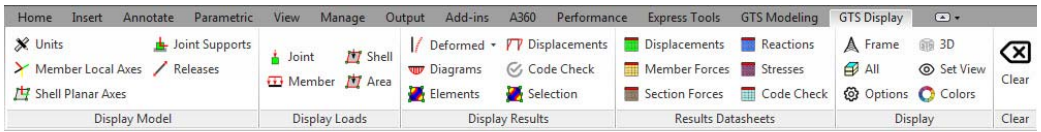


The icons can be erased by selecting the Clear option in the Inquire panel.

iii. The names for loads and load combinations are now checked for duplicate names.

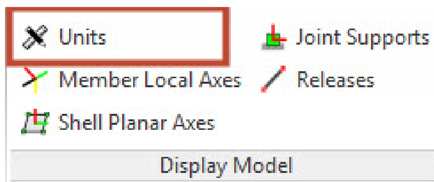
iv. When creating a gti file, the Section Force command will now have the word Automatic added to it so additional sections are computed for force and moment diagrams and envelopes. Further information on this may be found in Section 2.6 DBX.

4. The GTS Display tab on the Ribbon Bar has been modified and includes a number of new features. The revised GTS Display Ribbon Bar is shown below:

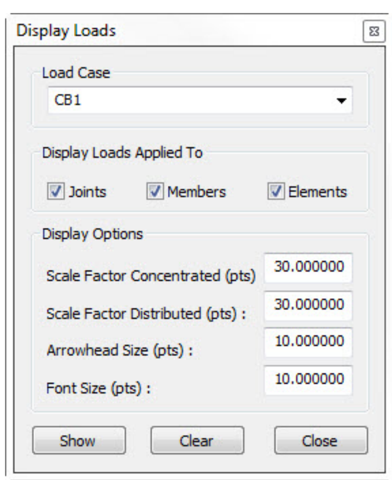


5. The following changes have made to the GTS Display Ribbon Bar and to items in the Ribbon Bar:

a. A Units option has been added to the Display Model panel as shown below which will make it easier to change the units while displaying various attributes of the model such as loads and displaying results:

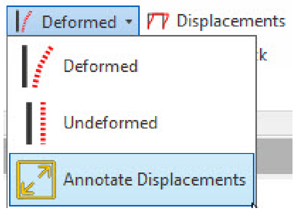


b. You may now display loads in Loading Combinations and Form Loadings. Select either the Joint, Member or Shell options in the Display Loads panel and the dialog shown below appears where you can select a Loading Combination or Form Load and then select the types of loads to display:

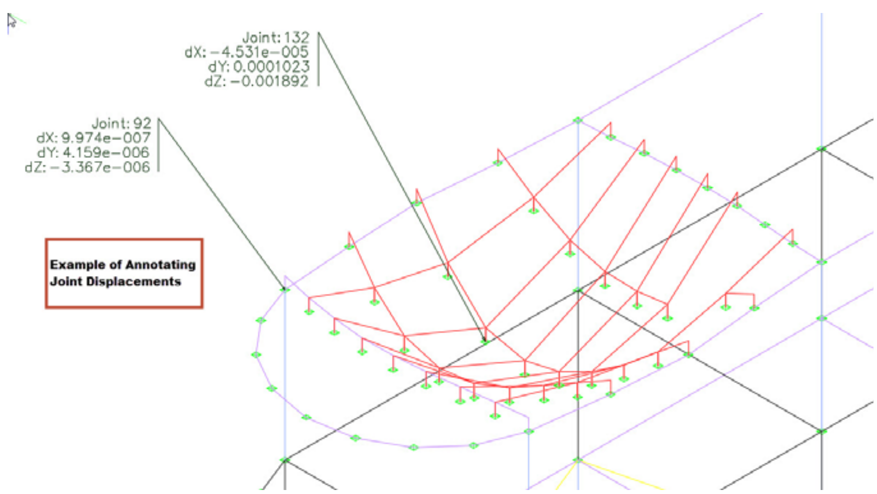


In the dialog above, Load Case CB1 is a loading combination and loads applied to Joints, Members and Elements will be displayed when the Show button is selected.

c. After selecting the Deformed option in the Display Results panel, you may now annotate the joint displacements on the display of the deformed structure. Click on the down arrow by Deformed and the following dropdown menu will appear where you may select Annotate Displacements as shown below:

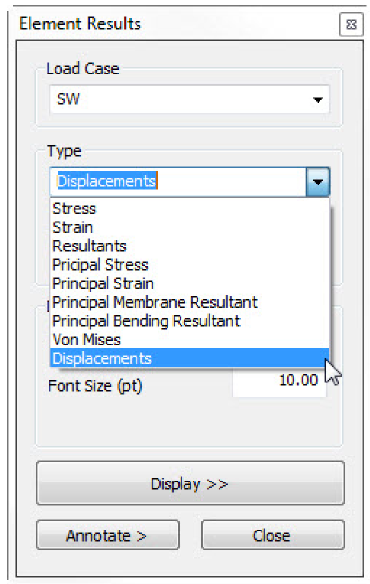


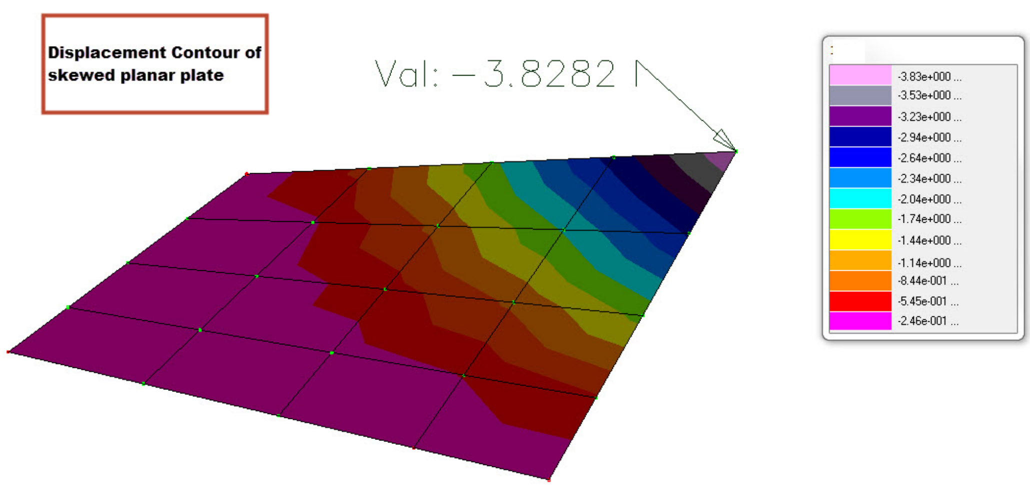
You may then click on a joint and then click on a placement location for the joint and joint displacements. An example is shown below:



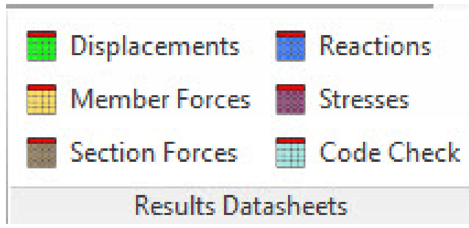
d. The Elements option in the Display Results panel now includes the ability to contour displacement results on a finite element model. The contouring of displacement results on some finite element models is easier to interpret than looking at the deformed shape of the structure.

The revised Element Results dialog showing the Displacement option and an example of the displacement contour of a skewed flat plate is shown on the next page:


Element Results
Load Case
SW
Type
Displacements
Stress
Strain
Resultants
Pricipal Stress
Principal Strain
Principal Membrane Resultant
Principal Bending Resultant
Von Mises
Displacements
Font Size (pt)
10.00
Display >>
Annotate >
Close

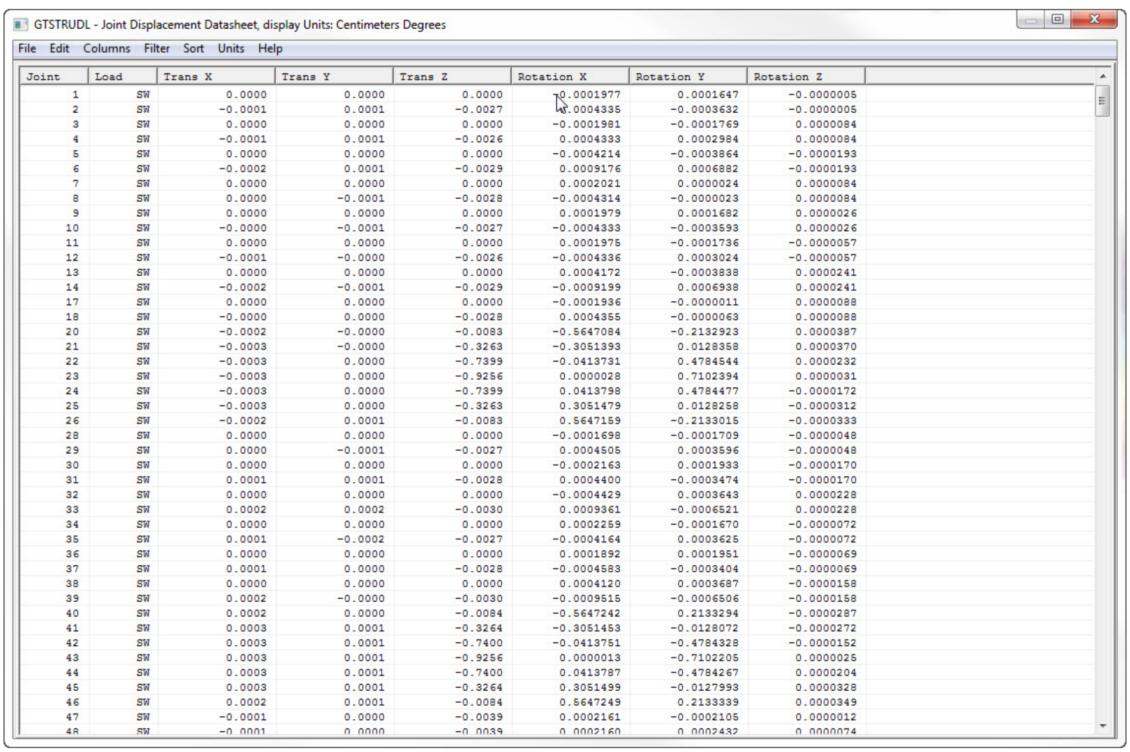

Displacement Contour of skewed planar plate
Val: −3.8282
-3.83e+000 ...
-3.53e+000 ...
-3.23e+000 ...
-2.94e+000 ...
-2.64e+000 ...
-2.34e+000 ...
-2.04e+000 ...
-1.74e+000 ...
-1.44e+000 ...
-1.14e+000 ...
-8.44e-001 ...
-5.45e-001 ...
-2.46e-001 ...

e. A new Results Datasheets panel has been added as shown below:



By selecting one of the options in the Results Datasheets panel, a datasheet of the selected results will appear. You can perform a variety of functions such as changing the units for the values in the datasheets, sort and filter the items and write the results to a text file. The datasheets are essentially the same as those in GTMenu and GTShell. Help is available on the Menubar of the datasheet.

An example of the Displacements datasheet is shown below:



GTSTRUDL - Joint Displacement Datasheet, display Units: Centimeters Degrees

File Edit Columns Filter Sort Units Help

Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y	Rotation Z
1	SW	0.0000	0.0000	0.0000	-0.0001977	0.0001647	-0.0000005
2	SW	-0.0001	0.0001	-0.0027	[illegible].0004335	-0.0003632	-0.0000005
3	SW	0.0000	0.0000	0.0000	-0.0001981	-0.0001769	0.0000084
4	SW	-0.0001	0.0001	-0.0026	0.0004333	0.0002984	0.0000084
5	SW	0.0000	0.0000	0.0000	-0.0004214	-0.0003864	-0.0000193
6	SW	-0.0002	0.0001	-0.0029	0.0009176	0.0006882	-0.0000193
7	SW	0.0000	0.0000	0.0000	0.0002021	0.0000024	0.0000084
8	SW	0.0000	-0.0001	-0.0028	-0.0004314	-0.0000023	0.0000084
9	SW	0.0000	0.0000	0.0000	0.0001979	0.0001682	0.0000026
10	SW	-0.0000	-0.0001	-0.0027	-0.0004333	-0.0003593	0.0000026
11	SW	0.0000	0.0000	0.0000	0.0001975	-0.0001736	-0.0000057
12	SW	-0.0001	-0.0000	-0.0026	-0.0004336	0.0003024	-0.0000057
13	SW	0.0000	0.0000	0.0000	0.0004172	-0.0003838	0.0000241
14	SW	-0.0002	-0.0001	-0.0029	-0.0009199	0.0006938	0.0000241
17	SW	0.0000	0.0000	0.0000	-0.0001936	-0.0000011	0.0000088
18	SW	-0.0000	0.0000	-0.0028	0.0004355	-0.0000063	0.0000088
20	SW	-0.0002	-0.0000	-0.0083	-0.5647084	-0.2132923	0.0000387
21	SW	-0.0003	-0.0000	-0.3263	-0.3051393	0.0128358	0.0000370
22	SW	-0.0003	0.0000	-0.7399	-0.0413731	0.4784544	0.0000232
23	SW	-0.0003	0.0000	-0.9256	0.0000028	0.7102394	0.0000031
24	SW	-0.0003	0.0000	-0.7399	0.0413798	0.4784477	-0.0000172
25	SW	-0.0003	0.0000	-0.3263	0.3051479	0.0128258	-0.0000312
26	SW	-0.0002	0.0001	-0.0083	0.5647159	-0.2133015	-0.0000333
28	SW	0.0000	0.0000	0.0000	-0.0001698	-0.0001709	-0.0000048
29	SW	0.0000	-0.0001	-0.0027	0.0004505	0.0003596	-0.0000048
30	SW	0.0000	0.0000	0.0000	-0.0002163	0.0001933	-0.0000170
31	SW	0.0001	0.0001	-0.0028	0.0004400	-0.0003474	-0.0000170
32	SW	0.0000	0.0000	0.0000	-0.0004429	0.0003643	0.0000228
33	SW	0.0002	0.0002	-0.0030	0.0009361	-0.0006521	0.0000228
34	SW	0.0000	0.0000	0.0000	0.0002259	-0.0001670	-0.0000072
35	SW	0.0001	-0.0002	-0.0027	-0.0004164	0.0003625	-0.0000072
36	SW	0.0000	0.0000	0.0000	0.0001892	0.0001951	-0.0000069
37	SW	0.0001	0.0000	-0.0028	-0.0004583	-0.0003404	-0.0000069
38	SW	0.0000	0.0000	0.0000	0.0004120	0.0003687	-0.0000158
39	SW	0.0002	-0.0000	-0.0030	-0.0009515	-0.0006506	-0.0000158
40	SW	0.0002	0.0000	-0.0084	-0.5647242	0.2133294	-0.0000287
41	SW	0.0003	0.0001	-0.3264	-0.3051453	-0.0128072	-0.0000272
42	SW	0.0003	0.0001	-0.7400	-0.0413751	-0.4784328	-0.0000152
43	SW	0.0003	0.0001	-0.9256	0.0000013	-0.7102205	0.0000025
44	SW	0.0003	0.0001	-0.7400	0.0413787	-0.4784267	0.0000204
45	SW	0.0003	0.0001	-0.3264	0.3051499	-0.0127993	0.0000328
46	SW	0.0002	0.0001	-0.0084	0.5647249	0.2133339	0.0000349
47	SW	-0.0001	0.0000	-0.0039	0.0002161	-0.0002105	0.0000012
48	SW	-0.0001	0.0000	-0.0039	0.0002160	0.0002432	0.0000074

2.4 Nonlinear Analysis

1. Nonlinear static analysis has been improved with the addition of a GT64M, 64-bit version of the high-performance nonlinear static analysis solver, which is executed when the ACTIVE SOLVER GT64M command has been specified prior to the specification of the NONLINEAR ANALYSIS command. In conjunction with this addition, a more efficient version of the GTSES high performance nonlinear static analysis solver also has been implemented. The new GT64M and updated GTSES nonlinear static analysis solvers have demonstrated overall nonlinear static analysis time improvements of up to 50 percent over the original GTSES nonlinear analysis solver. This implementation also applies to the PERFORM PUSHOVER ANALYSIS and PERFORM CABLE PRESTRESS ANALYSIS commands, both of which incorporate the general nonlinear static analysis function.

2. The GTSES option has been removed from the NONLINEAR ANALYSIS, PERFORM PUSHOVER ANALYSIS, and PERFORM CABLE PRESTRESS ANALYSIS commands. In previous versions, these commands included the specification of the GTSES option as follows:

   NONLINEAR ANALYSIS GTSES

   PERFORM PUSHOVER ANALYSIS GTSES

   PERFORM CABLE PRESTRESS ANALYSIS GTSES

   which executed the GTSES 32-bit high performance nonlinear static analysis solver. However, as of Version 2018, the use of the GTSES option in these commands is treated as an error condition with the following error message being issued:

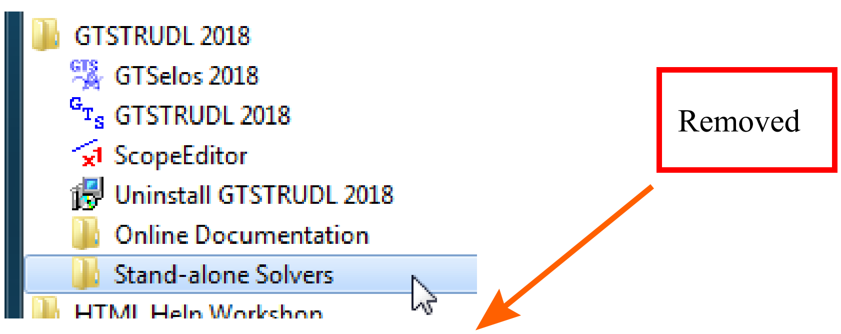
```
**** ERROR_NONL -- The GTSES option has been removed.  Use the ACTIVE
         SOLVER GTSES command to select the GTSES high-
         performance solver.
         Scan mode entered and analysis terminated.
```

   causing the specified analysis to be terminated and scan mode to be entered, which inhibits further processing. Version 2018 requires that if you intend these commands to execute either the GTSES or GT64M nonlinear static analysis solvers, the ACTIVE SOLVER GTSES/GT64M command must be given prior to any of these commands.

2.5 GTSES/GT64M High Performance Solvers

1. In previous versions of GT STRUDL the execution of linear and nonlinear static and dynamic analyses using the GTSES and GT64M high performance sparse solvers stored the analysis results in files under an automatically created GT STRUDL working directory subfolder named GTSresults/. As of Version 2018, the format for the name of this folder is changed to GTSresultsZZZZZZZZZ/, where ZZZZZZZZZ is a nine-digit code that represents the date and time when the folder was created during the execution of a job. Furthermore, the GTSresults ZZZZZZZZZ/ folder created by a job that is SAVEd is used when the job is later restored. Likewise, a GTSresults/ folder that is created by a job that was SAVEd under pre-2018 versions also is used when that job is restored under Version 2018.

   Note that as a consequence of this feature modification, the GTSES/GT64M static analysis split solver procedure described in earlier revisions of Section 2.1.13.3.2 is no longer applicable; therefore, in Revision 2018, this section has been removed. Furthermore, the Stand-alone Solvers option has been removed from the GT STRUDL 2018 Startup menu:



<u>Documentation</u>:

The ACTIVE SOLVER Command, Volume 1, Section 2.1.13.4

Split Solver Static Analysis Using the GTSES/GT64M Sparse Equation Solvers, Volume 1, Section 2.1.13.3.2, Revision 2017 and earlier.

2.6 DBX

1. The WRITE SECTION FORCES command has been improved to include an AUTOMATIC option. The AUTOMATIC option will add sections as needed at concentrated loads plus sections for the maximums and minimums for each degree of freedom. The maximum section count is limited to 50 or less. For each member, the added sections are determined for all active loads and the same number of sections are used for all loads.

   SECTION FORCES ([NS] i (AUTOMATIC))

   Documentation:

   Section 2. The WRITE Command, Volume 5, GT STRUDL Reference Manual

2. The WRITE MEMBER RESULTS command now accepts the GLOBAL option. This option will rotate member forces, start and end, to the global reference frame instead of the default local coordinate system reference frame. This option allows for easier determination of transfer and connection forces.

   WRITE MEMBER RESULTS (GLOBAL)

   Documentation:

   Section 2. The WRITE Command, Volume 5, GT STRUDL Reference Manual

2.7 Interfaces

1. The STAAD.Pro-to-GTSTRUDL interface ST32GTS has been improved to write JOINT COORDINATES with 4 decimal places instead of 2. This will allow a more accurate translation when the units are METERS.

2.8 Loadings

1. The optional SEISMIC DEAD LOAD specification has been added to the SEISMIC LOAD commands as highlighted in the General Structure Data command syntax below:

$$
* \left\{ \begin{array}{l}
\underline{\text{HEI}}\text{GHT } (\underline{\text{AXIS}}) \left\{ \begin{array}{c} \underline{\text{Y}} \\ \underline{\text{Z}} \end{array} \right\} \\[1em]
\underline{\text{DIR}}\text{ECTION } \left\{ \begin{array}{c} \underline{\text{X}} \\ \underline{\text{Y}} \\ \underline{\text{Z}} \end{array} \right\} \\[1em]
\underline{\text{SEIS}}\text{MIC } \underline{\text{WEI}}\text{GHT } \underline{\text{LOA}}\text{D } \left\{ \begin{array}{l} i_{sw} \\ 'a_{sw}' \end{array} \right\} \\[1em]
\mathbf{(\underline{SEIS}MIC\ \underline{DEA}D\ \underline{LOA}D} \left\{ \begin{array}{l} \mathbf{i_{DL}} \\ \mathbf{'a_{DL}'} \end{array} \right\} \mathbf{)} \\[1em]
\underline{\text{STO}}\text{RY } (\underline{\text{HEI}}\text{GHTS}) \left\{ \begin{array}{l} \textit{list of heights} \\ \underline{\text{JOI}}\text{NTS } \textit{list of joints} \end{array} \right\} (\underline{\text{TOL}}\text{ERANCE } v_{FTOL}) \\[1em]
(\underline{\text{FLO}}\text{OR } \underline{\text{TOL}}\text{ERANCE } v_{FTOL})
\end{array} \right\}
$$

The SEISMIC DEAD LOAD option specifies the previously-defined independent loading condition from which the vertical seismic joint load components are computed for all joints according to Section 12.4.2.2 of ASCE 7-05 and 7-10.

Documentation:

Section 2.1.11.3.10, Volume 1, GT STRUDL Reference Manual

2. The Additions, Deletions and Changes modes for the AREA LOAD Command have been improved as follows:

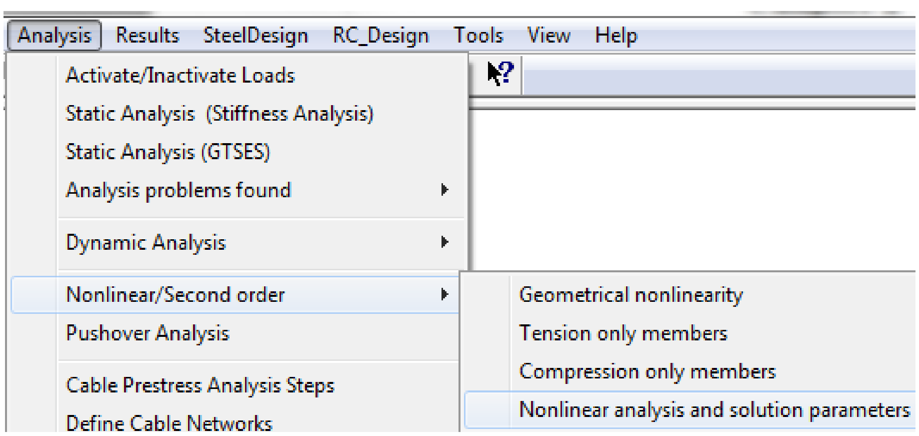
   a. Additions.
      - When there is no valid Elevation Data, and the Area Load command is ended, the command will be ignored and the Load is not created.
      - When an Area Load is created, it is no longer left as the active load. It will prevent users from assigning individual applied loads to an Area Load.
      - When the Area load command is terminated by starting another command rather than the "END AREA LOAD" statement, a warning is issued.

b. Deletions
- If the AREA LOAD command is issued in DELETIONS mode and the loads in the list are not Area Loads, these loads will not be deleted.

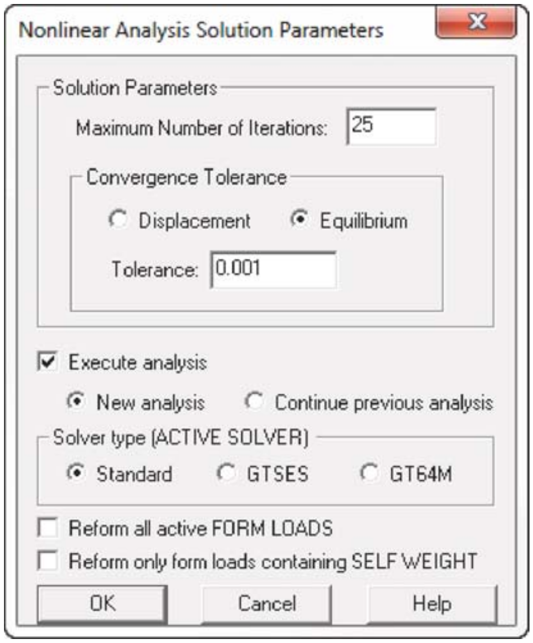
c. Changes
- If the AREA LOAD Command is issued while in CHANGES mode, the entire definition of the area load will be replaced. The user could take advantage of the "PRINT AREA LOAD PARAMETERS" command in order to get the current definition, and perform the required changes.
- When there is no valid Elevation Data, and the Area Load command is ended, the command will be ignored and the load is not changed.

2.9 GTShell (GT STRUDL Output Window)

1. The Nonlinear analysis solutions parameters dialog has been enhanced to include options for the solver type and Reform options. In addition, the default number of iterations has been increased to 25 (from 1) and the Tolerance changed to 0.001 (from 0.01). The Nonlinear Analysis Solutions Parameters dialog is reached from the Analysis menu:



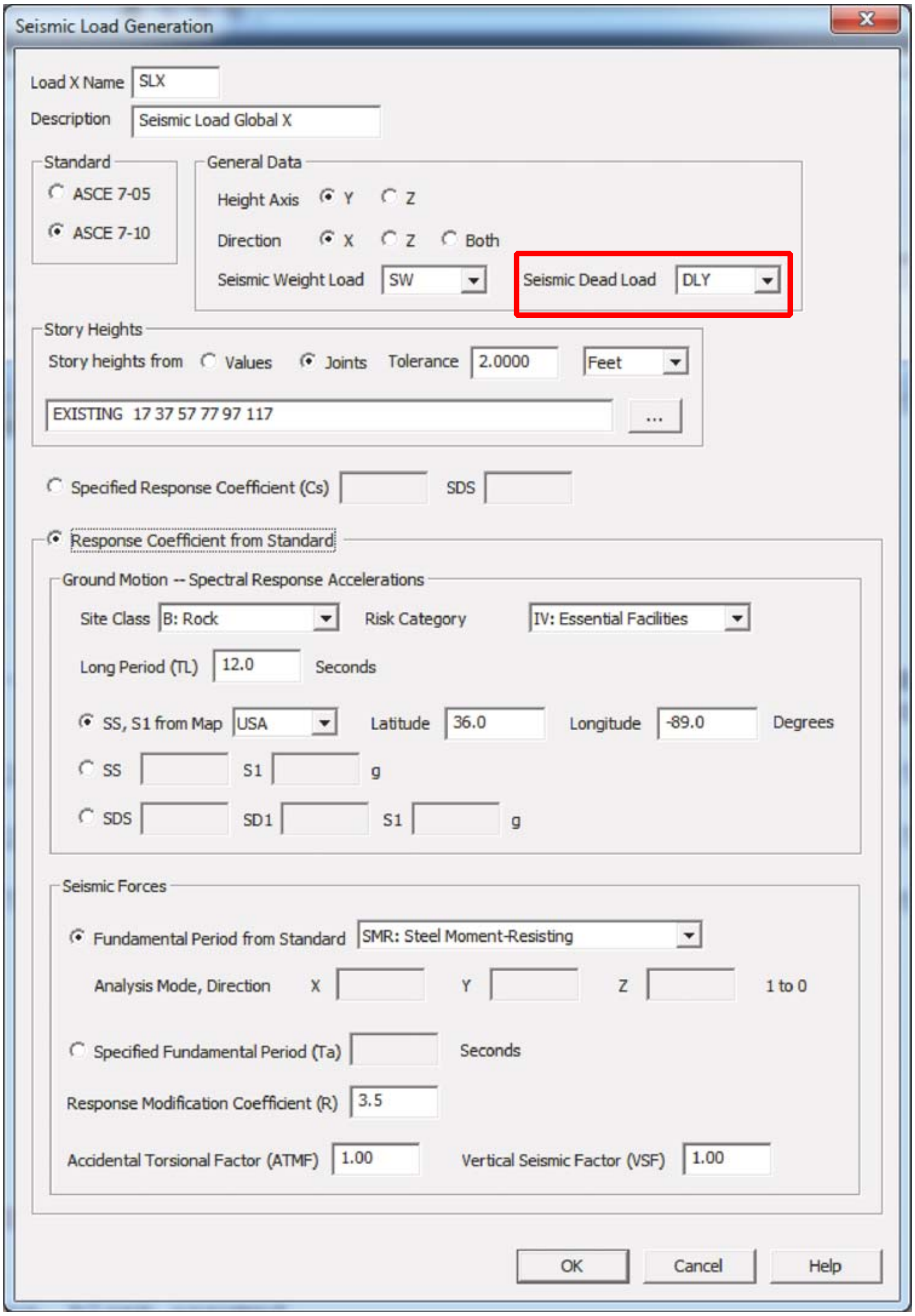
The revised dialog is shown on the next page:



Documentation:
Section 2.5 "Nonlinear Static Analysis", Volume 3, GT STRUDL Reference Manual

Section 2.1.11.3.2 "Independent FORM LOADING Command", Volume 1, GT STRUDL Reference Manual
Section 2.1.13.4 "The ACTIVE SOLVER Command", Volume 1, GT STRUDL Reference Manual

2. The Seismic Load dialog has been improved with the addition of the Seismic Dead Load control as highlighted by the red box below:

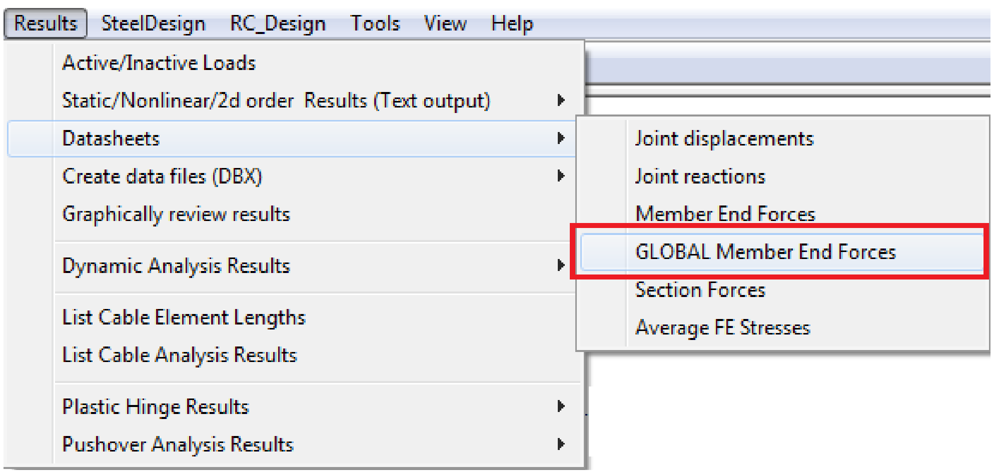


The Seismic Dead Load control permits the selection of an active or inactive independent load as the source of the applied force load components to be used for the computation of the vertical joint load components of the generated seismic load.

Documentation:

The SEISMIC LOAD Commands, Section 2.1.11.3.10, Volume 1, GT STRUDL Reference Manual.

3. A new datasheet has been added to the Results –> Datasheets menu. This datasheet displays member forces (axial force, shear and moments at the start and end of the member) that have been rotated to the global reference frame. This will facilitate determining transfer forces and required connection strength. The original "Member End Forces" datasheet displays each member's forces in the local coordinate system for that member.

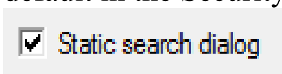


4. The GT STRUDL command text input file created by the GTShell *File->Create a NEW text input file* menu selection now includes the SEISMIC LOAD commands for all defined seismic loads.

5. The GT STRUDL command text input file created by the GTShell *File->Create a NEW text input file* menu selection now includes the WIND LOAD commands for all defined wind loads.

2.10 Installation

1. During installation of version 2018, the 'Static search dialog' option will be set by default in the Security Devices tab of the Options page as shown below:



Previously, this option was not set during installation. By setting this option, locating the security device on some networks has found to be improved.

For more information, select the Help button on the Security Devices tab.

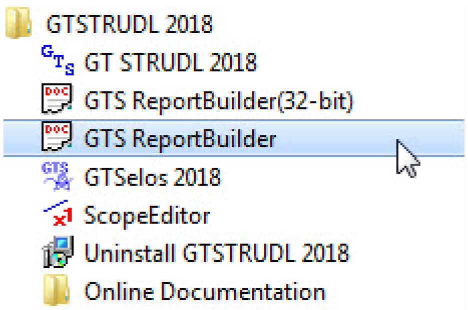
2.11 Base Plate Wizard

1. 'File –> Save to .gtbp' will now group anchor data other than coordinates into blocks of similar values when creating Base Plate Wizard .gtbp files. This will result in shorter and easier-to-read files.

2.12 Report Builder

GT STRUDL Report Builder is self-contained software, which generates reports graphically from GT STRUDL DBX database files. In order to use Report Builder, GT STRUDL 2018 must be installed on the computer and be running.

To open Report Builder, go to the Start button, select All Programs and then select GT STRUDL 2018 and the drop down menu will show two options for Report Builder as shown below:

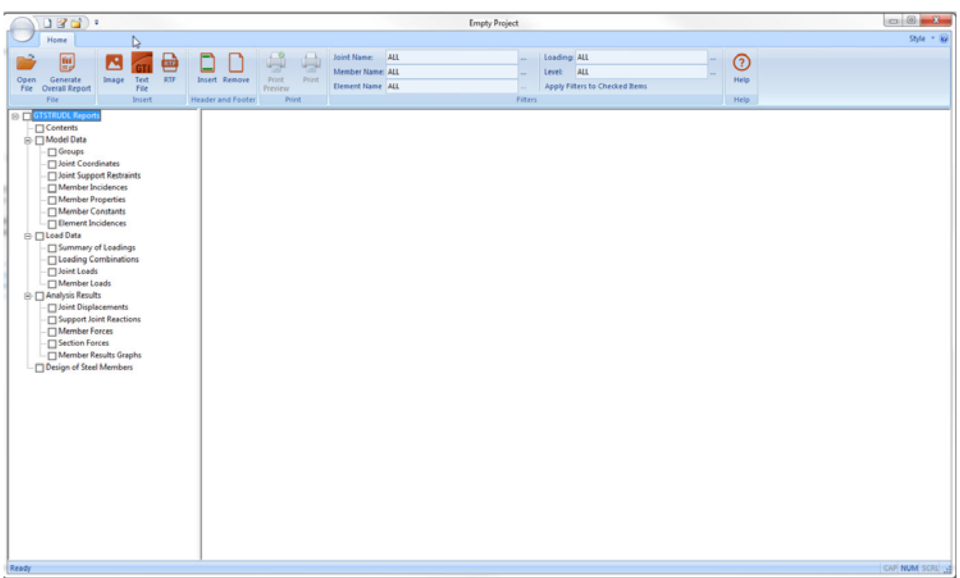


The main window of Report Builder is shown below and consists of three panes:

a) the ribbon including all the available functions

b) the data hierarchy tree which shows all the available items with data that can be loaded from a GT STRUDL database, and

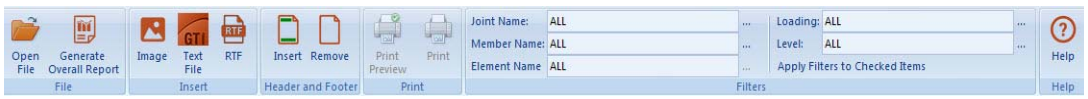
c) the viewer which shows data for the selected item.



GTS Report Builder commands can be accessed from the Ribbon Area at the top of the window. It includes the following functions:

1. open a database file and generate an overall report
2. insert images, GTI or RTF files
3. filter the data

The Ribbon Area is shown below:

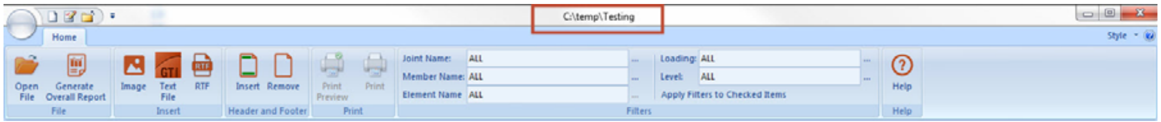


The available data from all DBX files within the same directory will be read and marked as bolded in the data categories tree. Items with captions that are not in bold are without data. An example of the DBX commands that may be used to create the data for Report Builder are shown below:

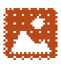
```
DBX ASCII80
WRITE REPLACE LOAD STATUS STATUS

DBX BINARY
WRITE REPLACE JOINT ATTRIBUTES JOINTS  EXISTING
WRITE REPLACE MEMBER ATTRIBUTES MEMBERS  EXISTING
WRITE REPLACE ELEMENT ATTRIBUTES ELEMENTS  EXISTING
WRITE REPLACE MEMBER PROPERTIES MEMBERS  EXISTING
WRITE REPLACE MEMBER CONSTANTS MEMBERS  EXISTING
WRITE REPLACE GROUP DATA ALL

WRITE REPLACE APPLIED JOINT LOADS JOINTS  EXISTING
WRITE REPLACE APPLIED MEMBER LOADS MEMBERS  EXISTING
WRITE REPLACE SECTION DISPLACEMENTS GLOBAL NS 5 MEMBERS  EXISTING

WRITE REPLACE JOINT RESULTS JOINTS  EXISTING
WRITE REPLACE MEMBER RESULTS MEMBERS  EXISTING
WRITE REPLACE SECTION FORCES NS 10 MEMBERS  EXISTING
WRITE REPLACE CODE MEMBERS  EXISTING
```

In many cases, the user will want to change the units for some of the items so multiple Units commands may be inserted between the WRITE commands shown above.

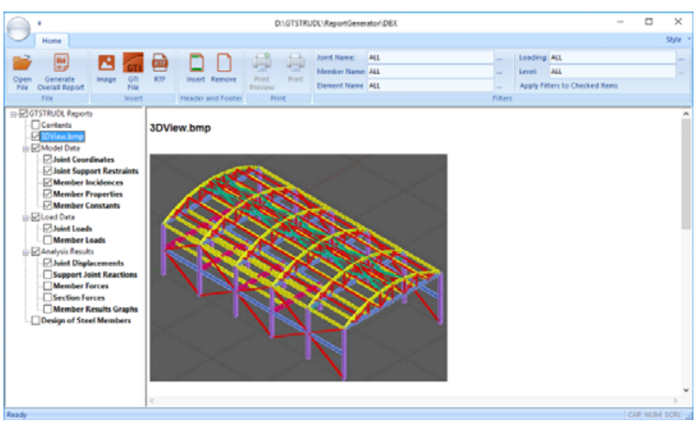
Once DBX files are opened, the directory appears at the top of the main GTS Report Builder window as shown in the figure below:



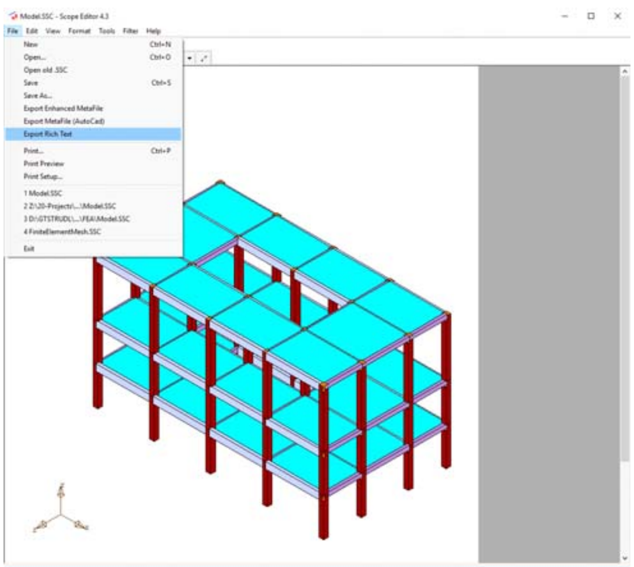
To load data for a specific item, double click on it or select "Refresh Item" by right clicking on it. If only one DBX file has been modified, you can refresh this item only.
Data for more than one item can be loaded by checking the preferred items and then by right clicking on it and choosing "Refresh Checked".

Images in JPG or BMP format can be inserted in the report by clicking on "Image" ribbon icon  of the "Insert" tab. The image is being placed after the selected item in the data categories tree.

To insert a BMP image, select an item in the data categories tree and then click on the "Image" ribbon icon. In the dialog window, "Select a BMP of JPG file" and then select the BMP file (e.g. 3DView.bmp) and click "Open". A new item with the BMP file name will be created in the data categories tree after the selected item as shown in the figure below:



The image can be moved to another position within the document, by right clicking on the item and selecting “Move Up” or “Move Down” from the pop-up menu.
Following the same procedure, a GTI file can be imported by clicking on “GTI File” ribbon icon of the “Insert” tab. The contents of GTI file are then placed after the selected item in the data categories tree. In the same manner, a RTF file can be imported by clicking on the “RTF” ribbon icon of the “Insert” tab. The contents of the RTF file will then be placed after the selected item in the data categories tree. A sample of RFT image produced by Scope Editor is shown below:



A header and footer template can be applied before generating the overall report by clicking on “Insert” ribbon icon of the “Header and Footer” tab. In the dialog window “Select a RTF File” select the RTF (e.g. sampleTemplate.rtf) and click “Open”. The header and footer can later be removed by clicking on the “Remove” ribbon icon of the “Header and Footer” tab.

An overall report can be created from the ribbon command “Generate Overall Report”. The report will be generated in RTF format in the DBX directory and automatically opened with the assigned software for RTF files on your computer. The default report filename is report.rtf. Only the selected elements in the structure tree will be included in this report. A Table of Contents of the report can be created by selecting the "Contents" item in the data categories tree.

The loaded data that fulfill several criteria can be filtered using the corresponding fields in the ribbon tab "Filters" as shown below. The loaded data can be filtered by Joint Name, Member Name, Element Name, Loading and Level. The filters can be either Global or Individual.

To apply a Global filter e.g. by Joint Name, type the joint name directly into the corresponding field"Joint Name" in the ribbon tab "Filters" (a blank space is needed to separate the values) or click on the "…" icon and select the joint name from a pop-up list. To apply the Global filter to the checked items, click on "Apply Filters to Checked Items" and then double click on an item to see the filtered data.

Whenever the Report Builder is exited, the preferences, selections, images, GTI files, filters, templates, etc are automatically saved. Whenever a DBX file is opened in the same folder, the previous preferences will be loaded automatically. In order to reset the preferences, delete the file "reportConfig.txt" that is created in the same directory.

Help is available by clicking on the Report Builder Help tab.

Chapter 3

Error Corrections

This chapter describes changes that have been made to GT STRUDL to correct errors. These errors may have produced aborts, incorrect results, or restricted use of a feature in previous versions of GT STRUDL. The error corrections are discussed by the primary feature areas of GT STRUDL.

3.1 Base Plate Wizard

1. The abort caused when multiple .gtbp files were restored in a single Base Plate Wizard session, without closing and then re-opening the Base Plate Wizard, has been fixed. This abort only occurred when the base plate models used the attachment extension option, where the attachment was created with 2D elements extending perpendicular to the base plate.

   (GPRF 2017.03)

2. When loading files with only values (type 701 with FX, MY, etc.) but without loading names (type 700) are used, the assumed load names will be displayed in the Loading page along with the specified values. Previously the Loading page would be blank in this situation, although the 701 data did exist internally and would be used in a' File –> Write GTSTRUDL file' request.

   Documentation:
   Section 9. Loadings, Base Plate Wizard Users Guide.

   (No GPRF issued)

3.2 CAD Modeler

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

1. An abort will no longer occur when you are meshing a 2D area and internal boundaries intersect the external boundaries of the area being meshed.

2. The Section Property information in the Model tab of the Member Properties dialog is now set to read-only since it can only be modified in the 2nd tab which is called Section Properties.

3. In BricsCAD V17, the edges of member profiles are now shown when displaying the members using 3D in the Display panel.

3.3 Joint Constraints

1. When multiple joint constraints of different types (JOINT TIE, RIGID PLATE, RIGID PLANE, RIGID SOLID) are specified with the same master joint, all slave joint associate with that same master joint are constrained according the last specified constraint type. The other constraint types are ignored. When this behavior is now detected, an error message is issued as illustrated by the following example:

```
{  162} > TYPE RIGID PIN
{  163} >  RIGID BODY INCIDENCES
{  164} >     'test1' 2 MASTER  3
{  165} >
{  166} > JOINT TIES
{  167} >   5 EQUALS  3

**** ERROR_THEQU -- Joint 3        has already been designated as a master
                    joint for a joint constraint of a type different from the
                    presently specified JOINT TIE type.  Different types of
                    joint constraints cannot be specified for the same master
                    joint. Analysis terminated and scan mode entered.

**** WARN_STDSY1 -- SCAN mode is ON.  Analysis will be halted.
```

(GPRF 2012.02)

3.4 DBX

1. Checking of member type (beam-type member vs. finite element) for WRITE MEMBER RESULTS is now correct. This issue only affected structures with both members and finite elements where ELEMENT INCIDENCES preceded MEMBER INCIDENCES.
(GPRF 2017.04)

3.5 Finite Elements

1. The following elements will now produce correct results when in-plane surface (PX,PY) and body forces (BX,BY) are applied in the LOCAL coordinate system of the element:
   Plane Stress elements
   CSTG, LST, Q6CDRL and all 2D isoparametric elements (elements that begin with IP such as the IPLQ, IPQL,….)
   Plate elements
   SBCT, SBMITC

   Most engineers do not specify surface or body forces in the LOCAL x and y directions since the LOCAL x and y system is a function of the first two element incidences and the order in which the engineer has created the element incidences (clockwise or counter clockwise order).

   (GPRF 2017.05)

3.6 GTMenu

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

1. GTMenu will no longer abort if the display of members and elements are turned off in the Label Setting dialog. This problem occurred in a model which had members and/or elements and Redraw Solid had been previously performed and then the display of members and elements was turned off and Redraw Solid was again selected.

2. If a gti file contains Construction Points and Reference (Local) Coordinates Systems, the Local Systems will now be drawn at their correct origin. Previously, the Local Systems would all be drawn at the lower left of the display.

3. Local Coordinate systems will now only be shown if they are in the current view. Previously, local systems outside of the current view would also be shown.

5. An error was corrected in the Create Element/Joints, and Create Member/Joints from Construction lines Dialogs. The error caused GTMenu to abort if an invalid spacing string was placed in the V direction, and the W direction was being edited in these dialogs. Also, the message for invalid data in these dialogs was improved.

6. An error which caused GTMenu to abort when Redraw Solid was used, and there are members created with the Create Members and Points using Lines and Points whose beta angle was defined by coordinates or joint, has been corrected. Also, Members whose beta angle was defined by coordinates or joint are now drawn with the correct beta angle.

7. The List option for selection using the TO parameter now recognizes Alphanumeric IDs with zero-padded numbers such as "SS00001" TO "SS00100".

8. The Create Materials Dialog will now show the predefined Materials (STEEL, ALUMINUM, CONCRETE, and DEF1) in the first four rows in order to allow the editing of user defined materials. Previously, user defined Materials were allowed to be placed in the first four rows, which are locked for editing.

9. Using the "LS" shortcut several times previously produced a stack of the same labels for supports in the graphical display area. Now, only one label is drawn no matter how many times the user enters the shortcut.

10. An error causing some models containing Double Angles to abort when the Color By Section option was used has been corrected.

11. The Color By Section option was fixed to show only members with the selected section properties. Previously, if the model contained more than 255 section properties definitions, and one of the selected properties section index was above 255, all members were shown.

12. The Check Model Loads function will now produce correct results for body and surface forces applied to plate bending, plate and all plane stress elements except for the transition elements (IPQLQ1,IPQLQ2,IPQLQ2B and IPQLQ3) which are rarely used. Previously, incorrect results would be reported for surface and body forces applied in the local, planar or global coordinate systems.

13. When refining finite elements using Hit or List Mode and after taking the first refinement of finite elements under All Mode and selecting Undo under Hit or List Mode, the Selection a Mesh of Finite Elements button did not work. This error has been corrected.

14. When refining finite elements by clicking Change to Higher Order Element, the refined elements were not correct if a higher order element was not available.

This error has been corrected.

15. When clicking “Select a Mesh of Finite Elements” button after taking the first refinement of finite elements, an abnormal message box was displayed. This message is no longer shown.

16. When displaying loads in List Mode in the Sum Forces At a Cut dialog, all loads were drawn. Now, only the loads for the selected selected entities are displayed.

17. Previously after clicking the Draw Load Arrows button in the Sum Forces At a Cut dialog, the Select Cut button did not work. This error has been corrected.

18. When double clicking a blank line inthe Inquire Output and Edit dialog, an unexpected dialog was displayed. This error has been corrected.

19. When defining spacing in the Extrude Model dialog, the selected direction was always reset to the +X direction. This has been corrected and the selected direction is no longer reset.

20. When extruding All 2D to 3D elements, the model was incorrect or an abort occurred. This error has been corrected.

21. In some cases, abort would occur when copying a model which contained member loads. This problem has been corrected.

22. If a model contained inactive joints which were the first joints created in the model, the list of loads in the Deformed Structure display was empty and results could not be displayed. This error has been corrected.

23. The error in the selection of all members in All mode after selecting Display Load Arrow in the Member Force dialog has been corrected.

24. An error in the selection of Construction Points during deletions of the Points has been corrected.

24. Correct prompts are now produced when deleting construction points.

25. Incorrect element load values are no longer produced in the Edit Element Load values dialog when you double click on a load in the Inquire Output and Edit

dialog..

26. When clicking Ky, Kz, Ly, Lz under Hit or List Mode in the Display Model dialog, All members were displayed. The Hit and List mode are now respected.

27. After deleting members or elements, there was discontinuity (gaps) in the member or element list due to the deleted IDs. The gap no longer occurs.

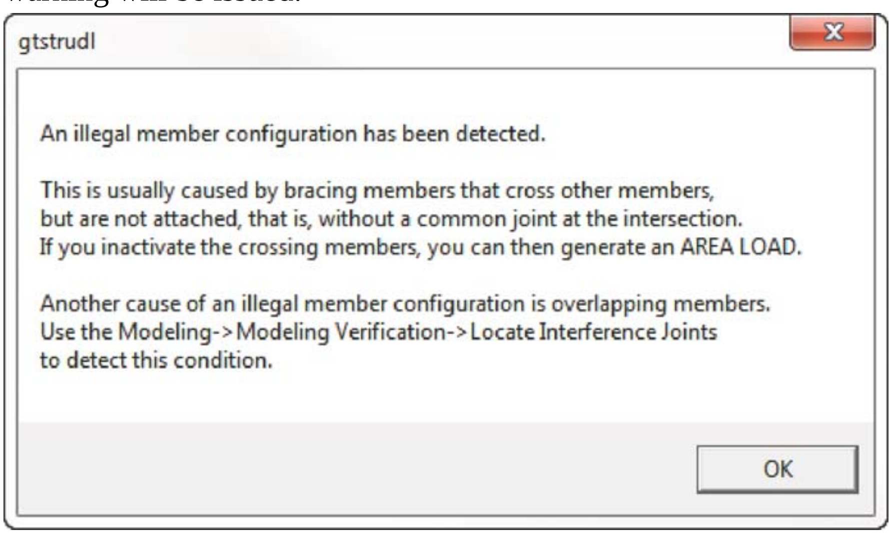
28. The display of members after reversing the incidence of members with eccentricities no longer occurs.

29. When double clicking on loading in the Inquire Output and Edit dialog after selecting loads using the Query button, incorrect messages such as NO JOINTS ARE SELECTED or NO MEMBERS ARE SELECTED no longer occur.

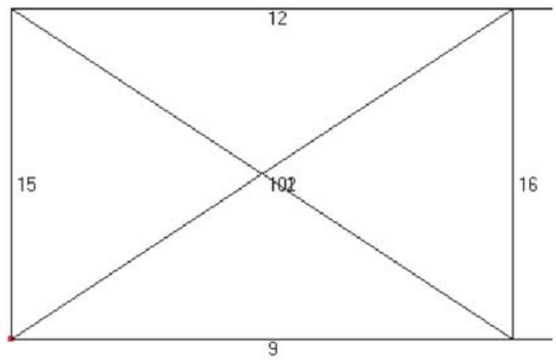
3.7 GTShell (Command Window)

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

1. An invalid member geometry for an elevation will no longer cause an abort. A warning will be issued:



and the elevation geometry will be displayed with the members labeled.



2. The Area Loads Dialog now produces the correct changes of values when using the "Edit Selected Areas" option in the menu. Also, the area limits issued in the command when the selected direction is Y are no longer flipped.

3. In versions prior to 2018, the GTShell menu function for displaying X-Y plots of harmonic analysis results versus frequency [*Results->Dynamic Analysis Results->Display Harmonic Results (vs Freq)*] would intermittently abort or produce incorrect plots when the harmonic analysis results were computed from GTSES/GT64M analyses. This problem has been corrected in Version 2018.

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 2017. 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. The Check Load option in CHECK MODEL dialog will produce incorrect load summations for body loads on all 3D solid finite elements.

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

   (No GPRF issued)

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

   Workaround: Verify that the loads are correct in the GT STRUDL Output Window using the PRINT LOAD DATA command or by checking the reactions using LIST SUM REACTIONS.

   (No GPRF issued)

5. GTMenu is limited to 1,000 views. If more than 1,000 views are created, incorrect displays may occur.
   (No GPRF issued)

6. The Deformed Structure display with the Deform between Joints option may produce inconsistent results for nonlinear geometric frame members. The deformed structure may show a discontinuity at the joints.
   (No GPRF issued)

7. GTMenu is limited to 10,000 Member Property Groups. If more than 10,000 property groups are created, incorrect results may occur. We have never encountered a model with more than 10,000 property groups.
   (No GPRF issued)

Chapter 5

Prerelease Features

5.1 Introduction

This chapter describes new features that have been added to GT STRUDL but are classified as prerelease features due to one or more of the following reasons:

1. The feature has undergone only limited testing. This limited testing produced satisfactory results. However, more extensive testing is required before the feature will be included as a released feature and documented in the GT STRUDL User Reference Manual.
2. The command formats may change in response to user feedback.
3. The functionality of the feature may be enhanced in response to user feedback.

The Prerelease features are subdivided into Design, Analysis, and General categories. The features in these categories are shown below:

5.2 Design Prerelease Features

5.2.1 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

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 the Table 1.3-7. The country names that are not listed in the 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**, 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 the Table 1.3-7 for each country.

1 The country names that are not listed in the 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 the Table 1.3-7 for each country.

1 the country names that are not listed in the 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 $\leq$ 2	b	0.34
	h/b > 2	c	0.49
Welded I cross-sections	h/b $\leq$ 2	c	0.49
	h/b > 2	d	0.76
For all other cross-sections		d	0.76

Table 1.3-9

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

Cross-section	Limits	Buckling curve	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:

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

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

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

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

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

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

where,

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

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

Explanation:

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

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

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

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

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

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

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

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

Requirements:

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

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

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

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

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

Usage:

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

Usage Example: Description of Example Structure

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

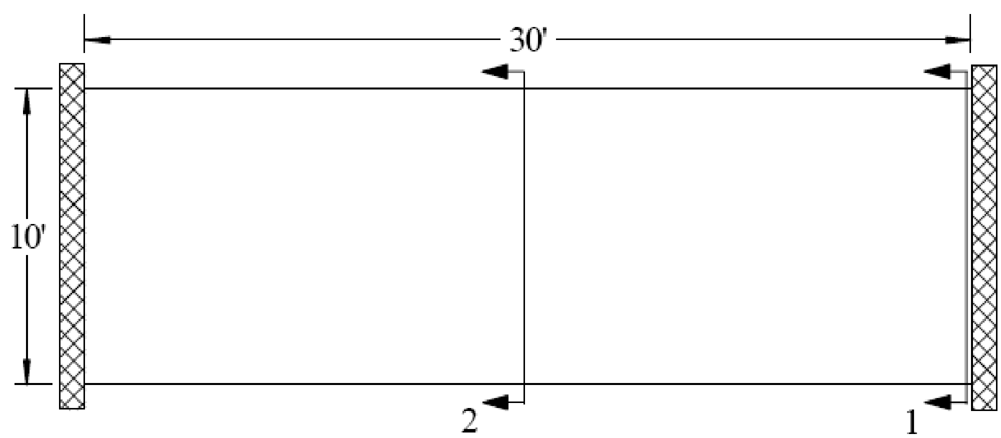


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

GT STRUDL Finite Element Model

The example structure was modeled in GT STRUDL using PLATE BENDING finite elements. The BPHQ element was utilized, and the configuration modeled corresponded to a mesh of ten elements by thirty elements. The model contained 300 finite elements and 341 nodes. The material properties were the default values associated with the MATERIAL REINFORCED CONCRETE command. All 6 degrees of freedom were restrained at each node along the supported ends of the slab system. Each element was loaded with a surface pressure of 150 psf, resulting in a confirmed summation of vertical reaction of 45,000 lb.

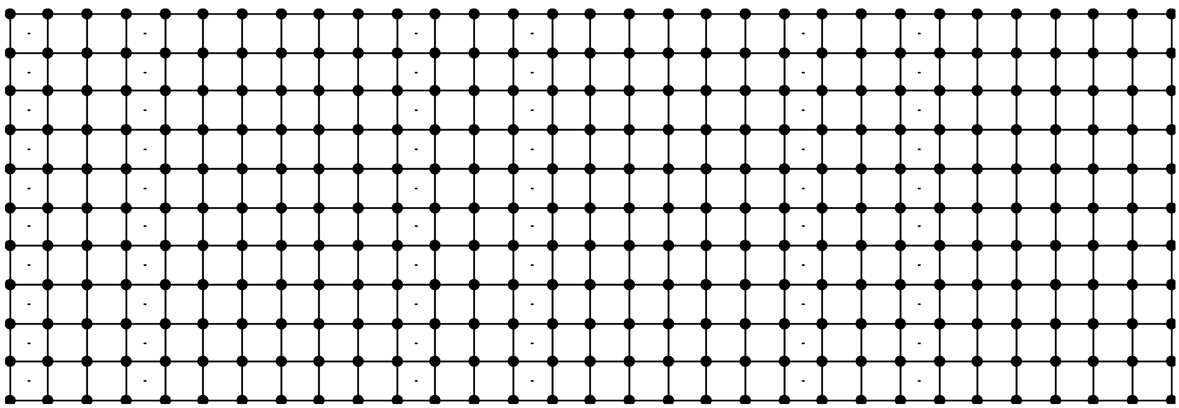


Figure 5.2.3-2 Example Finite Element Model

Definition of Cut Cross-Sections

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

Cut 1-1:

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

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

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

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

The 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:

$$\underline{\text{CAL}}\text{CULATE } \underline{\text{ERR}}\text{OR } (\underline{\text{EST}}\text{IMATE}) \ (\underline{\text{BAS}}\text{ED } \underline{\text{O}}\text{N}) \ -$$

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

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

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

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

Percent Maximum Difference Method

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

Percent Difference from Average Method
Normalized Percent Maximum Difference

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

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

Normalized Percent Difference from Average Method

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

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

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

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

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

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

General form:

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

Explanation:

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

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

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


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

      PROBLEM - None

      ACTIVE UNITS  FEET KIP  RAD  DEGF SEC
```

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



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

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

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

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

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

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

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:

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

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

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:

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

Explanation:

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

5.4.3 GTMenu POINT COORDINATE and LINE INCIDENCES Commands

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

(1) **GTMenu POINT COORDINATES**

General Form:

GTMenu POINT COORDINATES

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

.
.
.

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

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:

$\underline{\text{GTMenu}}$ $\underline{\text{LIN}}\text{E}$ $\underline{\text{INC}}\text{IDENCES}$

$$\left\{ 'a_1' \right\} \ \text{type}_1 \ \text{incidence-specs}_1$$

.
.
.

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

Elements:

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

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

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:

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

$$\begin{array}{l} \{'a_1'\}\ surface-specs_1 \\ \quad\vdots \\ \{'a_n'\}\ surface-specs_n \end{array}$$

Elements:

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

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

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

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

where,

‘a_1’, ‘a_2’, ..., ‘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'
```

End of Document