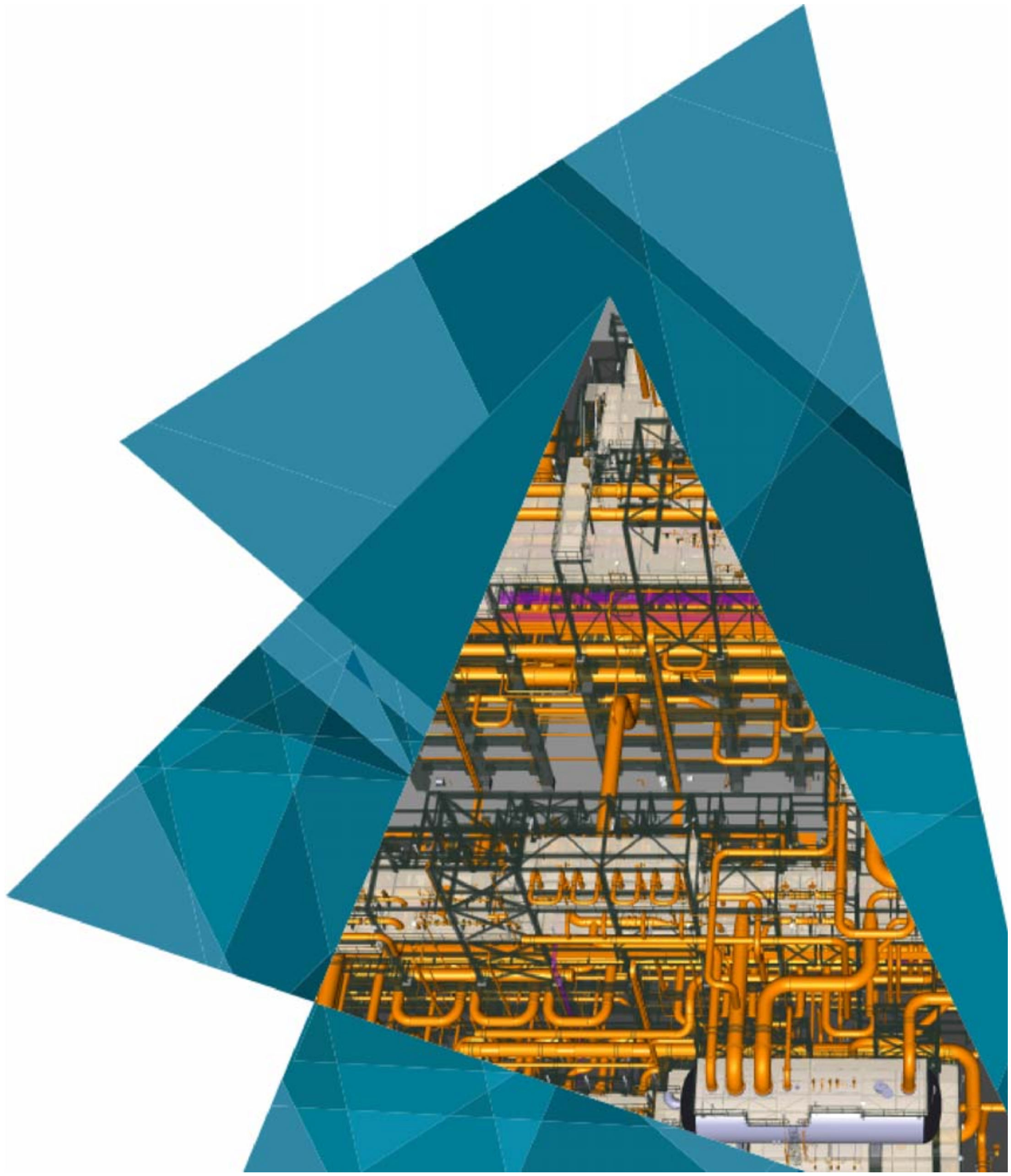
GT STRUDL® Version 2018 R1 Release Guide



Release Date: July 2018

HEXAGON PPM

Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 2018 R1 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 1

Introduction

Version 2018R1 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 37.1.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2018R1. In particular, Chapter 2 briefly describes an extensive list of new features including the following new features:

- New Chinese steel design code GB-2003
- New features in GTMenu (Automatic Views, Analysis executed within GTMenu, Mixed Units of Length, New Member Placement methods and numerous new features for the modeling and display of physical members)
- New Seismic Loading options
- New Automatic Load combinations
- New features in CAD Modeler (label, select and copy physical members)
- Ability to assign GUIDs to physical members and write the association between the GUIDs and physical members in a DBX file.

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

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

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

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

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

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 2018R1

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 2018R1. This release guide is also available online upon execution of GT STRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 Steel Design

1. A new Chinese National Standards GB 50017-2003 design code has been implemented. The GT STRUDL code name is called GB-2003 which is primarily based on the *People's Republic of China National Standard, GB 50017–2003, Code for Design of Steel Structures*, Published in April 25, 2003, Bulletin No. 147, Ministry of Construction of the People's Republic of China. This new code, GB-2003, may be used to select or check any of the following shapes:

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

   To use the GB-2003 code, you specify the Code Parameter as shown below:

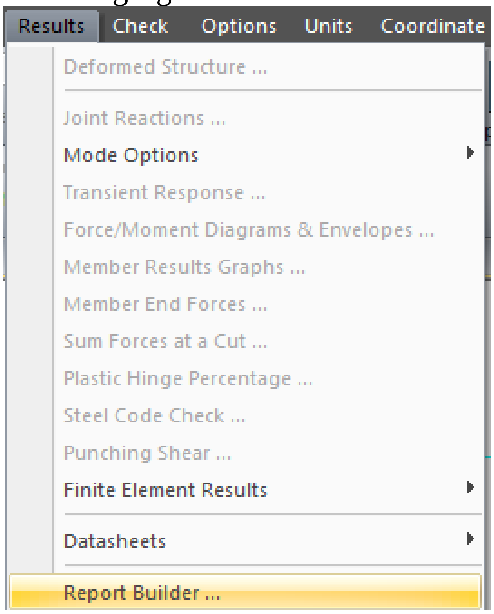
   ```
   PARAMETERS
     CODE   GB-2003   ALL
   ```

   Additional parameters and full documentation for the GB-2003 code may be found upon execution of GT STRUDL by selecting Help - Reference Documentation - Reference Manuals - Steel Design - GB-2003.

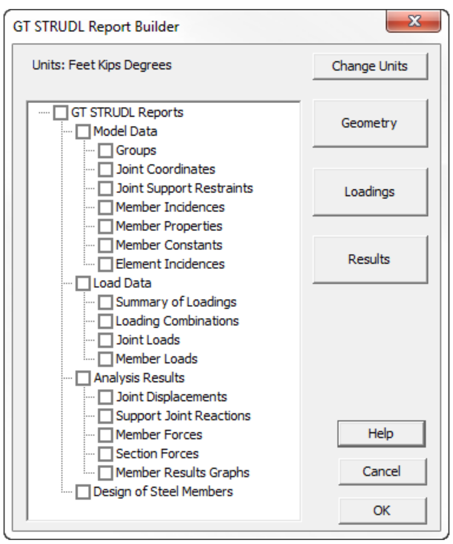
2. The National Annex of the country Spain is now available for the EC3-2005 steel design code. National Annex supported countries and parameters are documented in the Section 5.2.1 as well as Table 1.3 of the EC3-2005 User Reference Manual.

2.2 GTMenu

1. The new Report Builder dialog has been added to the Results menu as shown in the following figure:



This dialog allows you to select the type of data you want to include in a Report, writes the required DBX files in the proper format and then launches the GTSReportBuilder program which will automatically read the specified DBX files so you will not need to select them in the Report Builder.



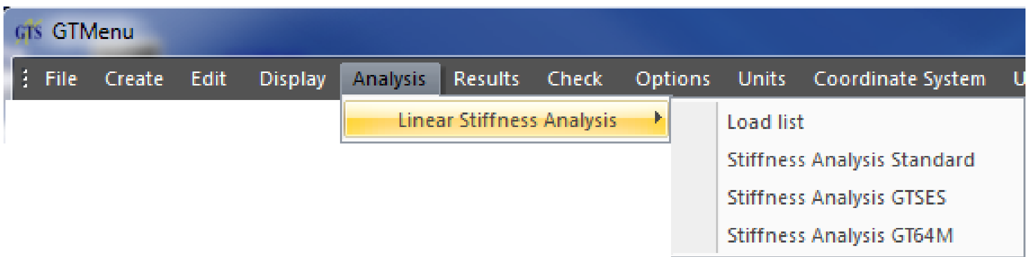
The Change Units button allows you to change the current units without leaving the dialog. This will also change the current GTMenu units.

The Help button will open the Report Builder documentation to explain what the various data types mean.

The Geometry, Loadings and Results buttons will check all the categories of that type. You can then un-check any data types you do not want to include.

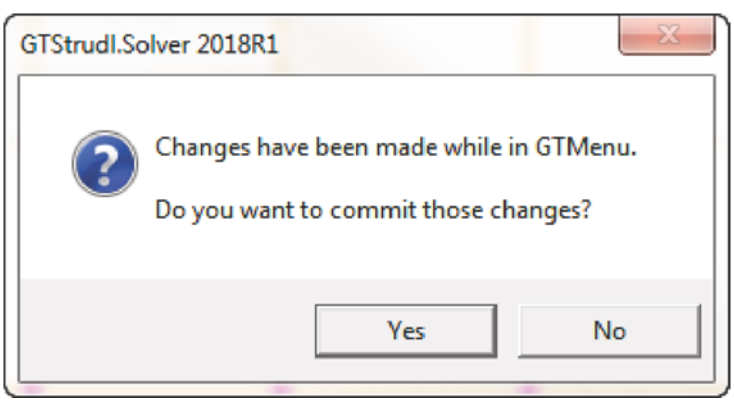
When you click the OK button, the Output Monitor dialog will open, where you can verify that the requested DBX files were written. Click the Done button to close the Output Monitor dialog and open Report Builder.

2. A new Analysis menu has been added to GTMenu.

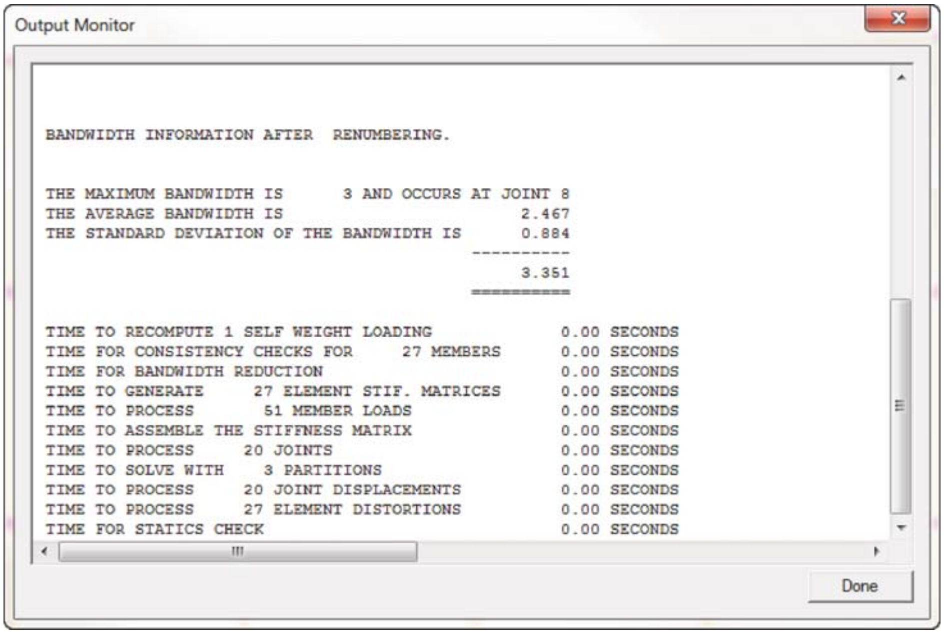


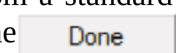
The Analysis menu allows you to run a linear stiffness analysis without exiting the GTMenu environment. In addition, you can also check and set the active loads to be considered during the analysis. You can choose the standard, in-core stiffness analysis or one of the external file analysis types.

Note that if changes have been made while in GTMenu, you must commit those changes to be able to use the Load List or Analysis options. A commit is equivalent to exiting GTMenu and saving changes:



When an analysis function is executed, all other functions in GTMenu are “frozen” and the Output Monitor is displayed.



The Output Monitor displays the output from the selected Analysis option; in the above graphic the output from a standard stiffness analysis is shown. When the option has been completed, the  button will be shown in the lower, right corner.

Click the Done button to close the Output Monitor and allow the other GTMenu functions to be accessible. After completion of an analysis option, the Results menu will check to determine which results are now available.

The addition of the new Report Builder feature and the Analysis feature now give you the ability to create a model, perform a static analysis and create a report without leaving GTMenu.

3. The Place Members Dialog has been significantly improved with a Copies option for the Specify Start & End, and the Start, Length, & Direction placement methods. This will allow the creation of arrays of members, honoring the other features in the dialog such as Split at Intersections and Physical Members.

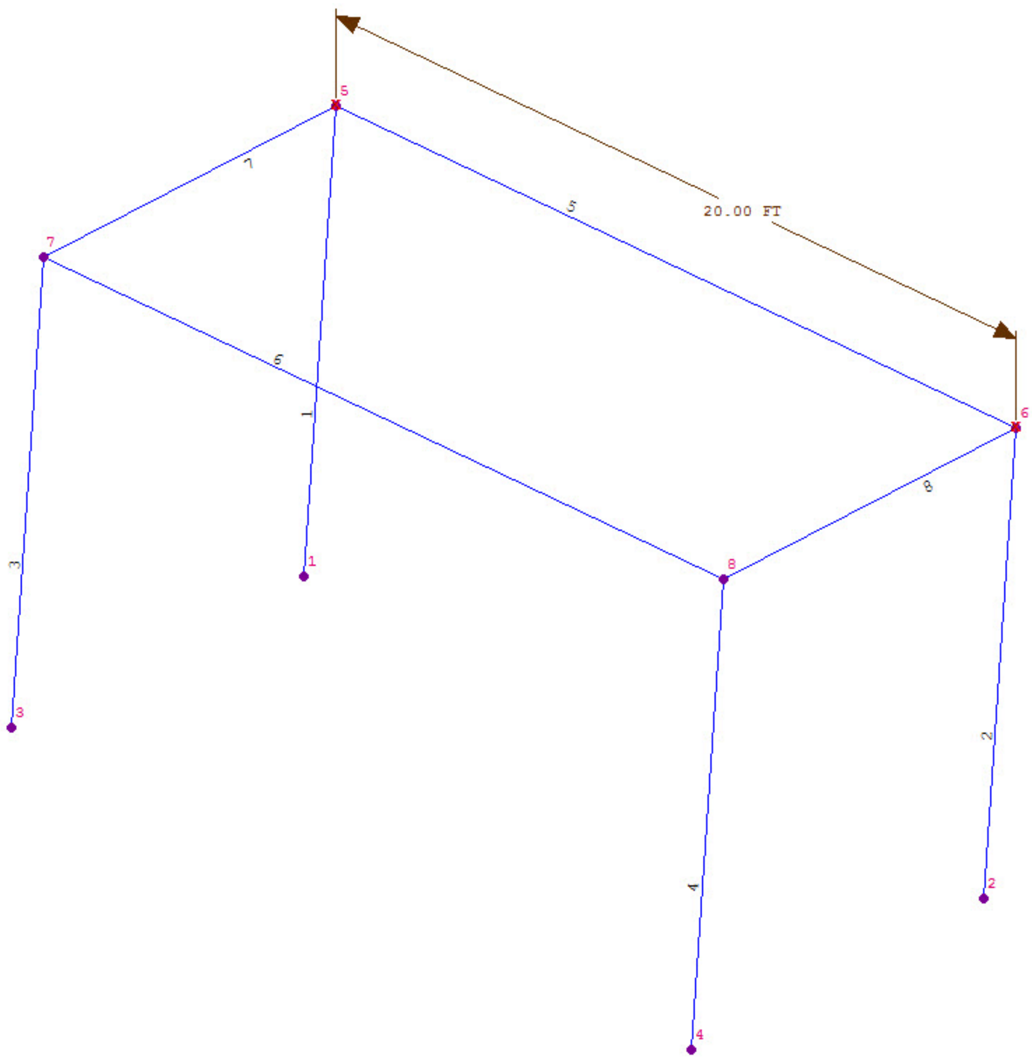
When using the Specify Start & End placement method, the number of copies will be in addition to the member defined by the two joints selected for start and end. The copies are created at the offset prescribed by the user. If an offset of (0,0,0) is given, no additional copies are generated. Previously, the offset option was not available when using the Specify Start & End placement method.

When using the Start, Length, & Direction placement method, no member is generated at the selected start joint if there is a valid offset. If an offset of (0,0,0) is given, a member will be generated at the start joint selected.

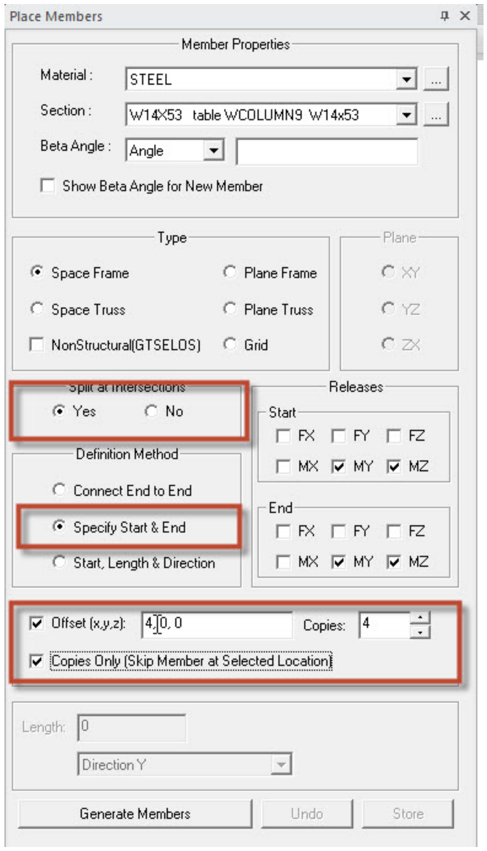
Additionally a Copies Only check box has been added in order to avoid generating the member at the selected joint/coordinate. This option is useful when there is already a member at the given location, and only the copies need to be generated.

A simple example to illustrate some of the new and enhanced features in the Place Members dialog is shown below:

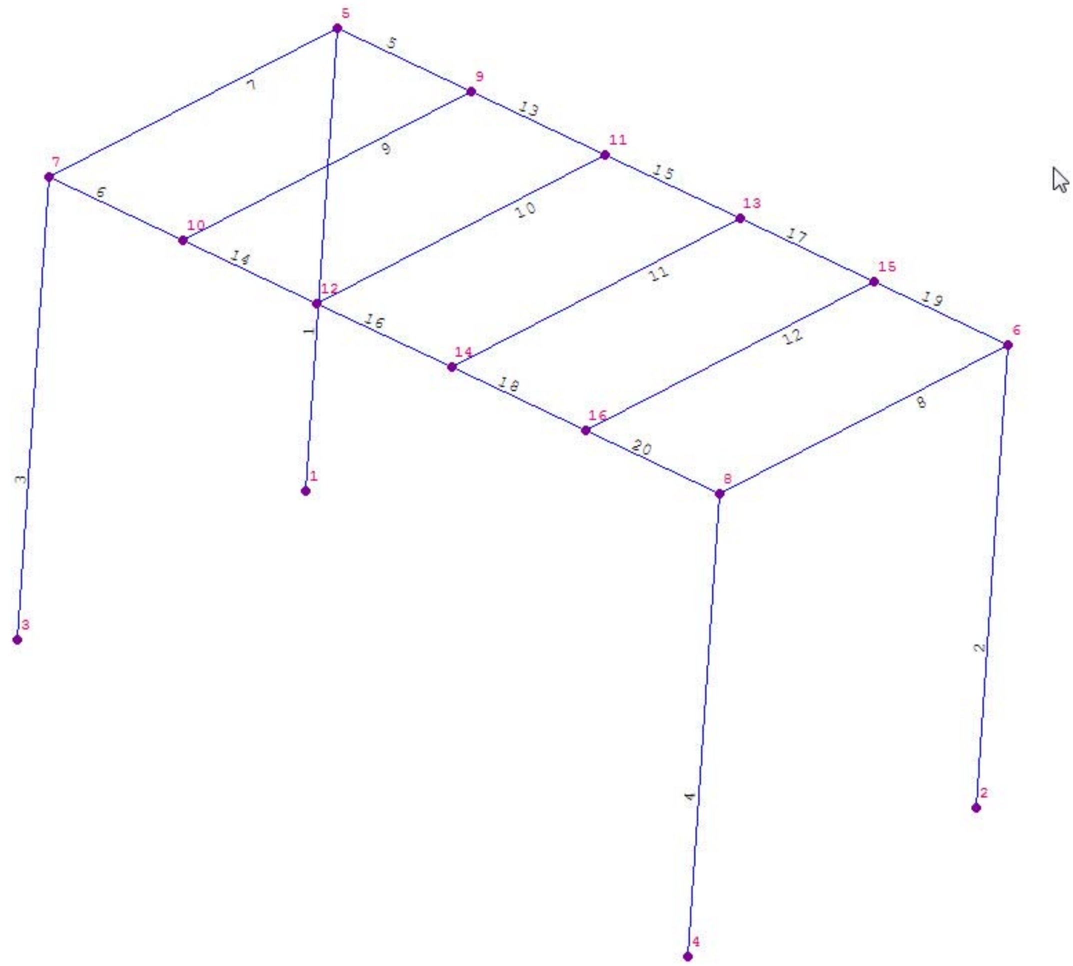
Perimeter framing:



The Place Member dialog is used to create 4 interior beams with Moment Y Z releases at the start and end and parallel to member 7 spaced uniformly at 4 feet is shown below. In this dialog, the Split at Intersections option is set to Yes which is now the default in this dialog. The Definition Method is set to Specify Start & End. The Offset is set to 4 ft in the X direction and you are going to create 4 Copies. In addition, the Copies Only box is checked so a member will not be created at the two joints that you select but the copies will be created.



After selecting the Generate Members button, you will then click on joints 5 and 7 which define the orientation and length of the copies that will be created. The interior beams will then be automatically created and members 5 and 6 will be automatically split into multiple analytical members as shown on the next page:

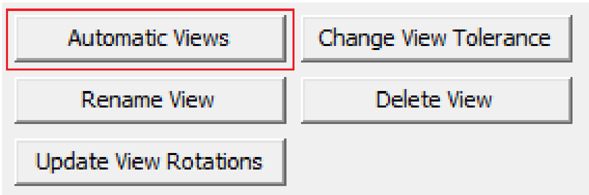


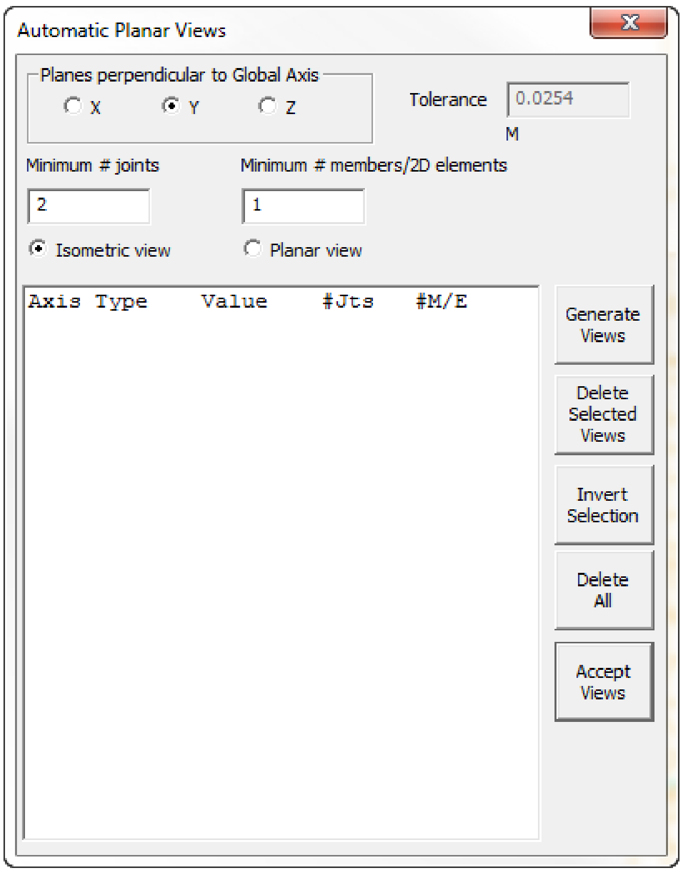
As can be seen in this example, these new and enhanced options can significantly reduce the number of steps required to create a model. Two new examples have been added to the GTMenu Guide to illustrate the use of these features for analytical and physical member models (Examples 9 and 10).

4. A new Automatic Planar Views dialog has been added to the Views dialog. Often a model will have planes of joints for floors, shear walls, etc. It can be useful to have a GTMenu View for each of these planes to make editing, property queries, result display, etc. easier and more understandable.

   The Automatic Planar Views feature will search the entire model for unbounded global planes, which is defined as a set of joints that lie within a volume orthogonal to the specified global axis, with a thickness along the specified axis of two times the current View tolerance. The user can specify which axis to search along and an isometric or planar view. In addition, the user can set a minimum number of joints to be considered as a plane and a minimum number of members or 2D elements. The detected planes are displayed in a list with the axis, view type, distance along the axis, number of joints in the plane and the number of members plus 2D elements that lie in the plane. The user can then choose from the list which planes are to become views.

The Automatic Planar Views dialog is launched from the Views dialog:





Set which axis to search along and any minimums, then click the Generate Views button: 

The model will be searched and the detected planes shown in the list:

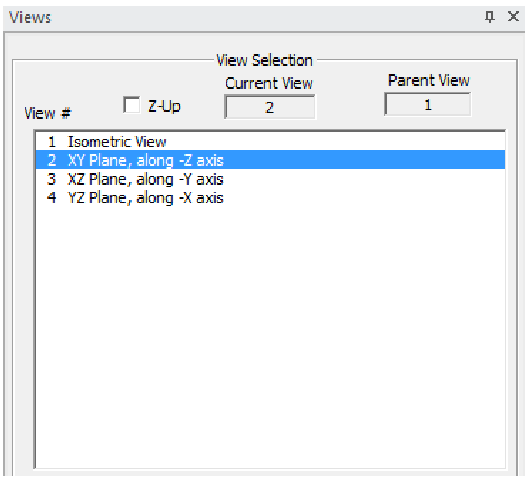
Axis	Type	Value	#Jts	#M/E
Y	Iso	14.200	4	4
Y	Iso	14.400	40	67
Y	Iso	19.340	9	9
Y	Iso	21.200	6	6
Y	Iso	22.400	51	93
Y	Iso	23.200	17	25
Y	Iso	25.000	75	117
Y	Iso	28.100	33	61
Y	Iso	30.700	21	21
Y	Iso	33.300	46	87
Y	Iso	36.400	24	20
Y	Iso	37.570	9	13
Y	Iso	39.400	23	22
Y	Iso	41.600	36	48
Y	Iso	43.300	18	22
Y	Iso	45.100	42	81
Y	Iso	47.600	21	21
Y	Iso	48.600	34	36
Y	Iso	48.720	12	16
Y	Iso	51.700	13	17

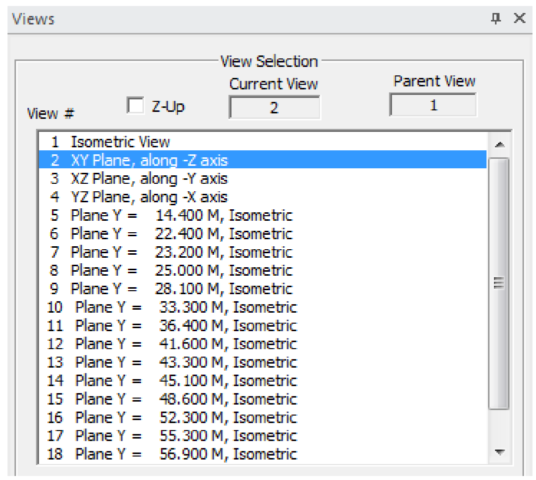
You can delete unwanted planes by selecting the planes and then click the ‘Delete Selected Views’ button:

Axis	Type	Value	#Jts	#M/E
Y	Iso	14.200	4	4
Y	Iso	14.400	40	67
Y	Iso	19.340	9	9
Y	Iso	21.200	6	6
Y	Iso	22.400	51	93
Y	Iso	23.200	17	25
Y	Iso	25.000	75	117
Y	Iso	28.100	33	61
Y	Iso	30.700	21	21
Y	Iso	33.300	46	87
Y	Iso	36.400	24	20
Y	Iso	37.570	9	13
Y	Iso	39.400	23	22
Y	Iso	41.600	36	48
Y	Iso	43.300	18	22
Y	Iso	45.100	42	81
Y	Iso	47.600	21	21
Y	Iso	48.600	34	36
Y	Iso	48.720	12	16
Y	Iso	51.700	13	17

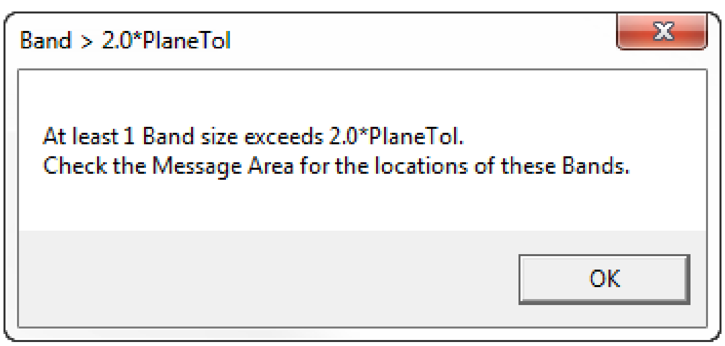
Then Delete Selected Views

The list box is a multi-select box which allows use of the standard Windows <Ctrl>click and <Shift>click selection functionality. When the list of planes is acceptable, use the Accept Views button to add the planes as Views for GTMenu:



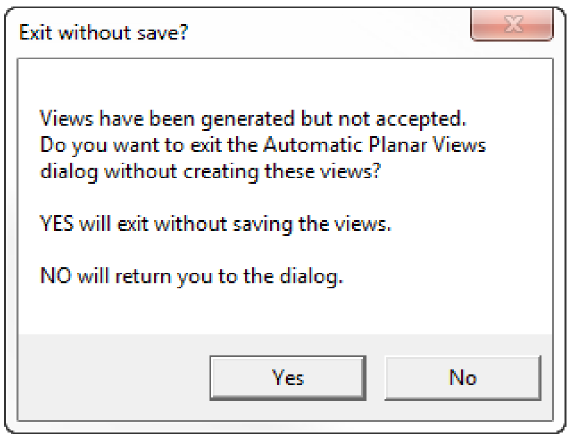


The chosen planes will appear in the View Selection list in the Views dialog as shown above. The list of planes in the Automatic Planar Views dialog will be cleared and you may continue to create planes and views as needed.
On occasion you may receive the following informative message after clicking the ‘Generate Views’ button:



When the model is searched for planes, the joint coordinates are separated into “bands” along the specified axis. Bands are defined as a set of joints where the lowest coordinate in the band is more than two times the View Tolerance greater than the next lower joint, and the highest coordinate in the band is more than two times the View Tolerance less that the next higher joint. Bands with a size (highest coordinate - lowest coordinate) of less than two times the View Tolerance can be viewed with a planar view and are entered into the displayed list. If any oversized bands exist, the above message is issued and the values of the ignored band are added to the GTMenu Message Area.

The Automatic Planar Views dialog is exited with the red X . If there are planes remaining in the list box, the following message is issued as a final confirmation that the generated planes are to be ignored. Click the Yes button if your intent is to ignore the displayed planes.



5. You may now specify mixed units of length regardless of the currently active length unit in all dialogs. This feature is especially useful when in feet or inches units but also is applicable when in metric units.

   The available length units are Feet, Inches, Millimeters, Centimeters, Meters and Mils which may be abbreviated as ft, in, mm, cm, m and mils respectively. You may use a single quote (') for feet and a double (") quote for inches. You can also use decimal and fractional units with a space required between the whole number and the fraction.

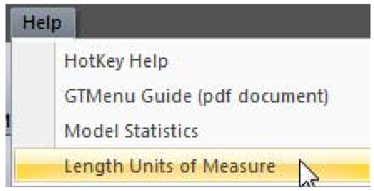
   Several examples are shown below for specifying a single distance in dialogs such as for member eccentricities, end joint size or split member absolute distances:

   20' or 20.5' or 3 1/2ft or 20'-3 1/2" (note the space after the 3)

   When specifying multiple distances or coordinates in a text box, a comma is required between each coordinate or distance as shown below:

20'-3 5/16", 14'-5", 6"
20ft-3 5/16in, 14ft-5in, 6in
x 20'-3 5/16", y 14'-5", z 6"

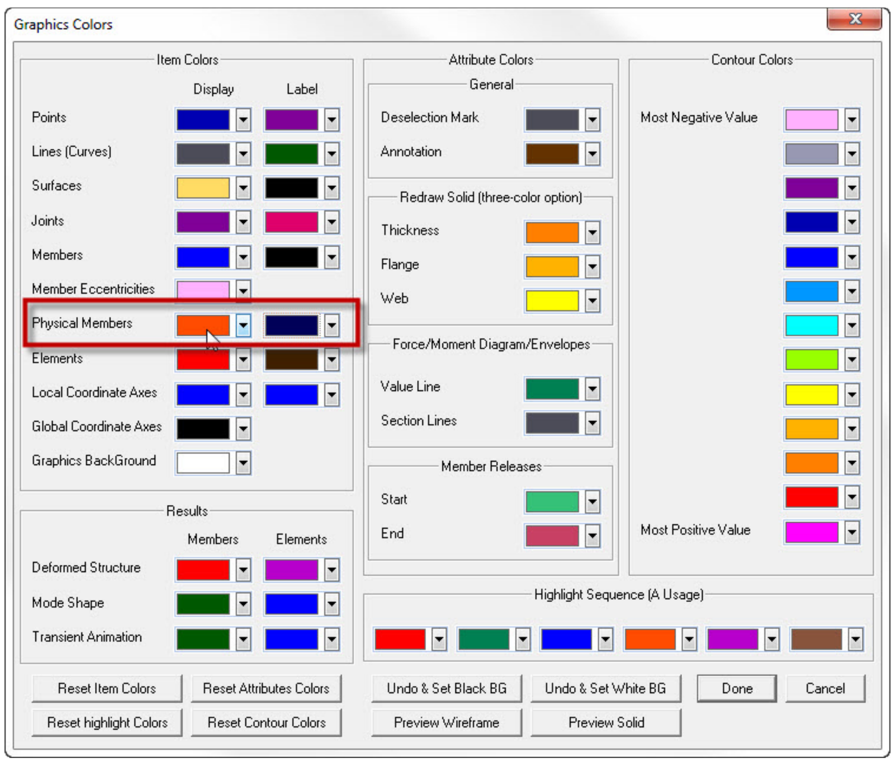
   These rules are also available under Help in GTMenu as shown below:



6. Numerous new features have been added for the display, modeling and editing of physical members.

<u>Display:</u>

You are now able to specify the colors for the display and labeling of physical members using the Graphics Colors Dialog as shown below:



<u>Placing Physical Members</u>

You are now able to place physical members using the Place Member Dialog. The Mode Bar must be set to Physical Members.

When placing physical members, the ID's of the physical members are automatically created as PMxxxxxx where xxxxxx is the next available integer. The first physical member created will be labeled PM1.

The following rules apply when placing physical members:

- Member Releases

  Start Member Releases will be applied at start joint ONLY of a newly created physical member. End Member Releases will be applied at end joint ONLY of the newly created physical member.

-Split at Intersections set to No

  Create one physical member between selected joints. In addition, each physical member will have only one analytical member.

-Split at Intersections set to Yes

  If intersection joints are the Start or End Joints of an already defined physical or analytical member, one physical member is created between the selected joints. In addition, the newly created physical member will have multiple analytical members created due to the intersections.

  If intersection joints are the intermediate joints of an already defined physical member, multiple physical members are created.
  If the intersection are not already a joint of an already defined physical member, multiple physical members are created.

  If the intersections are between the start and end of already defined analytical members, one physical member is created between the selected joints. In addition, each physical member will have multiple analytical members created due to the intersections.

Rules of thumb when placing physical members:

When placing a physical member with split at intersections enabled and two or more physical members intersect between their start and end joints, the order of creation will control which physical member retains its start and end. The physical member created first will retain its start and end joints while the other physical member will be split into multiple physical members.

If the placing of a physical member results in two physical members being created due to one of the above scenarios, the newly created physical members will have the member releases in the Place Member dialog applied to each new physical member.

Any existing physical members which are split due to the placing of a new analytical or physical member will retain member releases at its original start and end physical locations.

Edit Physical Members

You are now able to edit physical members directly using the Edit Member Data Dialog.

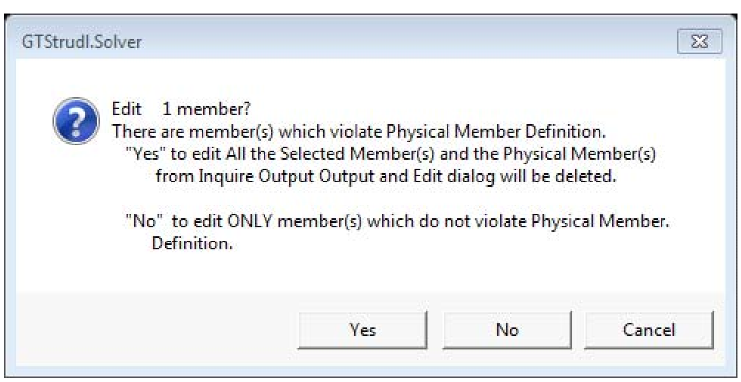
The following buttons and rules apply to the editing of physical members:

- Member Releases, End Joint Size,
  Apply start member releases or end joint size at the start joint only of a physical member.
  Apply end member releases or end joint size at the end joint only of a physical member.
- Type (Plane Frame, Space Frame,..), Beta Angle, Section Properties, Material Properties
  Apply the changes to all the analytical members belonging to the selected physical members.
- Eccentricities
  Applied uniform or linear variation to all analytical members which belong to the selected physical members.

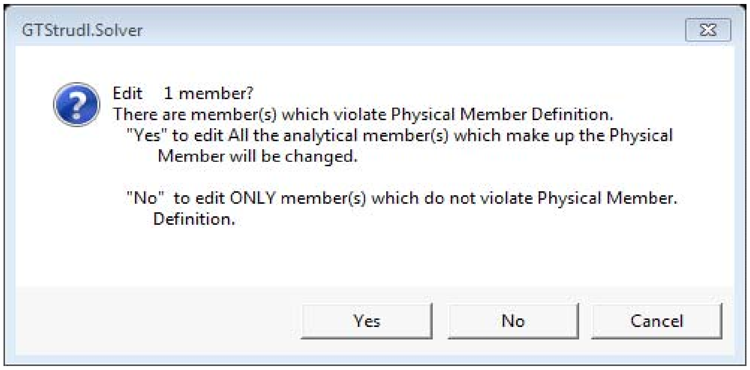
Edit Analytical Members belonging to Physical Members

Using the Edit Member dialog, you may edit analytical members which belong to a physical member. However, your editing changes must satisfy the physical member requirements. If you make changes which violate the physical member requirements, a warning message will pop-up.

The following Warning Message occurs when your editing changes violated the physical member requirements for member releases, end joint sizes, member incidences, member incidence orders, and member eccentricities:



The following Warning Message occurs when your editing changes violated the physical member requirements for member type, beta angle, section properties and material properties:



Edit Joints of Physical Members

You are now able to edit joints connected to physical members using the Edit Joint Data dialog. The joints in the physical members must remain in a straight line.

Split Member

When using the Split Member dialog and you have selected physical members to be split, each analytical member of the physical members will be split according to your specification. For instance, if you have selected to split uniformly into 2 spaces, each analytical member of the selected physical members will be split uniformly into two analytical members.

Copy Model

When using the Copy Model dialog and you have selected physical members to be copied, new physical members will be created. If you have selected analytical members and all the analytical members which comprise physical members are selected to be copied, corresponding new physical members will be created.

Refine Finite Element Mesh with Physical Members

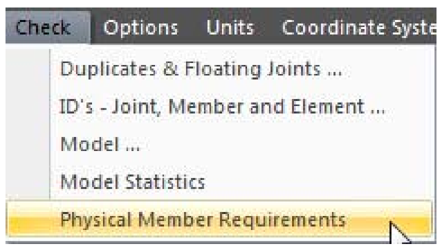
When using Refine Finite Element Mesh dialog and you have selected the option to Split any Framing Members, any framing members which are physical members will also be split.

Extrude Model

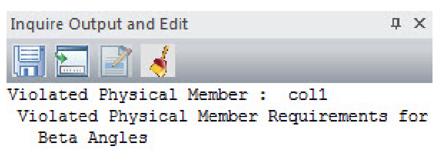
When using the Extrude Model dialog and you have selected physical members, the physical members will now be extruded. If you have selected analytical members and all the analytical members which comprise physical members are selected to be extruded, corresponding new physical members will be created.

Check Physical Member Requirements

The Check pulldown from the Menu Bar now has a new option to Check Physical Member Requirements as shown in the following figure:



Any members which violate the physical member requirements will be identified with a message in the Inquire Output and Edit dialog. An example is shown below where physical member col1 violated the Beta angle requirement.



Display Loads on Physical Members

The Display Load dialog now has an option to display loads on selected physical members. The loads on each analytical member of the physical members are displayed.

7. In order to avoid confusion, Material Property Groups are now referred to as Material Properties in the dialogs. In addition, Member Property Groups are now referred to as Section Properties in the dialogs.

2.3 Loadings

1. A sole vertical seismic load component may now be defined for a SEISMIC LOAD command block. For example, the Y axis can be selected for both HEIGHT AXIS and DIRECTION command options in order to apply a sole vertical Y component of a seismic loading to a structure.

```
SEISMIC LOAD 'SLY' 'Seismic Load Global Y' STANDARD ASCE7-10
  HEIGHT AXIS Y
  DIRECTION Y
  SEISMIC WEIGHT LOAD 'DLY'
  SEISMIC DEAD LOAD 'DLY'
  STORY HEIGHTS JOINTS EXISTING  37 57 77 97 117
  FLOOR TOLERANCE 2.0000
  SITE CLASS B
  RISK CATEGORY IV
  LONG PERIOD TL 12.0
  SPECTRAL ACCELERATION MAP USA LATITUDE 37.0 LONGITUDE -89.0
  FUNDAMENTAL PERIOD SMR
  RESPONSE MODIFICATION COEFFICIENT R 3.5
  ACCIDENTAL TORSION FACTOR ATMF 1.00
  VERTICAL SEISMIC FACTOR VSF 1.00
END OF SEISMIC LOAD DATA
```

2. Unlike the previous version, the combined seismic loading option, i.e. lateral and vertical seismic loading at the same time, for a SEISMIC LOAD command block is no longer allowed. Only a single seismic load component may be defined in a SEISMIC LOAD command block. When a non-zero VERTICAL SEISMIC FACTOR VSF is defined for a lateral seismic loading condition or a non-zero LATERAL SEISMIC FACTOR LSF is defined for a vertical seismic loading condition, the VSF or LSF will be disregarded and set to zero. If the combined seismic loading condition is still desired, the FORM LOADING command can be used to manually combine multiple loading conditions as shown below:

   FORM LOAD 'SLX+SLY' 'Combination of SLX and SLY' FROM 'SLX' 1.0 'SLY' 1.0

3. The LATERAL SEISMIC FACTOR LSF has been added to the SEISMIC LOAD commands to scale the lateral seismic loading in the Seismic Force Data command syntax
   below:

FUNDAMENTAL PERIOD { { *ASCE 7 - 05 struc type* / *ASCE 7 - 10 struc type* } (ANALYSIS MODE i_M) / TA v_{TA} }
(RESPONSE MODIFICATION (COEFFICIENT)) R v_R
(ACCIDENTAL (TORSIONAL) (MOMENT) (FACTOR)) ATMF v_{ATMF}
(LATERAL (SEISMIC) (FACTOR)) LSF v_{LSF}
(VERTICAL (SEISMIC) (FACTOR)) VSF v_{VSF}

Documentation:

Section 2.1.11.3.10, Volume 1, GT STRUDL Reference Manual

4. The REDUNDANCY FACTOR RF has been added to the SEISMIC LOAD command to accommodate users with another option to increase lateral seismic loading based on the ASCE standards (7-05 and 7-10) in the possible loss of seismic-resisting capabilities of structures especially for lateral forces to the structures. This new factor is defined in the Seismic Force Data command syntax below:

FUNDAMENTAL PERIOD { { *ASCE 7 - 05 struc type* / *ASCE 7 - 10 struc type* } (ANALYSIS MODE i_M) / TA v_{TA} }
(RESPONSE MODIFICATION (COEFFICIENT)) R v_R
(ACCIDENTAL (TORSIONAL) (MOMENT) (FACTOR)) ATMF v_{ATMF}
(LATERAL (SEISMIC) (FACTOR)) LSF v_{LSF}
(VERTICAL (SEISMIC) (FACTOR)) VSF v_{VSF}
(REDUNDANCY (FACTOR)) RF v_{RF}

5. Three new commands have been added that can be used to perform the automatic creation of standardized load combinations. The first command, STORE DESIGN LOAD VARIABLES, is used to store the associations of GTSTRUDL loading condition names with design load variables that are used to express standardized load combination equations. For example, a typical standard load combination equation may be expressed as 1.2D + 1.0E + 1.0L + 0.2S, in which the terms D, E, L, and S are design load variables that refer to the nominal dead load, earthquake load, live load, and snow load respectively. The syntax for the STORE DESIGN LOAD VARIABLES command is given as follows:

STORE DESIGN LOAD VARIABLES
    association specs
    ⋮
    association specs
END (STORE) (DESIGN) (LOAD) (VARIABLE)

where

association specs = LOADS llist VARIABLE lvn,

$$\text{lvn} = \left\{ \begin{array}{l} \text{D} \quad \text{DI} \quad \text{L} \quad \text{L100} \quad \text{LEX} \quad \text{LR} \quad \text{R} \quad \text{S} - \\ \text{W} \quad \text{WI} \quad \text{E} \quad \text{NL} - \\ \text{F} \quad \text{FA1} \quad \text{FA2} \quad \text{HA} \quad \text{HR} \quad \text{T} \quad \text{AK} - \\ \text{DELETE} \end{array} \right\}$$

llist = list of GTSTRUDL loading names to which a standard load combination design load variable name is associated. The GTSTRUDL loading names may refer to both independent and dependent static loads that have been previously defined.

lvn = design load variable name that is used in the load combination equations as defined by the selected governing standard.

The second command is a modified version of the CREATE AUTOMATIC LOAD COMBINATION command, shown as follows:

$$\underline{\text{CRE}}\text{ATE}\ \ \underline{\text{AUT}}\text{OMATIC}\ \begin{Bmatrix}(\underline{\text{LOA}}\text{D})\ \ \underline{\text{COM}}\text{BINATIONS}\\ \underline{\text{FOR}}\text{M}\ (\underline{\text{LOA}}\text{DS})\end{Bmatrix}\ \text{-}$$

$$\underline{\text{DES}}\text{IGN}\ \ (\underline{\text{STAN}}\text{DARD})\ \begin{Bmatrix}\underline{\text{ASCE710L}}\\ \underline{\text{ASCE710A}}\\ \underline{\text{AISC14L}}\\ \underline{\text{AISC14A}}\end{Bmatrix}\ \text{-}$$

$$(\underline{\text{EQU}}\text{ATIONS}\ \ ^{*}\begin{Bmatrix}\rightarrow \underline{\text{ALL}}\\ i_{EQ1},\ i_{EQ2},\ \cdots,i_{EQn}\end{Bmatrix})\ \text{-}$$

$$(\underline{\text{STA}}\text{RT}\ \ (\underline{\text{ID}})\ \ i_S)\ \ (\underline{\text{LOA}}\text{D}\ (\underline{\text{GRO}}\text{UP})\ \begin{Bmatrix}i_{LG}\\ 'a_{LG}'\end{Bmatrix})\ \ (\underline{\text{PRI}}\text{NT}\ \begin{Bmatrix}\rightarrow \underline{\text{ON}}\\ \underline{\text{OFF}}\end{Bmatrix})$$

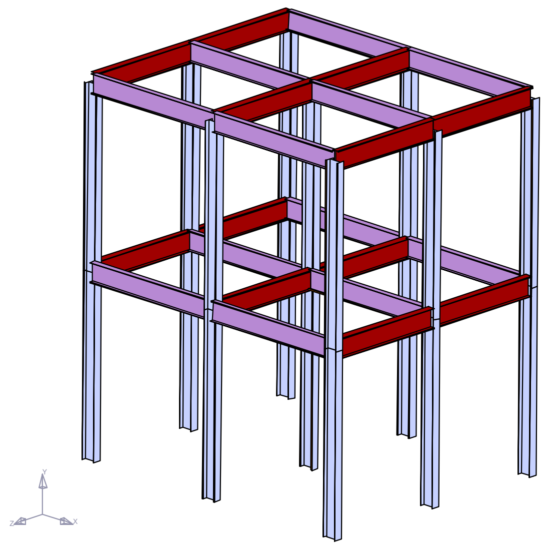
where

i_{EQ1}, i_{EQ2}, ... i_{EQn}	=	list of integers, each of which refers to a GTSTRUDL load combination equation number that corresponds to a load combination equation defined by the specified standard.
i_S	=	integer ID of the first loading combination or form load to be generated by this command. This integer is limited to eight digits.
i_{LG}/‘a_{LG}’	=	the integer or alphanumeric name of a load GROUP that is automatically defined and created with the names of all of the load combinations or form loads created by the current command. i_{LG} and ‘a_{LG}’ are limited to eight digits and eight characters respectively.

This new version of the CREATE AUTOMATIC LOAD COMBINATIONS command is used to execute the creation of the applicable standardized load combinations according to the selected governing code or standard and based on the design load variables that are associated with valid GTSTRUDL loading conditions.

The third command is the PRINT DESIGN LOAD VARIABLES DATA command, which produces a report that shows the association of loading conditions with design load variables.

The use of the STORE DESIGN LOAD VARIABLES, CREATE AUTOMATIC LOAD COMBINATION DESIGN, and the PRINT DESIGN LOAD VARIABLES DATA commands is illustrated in the following simple example, which also includes commentary identified by the characters “$* **” at the beginning of a line and the text reports produced by the execution of the commands.



Structural Model Data

Story Heights:	14 feet
Bay Widths, Global X:	12 feet
Bay Widths, Global Z:	10 feet
Material:	Steel
Column Sections:	W14X53
Beam Section, Global X:	W18X50
Beam Section, Global Z:	W18X35
Supports:	Fixed

Loading Data

Self Weight Load:	SLY
Live Load:	LL
Roof Live Load:	LLROOF
Snow Load:	SNOW
Wind Load, Global X:	WLX
Wind Load, Global Z:	WLZ

Load Combination Standard:	ASCE 7-10
Design Load Variable Definitions:	

Load SWY	Variable D
Load LL	Variable L
Load LLROOF	Variable LR
Load SNOW	Variable S
Loads WLX, WLZ	Variable W
Load Combination Equations:	1, 2, 3, 4, 6

```
$* **
$* ** Define structural data: geometry, material properties, member
$* ** structural properties
$* **
TYPE SPACE FRAME
MATERIAL STEEL
UNITS FEET KIPS

JOINT COORDINATES
$        Name           X coord          Y coord          Z coord
$    --------     -------------     ------------     ------------
            1          0.00000          0.00000          0.00000
            2         12.00000          0.00000          0.00000
            3         24.00000          0.00000          0.00000
            4          0.00000          0.00000         10.00000
            5         12.00000          0.00000         10.00000
            6         24.00000          0.00000         10.00000
            7          0.00000          0.00000         20.00000
            8         12.00000          0.00000         20.00000
            9         24.00000          0.00000         20.00000
           10          0.00000         14.00000          0.00000
           11         12.00000         14.00000          0.00000
           12         24.00000         14.00000          0.00000
           13          0.00000         14.00000         10.00000
           14         12.00000         14.00000         10.00000
           15         24.00000         14.00000         10.00000
           16          0.00000         14.00000         20.00000
           17         12.00000         14.00000         20.00000
           18         24.00000         14.00000         20.00000
           19          0.00000         28.00000          0.00000
           20         12.00000         28.00000          0.00000
           21         24.00000         28.00000          0.00000
           22          0.00000         28.00000         10.00000
           23         12.00000         28.00000         10.00000
           24         24.00000         28.00000         10.00000
           25          0.00000         28.00000         20.00000
           26         12.00000         28.00000         20.00000
           27         24.00000         28.00000         20.00000

DEFINE GROUP 'support' ADD JOINTS 1 to 9
STATUS SUPPORT JOINT GROUP 'support'
$ Remove the dollar sign ($) below for pinned support condition
$ JOINT GROUP 'support' RELEASES MOMENT X Y Z
$
MEMBER INCIDENCES
$        Name  Start joint  End joint
$    --------     --------   --------
            1            1         10
            2            2         11
            3            3         12
            4            4         13
            5            5         14
            6            6         15
            7            7         16
            8            8         17
            9            9         18
           10           10         11
```

```
           11          11         12
           12          13         14
           13          14         15
           14          16         17
           15          17         18
           16          10         13
           17          13         16
           18          11         14
           19          14         17
           20          12         15
           21          15         18
           22          10         19
           23          11         20
           24          12         21
           25          13         22
           26          14         23
           27          15         24
           28          16         25
           29          17         26
           30          18         27
           31          19         20
           32          20         21
           33          22         23
           34          23         24
           35          25         26
           36          26         27
           37          19         22
           38          22         25
           39          20         23
           40          23         26
           41          21         24
           42          24         27
$ Group members by geometry
DEFINE GROUP 'columns' ADD MEMBERS 1 TO 9 22 TO 30
DEFINE GROUP 'Xbeams'  ADD MEMBERS 10 TO 15 31 TO 36
DEFINE GROUP 'Zbeams'  ADD MEMBERS 16 TO 21 37 TO 42

MEMBER PROPERTIES
  GROUP 'columns'   T 'WCOLUMN9' 'W14x53'
MEMBER PROPERTIES
  GROUP 'Xbeams'   T 'WBEAM9' 'W18x50'
MEMBER PROPERTIES
  GROUP 'Zbeams'   T 'WBEAM9' 'W18x35'





$* **
$* ** Define independent loading conditions:
$* **
$* **
$* ** Self Weight of structure
$* **
SELF WEIGHT 'SWY' 'Self weight of all members' DIR -Y ALL MEMBERS

$* **
$* ** Live Load, Elev. 14.0 ft, 150 psf
$* **
AREA LOAD 'LL' 'Floor Live Load' DIRECTION Y PLANE TOLERANCE 0.167
  ELEVATION 14.000000 VALUE 0.150000 TWO WAY
END AREA LOADS

$* **
$* ** Roof Live Load, Elev. 28.0 ft, 100 psf (pattern)
$* **
AREA LOAD 'LLROOF' 'Roof Live Load' DIRECTION Y PLANE TOLERANCE 0.167
```

```
  ELEVATION 28.000000 EXCEPT LIMITS MEMBERS 31 39 33 37 LIMITS MEMBERS 34 42 -
  36 40 VALUE 0.100000 TWO WAY
END AREA LOADS

$* **
$* ** Snow Load on Roof, Elev. 28.0 ft, 100 psf
$* **
AREA LOAD 'SNOW' DIRECTION Y PLANE TOLERANCE 0.167
  ELEVATION 28.000000 VALUE 0.100000 TWO WAY
END AREA LOADS

$* **
$* ** Wind Loads, Global X and Z Directions
$* **
WIND LOAD  'WLX'  'Wind Load, Global X'
  UNITS  FT  LBS  DEG  SEC  StdMASS
  STANDARD ASCE7-05
  ELEVATION AXIS Y
  EXPOSURE CATAGORY C
  SPEED MPH 100.0
  DIRECTION ANGLE 0.0
  DIRECTIONALITY FACTOR 1.00
  TOPOGRAPHIC FACTOR KZT 0.85
  GUST FACTOR G 0.85
  GROSS AREA AG 1800.00
  ADDED FORCE AREA AFADD 0.0
  UNITS  FT  LBS  DEG  SEC  StdMASS
  MEMBERS EXISTING  1 4 7 16 17 22 25 28 37 38 -
    TYPE INDUSTRIAL FRAME -
    AREA AND FORCE PROPERTIES CDG 1.1 FLD 1.0
END WIND LOAD DATA

WIND LOAD  'WLZ'  'Wind Load, Global Z'
  UNITS  FT  LBS  DEG  SEC  StdMASS
  STANDARD ASCE7-05
  ELEVATION AXIS Y
  EXPOSURE CATAGORY C
  SPEED MPH 100.0
  DIRECTION ANGLE -90.0
  DIRECTIONALITY FACTOR 1.00
  TOPOGRAPHIC FACTOR KZT 0.85
  GUST FACTOR G 0.85
  GROSS AREA AG 1800.00
  ADDED FORCE AREA AFADD 0.0
  UNITS  FT  LBS  DEG  SEC  StdMASS
  MEMBERS EXISTING  1 2 3 10 11 22 23 24 31 32 -
    TYPE INDUSTRIAL FRAME -
    AREA AND FORCE PROPERTIES CDG 1.1 FLD 1.0
END WIND LOAD DATA
```

```
$* **
$* ** Design Load Variables for Load Combination Creation
$* **
STORE DESIGN LOAD VARIABLES
  LOAD 'SWY'                 VARIABLE D
  LOAD 'LL'                  VARIABLE L
  LOAD 'LLROOF'              VARIABLE LR
  LOAD 'SNOW'                VARIABLE S
  LOADS 'WLX' 'WLZ'          VARIABLE W
END DESIGN LOADS

$* **
$* ** Produce a text report of the defined Design Load
```

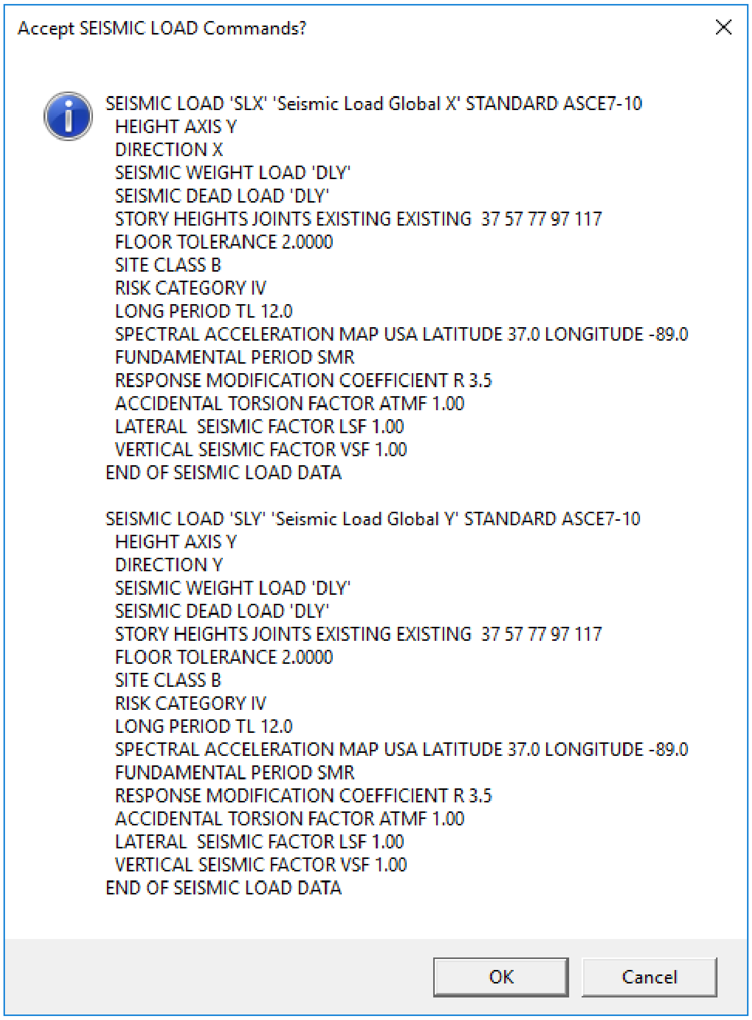
```
$* ** Variable associations
$* **
PRINT DESIGN LOAD VARIABLES


 *************************************
 * Problem Data From Internal Storage *
 *************************************

  Problem: NONE     Title: NONE GIVEN


  Active Units:  FEET LB   DEG  DEGF SEC



 Loading       Status                     Design Load Variable
---------    ----------   /----------------------------------------------------/

 SWY          ACTIVE           D       Dead Load
 LL           ACTIVE           L       Live Load
 LLROOF       ACTIVE           LR      Roof Live Load
 SNOW         ACTIVE           S       Snow Load
 WLX          ACTIVE           W       Wind Load
 WLZ          ACTIVE           W       Wind Load


$* **
$* ** Create Load Combinations
$* **   Generated load combination ID numbers start with 2001
$* **   Generated load combination IDs are stored in Group LCGRP
$* **
CREATE AUTOMATIC COMBO LOADS DESIGN STANDARD ASCE710L EQUATIONS 1 2 3 4 6 -
  START ID 2001 GROUP 'LCGRP  ' PRINT ON


ASCE 7-10 LRFD Load Combination Equation 1
------------------------------------------
  LOAD COMB  2001 SPECS      'SWY' 1.400

ASCE 7-10 LRFD Load Combination Equation 2
------------------------------------------
  LOAD COMB  2002 SPECS      'SWY' 1.200  'LL' 1.600  'LLROOF' 0.500
  LOAD COMB  2003 SPECS      'SWY' 1.200  'LL' 1.600  'SNOW' 0.500

ASCE 7-10 LRFD Load Combination Equation 2, Ice
-----------------------------------------------
**** INFO_ASCE710LAUTOLC -- No valid loading condition is associated with one or more
                            of design load variables D, L, and DI.
                            No loading combination is generated for ASCE 7-10 LRFD
                            Load Combination Equation 2, Ice.

ASCE 7-10 LRFD Load Combination Equation 3
------------------------------------------
  LOAD COMB  2004 SPECS      'SWY' 1.200  'LLROOF' 1.600  'LL' 1.000


  LOAD COMB  2005 SPECS      'SWY' 1.200  'LLROOF' 1.600  'WLX' 0.500
  LOAD COMB  2006 SPECS      'SWY' 1.200  'LLROOF' 1.600  'WLZ' 0.500

  LOAD COMB  2007 SPECS      'SWY' 1.200  'SNOW' 1.600  'LL' 1.000

  LOAD COMB  2008 SPECS      'SWY' 1.200  'SNOW' 1.600  'WLX' 0.500
  LOAD COMB  2009 SPECS      'SWY' 1.200  'SNOW' 1.600  'WLZ' 0.500
```

```
ASCE 7-10 LRFD Load Combination Equation 4
------------------------------------------
  LOAD COMB  2010 SPECS      'SWY' 1.200  'LL' 1.000  'LLROOF' 0.500  'WLX' 1.000
  LOAD COMB  2011 SPECS      'SWY' 1.200  'LL' 1.000  'LLROOF' 0.500  'WLZ' 1.000

  LOAD COMB  2012 SPECS      'SWY' 1.200  'LL' 1.000  'SNOW' 0.500  'WLX' 1.000
  LOAD COMB  2013 SPECS      'SWY' 1.200  'LL' 1.000  'SNOW' 0.500  'WLZ' 1.000

ASCE 7-10 LRFD Load Combination Equation 4, Ice
-----------------------------------------------
**** INFO_ASCE710LAUTOLC -- No valid loading condition is associated with one or more
                            of design load variables D, DI, L and/or L110, and WI.
                            No loading combination is generated for ASCE 7-10 LRFD
                            Load Combination Equation 4, Ice.

ASCE 7-10 LRFD Load Combination Equation 6
------------------------------------------
  LOAD COMB  2014 SPECS      'SWY' 0.900  'WLX' 1.000
  LOAD COMB  2015 SPECS      'SWY' 0.900  'WLZ' 1.000

ASCE 7-10 LRFD Load Combination Equation 6, Ice
-----------------------------------------------
**** INFO_ASCE710LAUTOLC -- No valid loading condition is associated with one or more
                            of design load variables D, DI, and WI.
                            No loading combination is generated for ASCE 7-10 LRFD
                            Load Combination Equation 6, Ice.


**** INFO_AUTOLCGRP -- Group LCGRP    has been created from the above loads.
```

Documentation:
Section 2.1.11.3.6, Volume 1, GT STRUDL Reference Manual

2.4 GT STRUDL Output Window (GTShell)

1. The Seismic Load Generation dialog has been modified in the General Data section in conjunction with the changes in the SEISMIC LOAD commands:
   a. Since a combined seismic loading is no longer available for a SEISMIC LOAD command block, when a non-zero VERTICAL SEISMIC FACTOR VSF is defined for a lateral seismic loading condition or a non-zero LATERAL SEISMIC FACTOR LSF is defined for a vertical seismic loading condition, the VSF or LSF will be disregarded and set to zero. Thus, each SEISMIC LOAD command block as shown below will apply a single seismic load component to the structure.

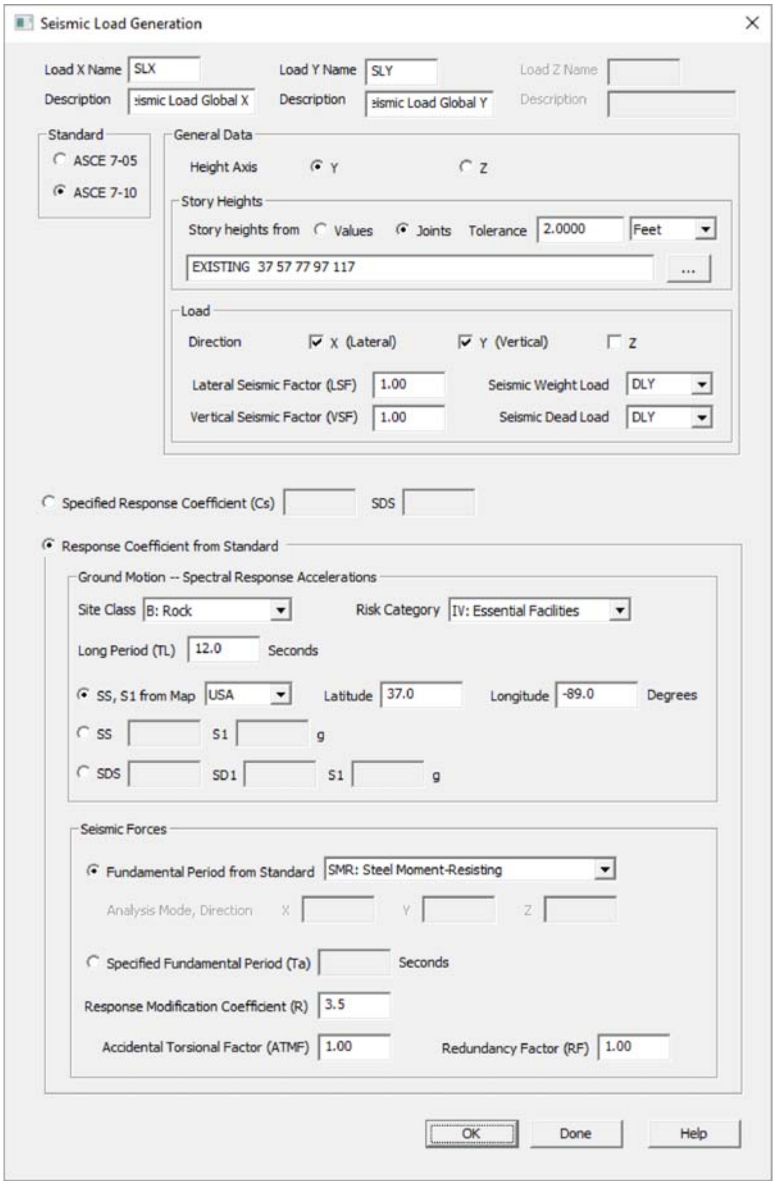


Accept SEISMIC LOAD Commands?

```
SEISMIC LOAD 'SLX' 'Seismic Load Global X' STANDARD ASCE7-10
 HEIGHT AXIS Y
 DIRECTION X
 SEISMIC WEIGHT LOAD 'DLY'
 SEISMIC DEAD LOAD 'DLY'
 STORY HEIGHTS JOINTS EXISTING EXISTING  37 57 77 97 117
 FLOOR TOLERANCE 2.0000
 SITE CLASS B
 RISK CATEGORY IV
 LONG PERIOD TL 12.0
 SPECTRAL ACCELERATION MAP USA LATITUDE 37.0 LONGITUDE -89.0
 FUNDAMENTAL PERIOD SMR
 RESPONSE MODIFICATION COEFFICIENT R 3.5
 ACCIDENTAL TORSION FACTOR ATMF 1.00
 LATERAL  SEISMIC FACTOR LSF 1.00
 VERTICAL SEISMIC FACTOR VSF 1.00
END OF SEISMIC LOAD DATA

SEISMIC LOAD 'SLY' 'Seismic Load Global Y' STANDARD ASCE7-10
 HEIGHT AXIS Y
 DIRECTION Y
 SEISMIC WEIGHT LOAD 'DLY'
 SEISMIC DEAD LOAD 'DLY'
 STORY HEIGHTS JOINTS EXISTING EXISTING  37 57 77 97 117
 FLOOR TOLERANCE 2.0000
 SITE CLASS B
 RISK CATEGORY IV
 LONG PERIOD TL 12.0
 SPECTRAL ACCELERATION MAP USA LATITUDE 37.0 LONGITUDE -89.0
 FUNDAMENTAL PERIOD SMR
 RESPONSE MODIFICATION COEFFICIENT R 3.5
 ACCIDENTAL TORSION FACTOR ATMF 1.00
 LATERAL  SEISMIC FACTOR LSF 1.00
 VERTICAL SEISMIC FACTOR VSF 1.00
END OF SEISMIC LOAD DATA
```

OK Cancel

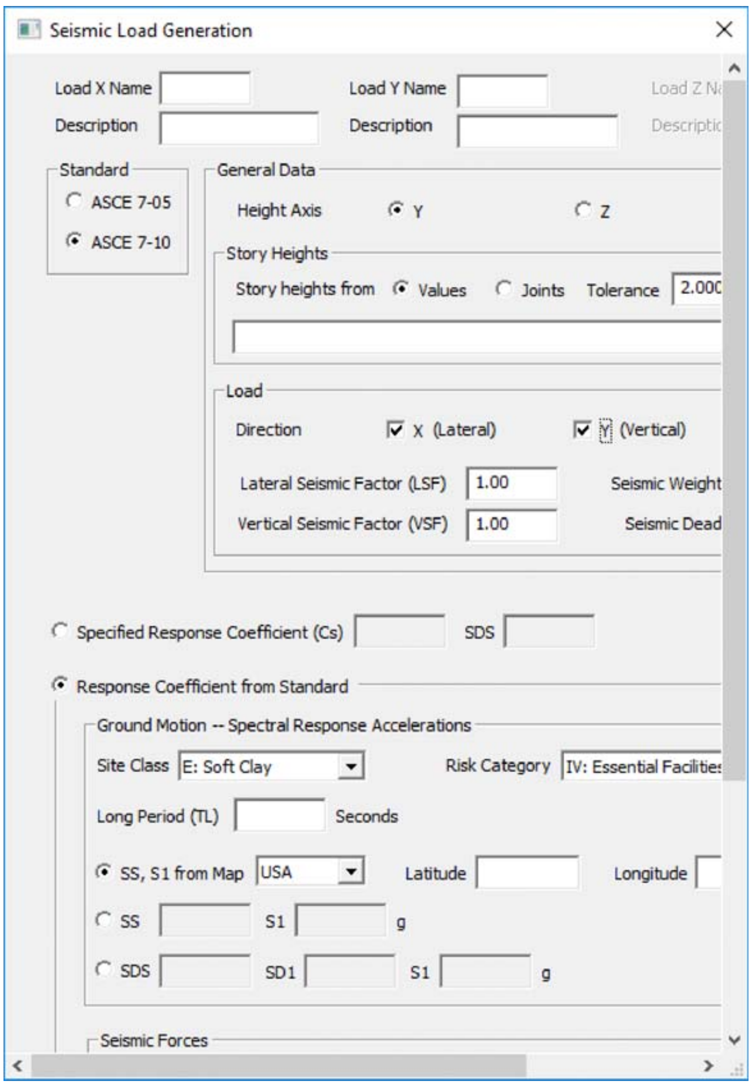
b. Users may select up to three seismic Directions in the Load sub-section: two lateral and one vertical Directions. If all three Directions are selected, the dialog will generate three SEISMIC LOAD command blocks. The following figure shows an example where two seismic Directions are selected:

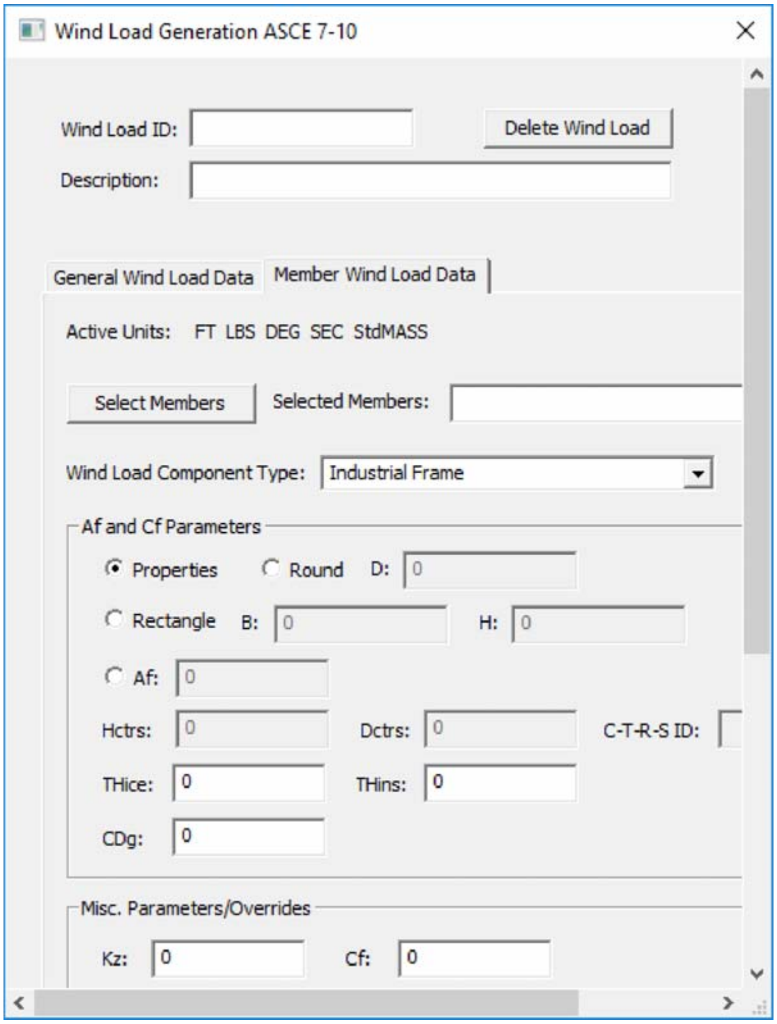


c. The Redundancy Factor (RF) edit box has been added to the Seismic Load Generation dialog in conjunction with the revised SEISMIC LOAD command option REDUNDANCY FACTOR RF. The edit box is located next to the Accidental Torsional Factor (ATMF) at the bottom of the above dialog. The default value of this new factor is set to 1.0.

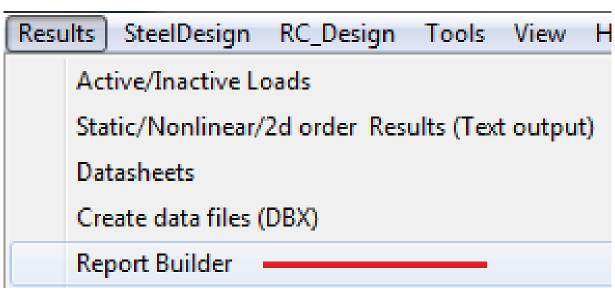
2. In the Seismic Load Generation dialog, the Load sub-section of the General Data section has been changed regarding Seismic Factors:
   a. The Lateral Seismic Factor (LSF) edit box has been added to the Load sub-section and can be used to scale a lateral seismic loading just as the Vertical Seismic Factor (VSF).
   b. The Vertical Seismic Factor (VSF) edit box has been moved from the Seismic Forces section to the Load sub-section and is placed below the Lateral Seismic Factor (LSF) edit box.

3. Both Seismic and Wind Load Generation dialogs are now capable of dialog resizing followed by scrolling so that both dialogs can be seamlessly used in displays of any resolution. The following two figures show the resizing and scrolling capabilities on Seismic and Wind Load Generation dialogs, respectively:

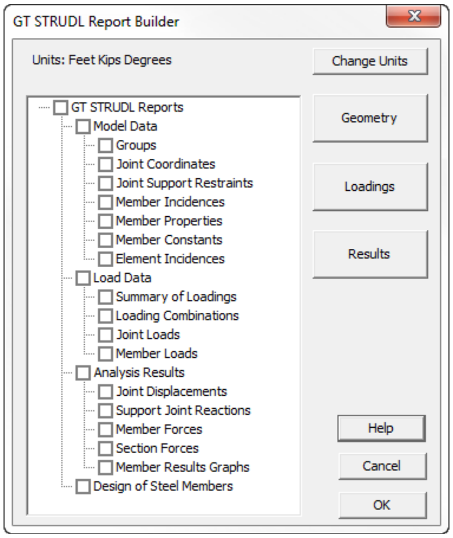




4. The new Report Builder dialog has been added to the Results menu,



This dialog allows you to select the type of data you want to include in a Report, writes the required DBX files in the proper format and then launches the GTSReportBuilder program which will automatically read the specified DBX files so you will not need to select them in the Report Builder.

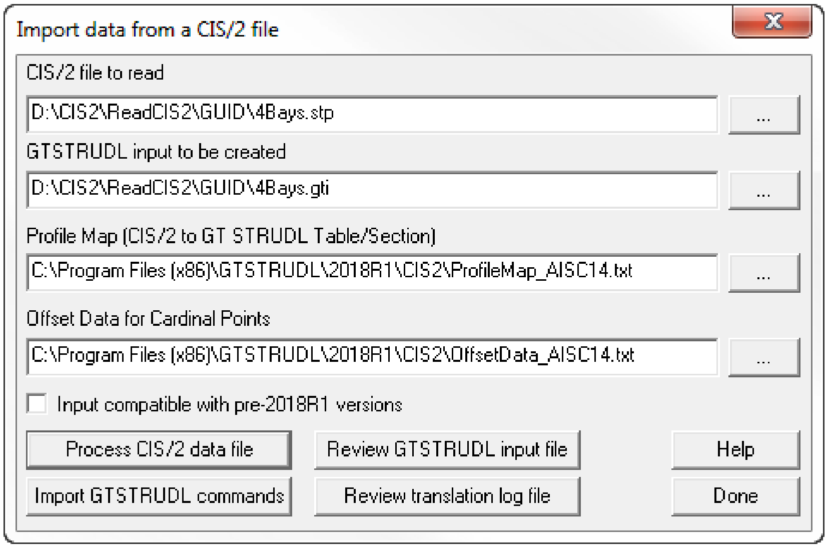


The Change Units button allows you to change the current units without leaving the dialog.

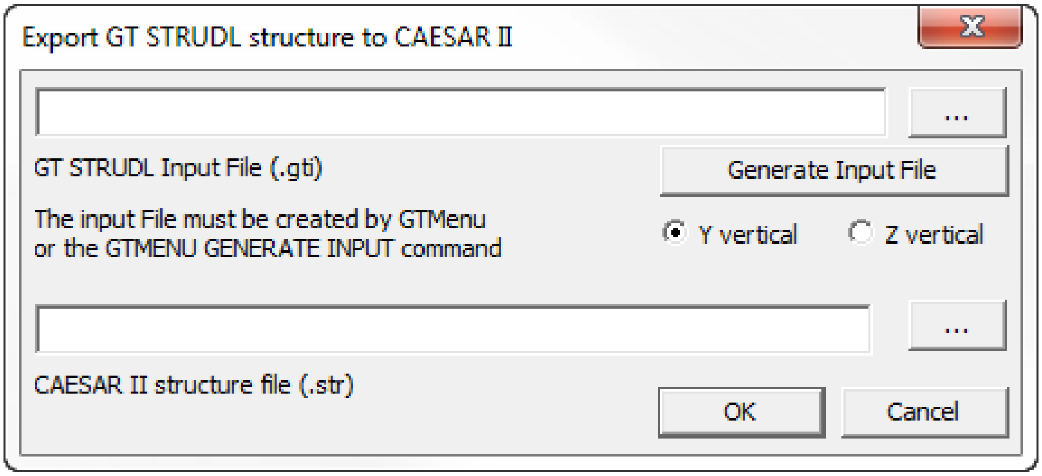
The Help button will open the Report Builder documentation to explain what the various data types mean.

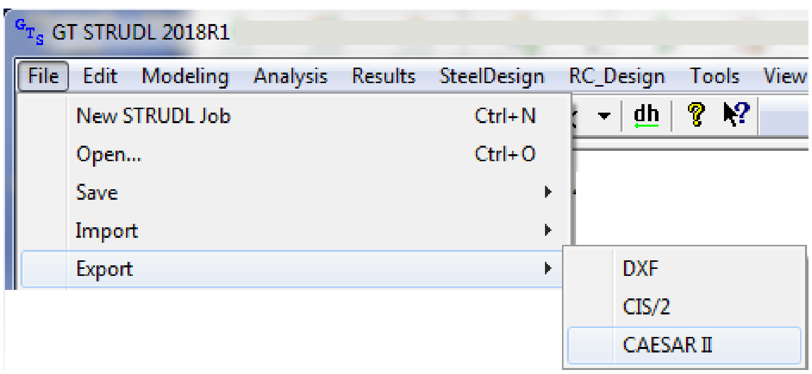
The Geometry, Loadings and Results buttons will check all the categories of that type. You can then un-check any data types you do not want to include.

5. The File - Import - CIS/2 dialog has new edit boxes to select a Profile Map and Offset Data files. By default, these files are set to the AISC 14th Edition files. Use the Help button to read more about how these files work.



6. A new prerelease Export –> CAESAR II dialog has been added to the File menu,





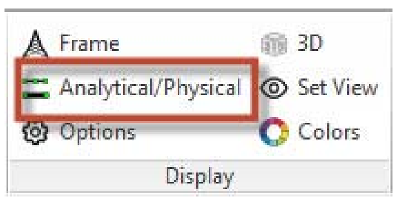
This feature will convert a GT STRUDL input file into a CAESAR II structure file. The input file must have been created in GTMenu or by using the GTMENU GENERATE INPUT command to ensure a readable (by the translator program) syntax. Note that the GT STRUDL input file must have a ".gti" extension and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

7. The Code Check datasheet now displays results when the MAXIMUM ENVELOPE VALUES option is used. Previously, code check results with this option would be omitted from the datasheet.

2.5 CAD Modeler

1. When placing physical members, the ID's of the physical members are automatically created as PMxxxxxx where xxxxxx is the next available integer. The first physical member created will be labeled PM1.

2. You are now able to select, label and copy physical members. You can split analytical members which are part of a physical member and the order of the analytical members in the physical member will automatically be adjusted as well as the member releases in the analytical members. The releases can only be at the start of the first analytical member and the end of the last analytical member in the physical member. A new options has been added to the Display panel which will allow you to easily switch between the Analytical and Physical display and labeling of members as shown below:

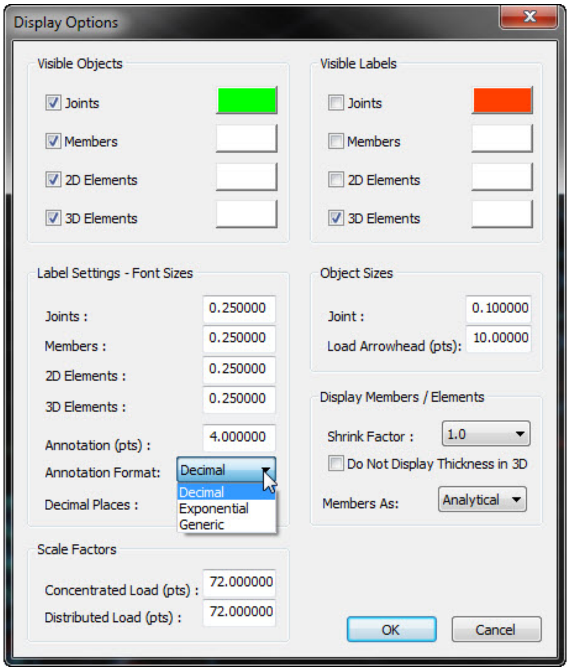


   The Options dialog also has an option to allow you to switch between Analytical and Physical display and labeling.

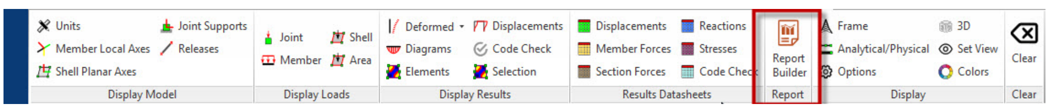
3. The load description is now shown in dialogs and in the legend that you can place on the display.

4. The Shrink option can now be used for wire frame displays. Previously, the Shrink option was available only for 3D (solid) displays.

5. You can now control the format of the annotation for results. From the revised Options dialog shown below, you can select from Decimal, Exponential or Generic formats. The Generic format will automatically switch between Decimal and Exponential based on the magnitude of the number to be displayed. You may also control the number of decimal places for the annotation.



5. The ability to use Report Builder directly from CAD Modeler has been added to the GTS Display ribbon as shown below:



Prior to using this option, you must have created a GTI file and Executed GT STRUDL. When Report Builder is selected, the DBX files needed to build the report are automatically written and Report Builder is started where you can select the data to include in your report.

6. The current drawing can be exported to into CAESAR II Modeler (str file) from the menu "GTS Modeling>CAESAR II> Export STR" or by typing GTSExportSTR at the command prompt. This feature is a prerelease feature. You select the members to be exported and an STR File is generated. Immediately after the log file appears on the screen. A typical log file is:

GT STRUDL Version 2018.R1

GTS2CII Version 2018.R1.01

GTS2CII Binary Dir

C:\\Program Files (x86)\GTStrudl\2018R1\Utilities\GTS2CII\

Project Dir F:\\HexagonPPM\CaesarII\PlantStructure\

Total Number of Sections: 6

Total Number of Joints: 170

Total Number of Members: 233

The model will be saved in 1 STR file(s)

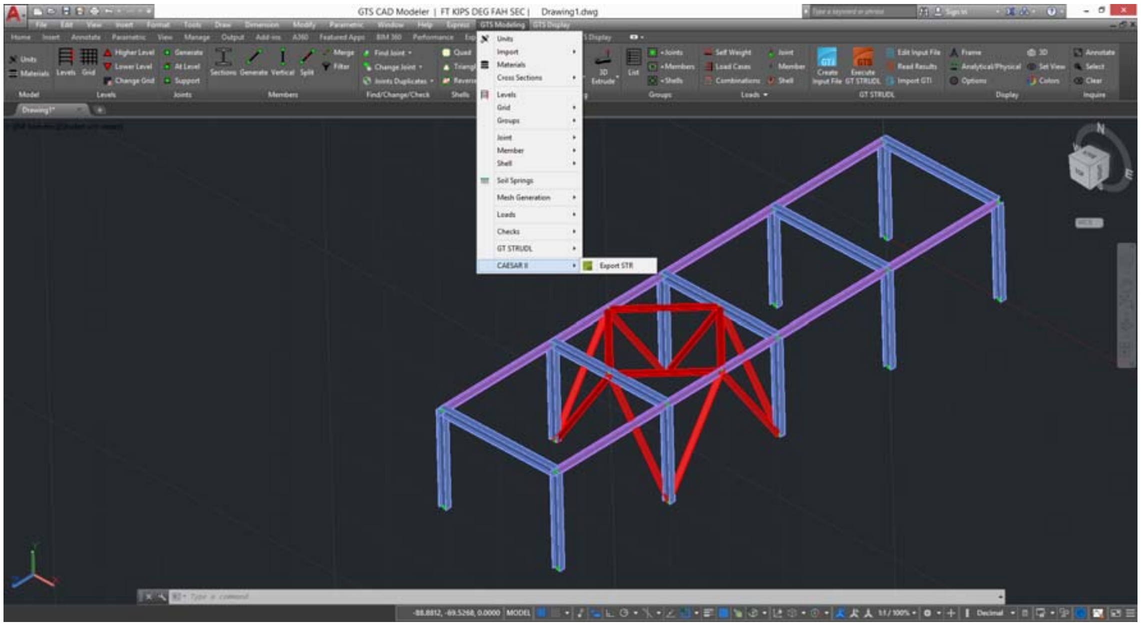
File

F:\\HexagonPPM\CaesarII\PlantStructure\\PStructure_0708_01.str created

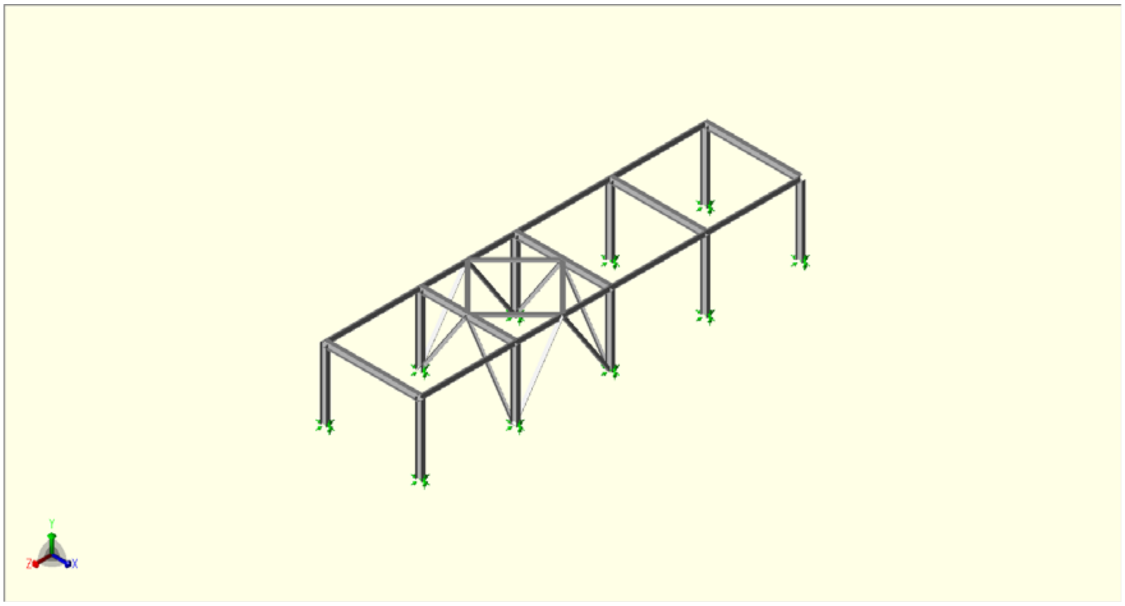
If the cross sections used in CAD Modeler (and GT STRUDL) are not available in CAESAR's section library, a warning message such as the one shown below will appear:

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

Shown below are images of the same structural model in CAD Modeler and in CAESAR II.



Structural model in CAD Modeler



Structural Model in CAESAR II

2.6 General

1. New commands have been added to create GUIDs associated with Physical Members. The ASSIGN GUID command will create the association. The PRINT GUID command will print the GUID / Physical Member associations.

Format:

ASSIGN GUID 'guid' (TO) PHYSICAL (MEMBER) id

ASSIGN GUID (TO) PHYSICAL (MEMBER) list

guid The GUID must contain 32 hexadecimal digits (0-9, A-F or a-f). Case and hyphenation are ignored. Braces {} are permitted, but not stored. When specified GUIDs are printed or exported, they are formatted to the standard 8-4-4-4-12 format in upper case without braces.

id The name of a single Physical Member as given in a DEFINE PHYSICAL MEMBER command or when created in GTMenu.

list A list of one or more Physical Members to receive a generated GUID.

Examples:

```
{   82} > ASSI GUID '{12345678-abcd-3456-CDEF-123456789012}' PHY 'PM1'

{   83} > ASSIGN GUID TO PHYSICAL MEMBERS 'PM2' TO 'PM5'
  EBD32B8D-C547-43D6-8A12-5B02AD09B18C assigned to Physical Member PM2
  86FA1C21-84DE-4DF8-A796-087CAF256C68 assigned to Physical Member PM3
  15414731-60CF-4673-84C9-59F2AE6862B4 assigned to Physical Member PM4
  4EF85EF2-23E2-49AA-93B6-7098F72CEE3B assigned to Physical Member PM5

{   83} > PRINT GUIDS

 Phys Memb  GUID
  --------  ------------------------------------
  PM1       12345678-ABCD-3456-CDEF-123456789012

 ****INFO_PrintGUIDS -    1 Physical Member GUIDS written.
```

Documentation:

Section 2.1.12.21 GUIDs in GT STRUDL, Volume 1, GT STRUDL Reference Manual

2. A new check has been added to Consistency Checking for models which contain finite elements. A check is now made to determine if an element has duplicate incidences. If duplicate incidences are detected, an error message is output indicating the first duplicate incidence of an element and then scanning mode is entered which prevents completion of the analysis.

2.7 DBX

1. A new WRITE GUIDS command has been added. This feature will write a file in ASCII80 or BINARY, SEQUENTIAL format containing the associations of Physical Members and GUIDS.

Example:

WRITE GUIDS PHYSICAL MEMBERS ‘PM1' TO ‘PM15'

Documentation:

Table 1-32, Volume 5, GT STRUDL Reference Manual.

2.8 Base Plate Wizard

1. The ACTIVE SOLVER command is now used when the GTSES analysis option (out of core solver) is requested.

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 GTSES option no longer produces an error when running analysis.
(GPRF 2018.02)

3.2 CAD Modeler

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

1. Member releases are no longer displayed at the opposite end of a member where they were specified when the orientation of the member is opposite from the text/label orientation of the releases as they will be displayed.

2. The pulldown menu for the contouring of finite element resultants had Vxx incorrectly listed twice. The second Vxx in the pulldown has been changed to Vyy.

3.3 Finite Elements

1. The TRIP finite element now produces correct results when the element is tapered between the triangular faces. Previously, the element would produce correct results only if the triangular area varied linearly between the two triangular faces.
(GPRF 2018.01)

2. The 20 node solid IPQS element will now produce correct results when Local surface force loadings are applied to curved faces of the element. In addition, correct results will be computed if multiple local loads are applied to the same face of an element in the same loading.
(GPRF 2018.04)

3. When identical coordinates are specified for quadrilateral element nodes on elements such as the SBHQ, SBHQ6, PSHQ and BPHQ, an abort no longer occurs. The problem nodes are now reported and SCAN mode is entered. Although previously an abort did not always occur, depending on which nodes had identical coordinates, the reported error message of “ELEMENT HAS ZERO AREA” has been replaced with a more informative message with the problem node names.
(GPRF 2018.07)

3.4 GTMenu

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

1. An abort no longer occurs when elements have been defined before any members, and the Member Graph feature is used.

2. GTMenu will now display correct Member End Forces/Moments. In some instances, incorrect values were displayed for the selected members.

3. An abort will no longer occur when Member End Forces/Moments or Member Graphs are requested for newly created members which do not have results.

4. You are now able to select objects in a view when the dialogs are detached from the Main Window.

5. An abort no longer occurs when using the Contour Modes Dialog when the previously selected stress does not contain valid limits.

6. The Weight and Center of Gravity option in the Check Model dialog will now produce the correct weight and center of gravity for 3D solid elements IPLS, IPSL, IPSLIM, IPQS, IPSQ, TRIP and WEDGE15.

7. The Check Loads option in the Check Model dialog will now produce the correct summation of body and surface loads for 3D solid elements IPLS, IPSL, IPSLIM, IPQS, IPSQ, TRIP and WEDGE15.

8. The Check Loads and Weight and Center of Gravity options in the Check Model dialog will now produce warning messages for the 2D and 3D transition elements:
   a. For the 2D IPQLQ1, IPQLQ2, IPQLQ2B and IPQLQ3 elements, the Check Loads option will assume that the edges of these elements are straight for the body and surface load calculations and for the Check Weight and CG option.

Warning messages will be output in the Output dialog regarding the above.

b. For the 3D TRANS3D element, the Check Loads option will assume that the edges of the element are straight for the uniform surface loads and for body load calculations and for the Check Weight and CG option. Variable surface loads on faces of the TRANS3D element will be ignored in the Check Loads option. Warning messages will be output in the Output dialog regarding the above.

9. The Check Loads option in the Check Model dialog will now produce the correct summation of edge loads applied to the 2D isoparametric elements IPLQ, IPQL, IPQQ, IPQLQ4 and IPCQ.

10. Redraw Solid will now always draw members. Previously, under certain orientations of a member or when duplicate members existed, Redraw solid would not draw the members.

11. The Generate Input File function will now produce correct variable edge loads and variable surface forces on models which contain elements with different numbers of nodes such as a mixture of 4 and 8 node plane stress elements or 8 and 20 node solid elements.

12. An abort will no longer occur when the Remaining Spaces is set to zero in the Split Members Dialog.

15. An infinite loop will no longer occur when placing members using the Start, Length, & Direction placement method in List Mode.

16. The list of IDs given by user under List mode are no longer sorted or compacted for certain features for which the order of the IDs is important such as Place Members.

17. The Extrude Dialog now correctly assigns the Material and Section properties for Downward Columns, and Connected Members as specified by the user in the corresponding fields. Also, an abort which could happen when the downward column option is used under certain circumstances has been fixed.

18. An abort will no longer occur under certain circumstances while finding duplicate members or elements.

19. An abort will no longer occur when closing GTMenu while animation is running.

20. An abort will no longer occur when exiting GTMenu after deleting all members.

21. Member release labels will no longer be displayed when GTMenu is entered but no members existed.

22. Member material and sections are now applied correctly when placing members in List or Coordinate mode and using the Split option.

23. An abort will no longer occur when new joints have been deleted in GTMenu and GTMenu was closed without creating members or elements.

24. When editing Member Data, the correct dialog will now be displayed for the property being edited.

25. An abort will no longer occur in the Refine Mesh Dialog while storing new elements if you have previously canceled the Deletion of Duplicate Joints in the Check Duplicate Objects and Floating Joints dialog.

26. An abort will no longer occur when you are deleting duplicate members using the Delete Selection button.

27. The display of members will no longer change to dashed lines when you stop the animation of mode shapes after you have consecutively changed the mode shape number being animation.

28. After a joint is deleted in GTMenu, an abort will no longer occur in the story height calculation for a seismic load.

28. The combo box titles in the Mode Shape and Static Joint Displacement dialogs are now updated in real time when you change the mode or loading using the arrow keys while animating the results.

30. The Deformed Structure and Transient Response options are now grayed out and not available for selection if the results needed for these options are not available.

31. Members with eccentricities are now split when using using the Split Framing Members option in the Refine Finite Element mesh dialog.

32. The scroll bar in the the View, Member Forces, Member End Forces and Groups dialog now allows you to access all of the button in the dialogs.

33. Redraw Solid now draws the complete model if a physical member does not have any properties or has prismatic properties.

34. An abort will no longer occur during Copy Model if a model contains members and elements which are loaded and multiple copies are performed.

35. An abort will no longer occur if you select No when asked “Do you want to keep the Loads.”

36. The loads are now correct when Refining a Finite Element Mesh and you selected the option to Split Framing Members.

37. An abort will no longer occur when displacing the deformed structure and you right click on the mouse to perform an inquire on a member.

38. After deleting Interference Joints and then performing a Check Interference Joints, the members corresponding to the deleted joints are no longer listed in the Check Interference Joint dialog.

39. The default name for member (now called section) properties is now limited to 8 characters. Previously, a user could enter more than 8 characters but the name was truncated to 8.

40. An abort will no longer occur when you select Deform Between Joints and the structure does not contain any members in the Static Joint Displacements and Moving Load dialogs.

41. If a Group or Physical Member does not have a Description and you select Refine Finite Element Mesh with the option to Change to a Higher Order Element, the Group Description will still be blank.

42. In the Create/Edit dialog if you click Joints, Members and then Elements buttons to display each Group, the display on the model is now updated to reflect your selection.

43. The section property type (table, prismatic, pipe) is now displayed only once in the Inquire Output dialog.

44. In the Display Load dialog, you will no longer be allowed to enter a value for the Load Scale Factor which is out of the range (0.1 - 5.0) allowed in the dialog. If you enter a value outside of this range, a pop-up will inform you that you must enter a value in the range.

45. When you double click on a joint coordinate in the Inquire Output dialog and select the option to Respecify the coordinates, the existing joint coordinates are displayed and an erroneous message indicating that multiple joints are selected no longer occurs. You can also now respecify the coordinates after selecting the Inquire button and then Joint Coordinates in the Edit Joint Data dialog followed by double click on the coordinates in the Inquire Output dialog.

46. The display of dynamic mode shapes with the Undeformed Structure with Mode Shape Overlay option selected now displays both the mode shape and the undeformed structure before and after windowing in on the model.

47. The All option is now available on the Mode Bar in the Display Model dialog after creating a Global Plane view.

48. The gti file will no longer having missing finite elements after performing a Refine Finite Element Mesh.

49. Steel Design Parameters will now be correct in the gti file for models which contain a mixture of integer and alphanumeric member Ids.

50. An erroneous error message will no longer occur after you have deleted 8 Construction Lines. Previously after deleting 8 Lines and you selected the option Create Joints and Elements Using Construction Lines, an erroneous message would pop-up indicating that at least 2 curves or a surface were required.

3.5 Analysis

1. A GTSES/GT64M nonlinear static analysis will no longer abort prematurely and /or erroneously for large jobs for which the product NDOF*NLD exceeds 4 million, where NDOF = the number of model degrees of freedom and NLD = the number of independent loading conditions active for the analysis.
   (GPRF 2018.03)

2. Linear and nonlinear static analyses now produce correct results for the case when concentrated member loads are applied at points within the length of member end sizes at the start of a member.
   (GPRF 2018.05)

3.6 General

1. The GLOBAL option for LIST FORCES, LIST JOINT FORCES and WRITE MEMBER RESULTS no longer has the possibility of producing incorrect output when RMS or ABSOLUTE loadings are active.
   (GPRF 2018.06)

2. If a Group is deleted, the group title is no longer printed when using the PRINT GROUP command.
   (No GPRF issued)

3. The Physical Member Compliance Check command now checks that the material properties are the same for all analytical members that comprise a physical member.
   (No GPRF issued)

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. Projected element loads will be displayed incorrectly when they are created or when

they are displayed using Display Model → Loads.

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

(No GPRF issued)

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

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

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

7. The Label Structural Attributes options in the Label Settings dialog will not display if the Inquire Output dialog is open. For instance, if you have checked the Support Status option in Label Structural Attributes, the legend for the support status will disappear if the Inquire Output dialog is open.

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

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

5.2 Design Prerelease Features

5.2.1 A new national annex parameter for EC3-2005 steel design code

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

Table 1.3-1

EC3-2005 Code Parameters

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

Table 1.3-7

Country Names and the National Annex Parameter Values

Country[1]	National Annex Parameter Values
EC3-2005 (defaults)	GM0 = 1.0, GM1 = 1.0, GM2 = 1.25 Beta = 0.75, LamdaLT0 = 0.4
Cyprus, **Greece**, **Netherlands**[2], **Slovenia**, **Spain**, and **Sweden** use above EC3-2005 default values	
Belgium	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Bulgaria	GM0 = 1.05, GM1 = 1.05
Denmark	GM0 = 1.1, GM1 = 1.2, GM2 = 1.35
Finland	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
France	For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Germany	GM1 = 1.1
Italy	GM0 = 1.05, GM1 = 1.05 Also see Table 1.3-8 for lateral torsional buckling curve for cross-sections using equation (6.57) of the 1993-1-1:2005(E)
Malaysia	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
Norway	GM0 = 1.05, GM1 = 1.05
Poland	GM2 = 0.9(f_u / f_y) $\geq$ 1.1

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

1 The country names that are not listed in Table 1.3-7 are accepted but a warning message is given that the EC3-2005 default values are used.

2 Country names more than 8 characters are stored and displayed based on the first 8 characters.

Table 1.3-7 (continued)

Country Names and
the National Annex Parameter Values

Country[1]	National Annex Parameter Values
EC3-2005 (defaults)	GM0 = 1.0, GM1 = 1.0, GM2 = 1.25 Beta = 0.75, LamdaLT0 = 0.4
Portugal	Beta = 1.0, LamdaLT0 = 0.2
Singapore[2]	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2
UK (United Kingdom)	GM2 = 1.1 For welded cross-sections, Parameter SECTYPE = WELDED Beta = 1.0, LamdaLT0 = 0.2 Also see Table 1.3-9 for lateral torsional buckling curve for cross-sections using equation (6.57) of the 1993-1-1:2005(E)

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

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

Table 1.3-8

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

Cross-section	Limits	Buckling curve	α_{LT}
Rolled I cross-sections	h/b ≤ 2	b	0.34
	h/b > 2	c	0.49
Welded I cross-sections	h/b ≤ 2	c	0.49
	h/b > 2	d	0.76
For all other cross-sections		d	0.76

Table 1.3-9

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

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

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

The goal of the DESIGN SLAB command is to select reinforcing steel for concrete flat plate systems using finite elements as a tool for the determination of design moments.

Instead of dealing with results on an element-by-element basis, the user will be able to design the reinforcing steel for slab systems based on cuts. Here, the term *cut* refers to the cross-section of a strip at a particular location to be designed. A cut is defined by two nodes identifying the start and end of the cut, and by an element in the plane of the cut.

Once the definition of the cut has been determined, the resultant forces along the cut are computed using either moment resultants (otherwise known as the Wood and Armer method) or element force results (using the CALCULATE RESULTANT command, as described in Section 2.3.7.3 of Volume 3 of the Reference Manuals). The final design moment is determined by computing the resultant moment acting on the cut for each loading condition, and reducing these moments to a design envelope.

Once the design envelope is computed, the cross-section is designed according to ACI 318-05 either using default design parameter or with certain user specified design parameters such as the bar size or spacing.

An important distinction is to note that each cut is designed independently from all other cuts. That is, a cut specified in one region is independent with respect to a design in another region. As such, if the user wishes to use the same bar size over multiple adjacent cuts, this information must be specified for each cut.

The form of the command is as follows:

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

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

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

$$^{*}\left\{\begin{array}{l} \underline{\text{TOP}}\text{ (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S i}_2\text{) (}\underline{\text{SPA}}\text{CING v}_1\text{)} \\ \underline{\text{BOT}}\text{TOM (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S i}_3\text{) (}\underline{\text{SPA}}\text{CING v}_2\text{)} \\ \underline{\text{BOTH}}\text{ (}\underline{\text{FACE}}\text{S) (}\underline{\text{BAR}}\text{S i}_4\text{) (}\underline{\text{SPA}}\text{CING v}_3\text{)} \end{array}\right\}\text{–}$$

$$\left\{\begin{array}{l} \to \underline{\text{IN}}\text{NER (}\underline{\text{LAY}}\text{ER)} \\ \quad\underline{\text{OUT}}\text{ER (}\underline{\text{LAY}}\text{ER)} \end{array}\right\}\text{ (}\underline{\text{COV}}\text{ER v}_4\text{) (}\underline{\text{LIN}}\text{EAR (}\underline{\text{TOL}}\text{ERANCE) v}_5\text{) –}$$

$$\text{(}\underline{\text{TOR}}\text{SIONAL (}\underline{\text{MOM}}\text{ENT) (}\underline{\text{WARN}}\text{ING) v}_6\text{)}$$

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.

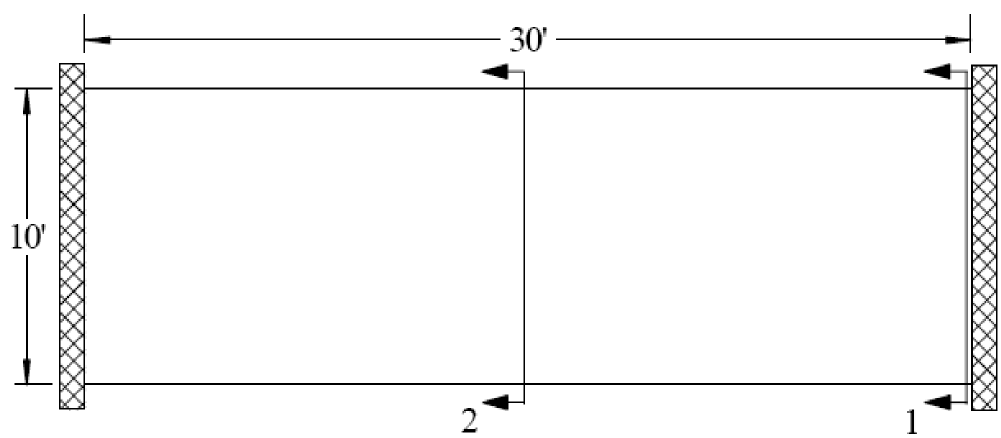


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

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.

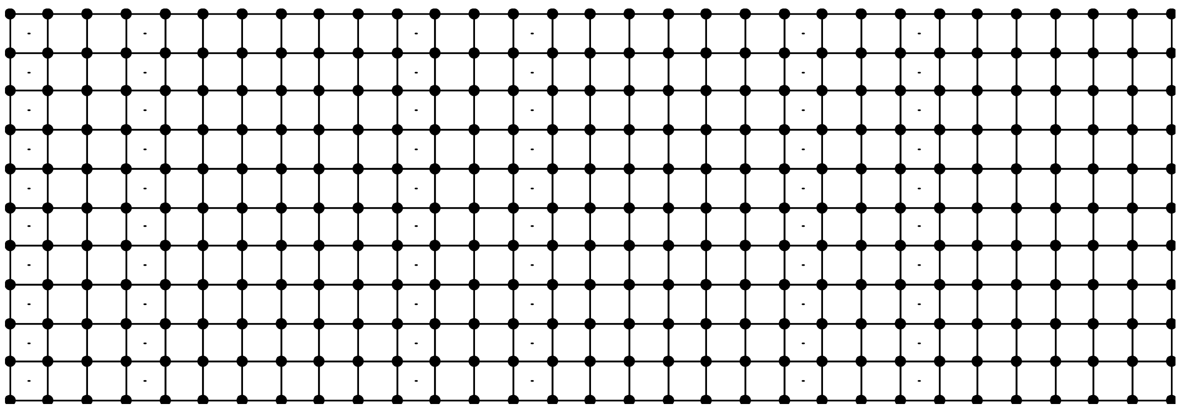


Figure 5.2.3-2 Example Finite Element Model

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:

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

$$* \left\{ \begin{array}{l} \text{\underline{ENERGY} (\underline{NORM})} \\ \text{\underline{MAX} \underline{DIFF}ERENCE} \\ \text{\underline{DIFF}ERENCE \underline{FROM} \underline{AVER}AGE} \\ \text{\underline{PERCENT} \underline{MAX} \underline{DIFF}ERENCE} \\ \text{\underline{PERCENT} \underline{DIFF}ERENCE \underline{FROM} \underline{AVER}AGE} \\ \text{\underline{NORM}ALIZED \underline{PERCENT} \underline{MAX} \underline{DIFF}ERENCE} \\ \text{\underline{NORM}ALIZED \underline{PERCENT} \underline{DIFF}ERENCE \underline{FROM} \underline{AVER}AGE} \end{array} \right\} -$$

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

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

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

Difference from Average Method

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

Percent Maximum Difference Method

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

Percent Difference from Average Method
Normalized Percent Maximum Difference

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

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

Normalized Percent Difference from Average Method

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

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

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

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

1. Enter GTMenu.

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

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

4. Select the Loading.

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

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

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

General form:

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

Explanation:

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

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

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


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

   PROBLEM - None

   ACTIVE UNITS  FEET KIP  RAD  DEGF SEC
```

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



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

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

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

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

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

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

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:

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

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

Explanation:

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

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

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

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

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

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

In the second case, three joints are required. Each of the three joints may be defined either by a joint identifier using the JOINT option of the command or by its global X, Y, and Z coordinates. If the joint identifier option is used, however, the coordinates of the joint must be specified previously by the JOINT COORDINATES command. The first time (i_2 or 'a_2' or v_4, v_5, and v_6) defines the origin of the reference system; the X-axis of the reference system is determined by the first and second joints (i_3 or 'a_3' or v_7, v_8, and v_9). The positive X-axis is directed from the first to the second joint. The third joint (i_4 or 'a_4' or v_{10}, v_{11}, and v_{12}) is used to define the XY-plane of the reference system. The positive Y-axis is directed toward the third joint. The Z-axis then is determined by the right-hand rule.

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

Modifications of Reference Systems:

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

Examples:

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

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

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

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

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

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

d) CHANGES
REFERENCE COORDINATE SYSTEM 'FLOOR2' -
ORIGIN 10 20 30
ADDITIONS

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

e) DELETIONS
REFERENCE SYSTEM 2
ADDITIONS

The above command deletes Reference System 2.

5.4.2-1 Printing Reference Coordinate System Command

General form:

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

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:

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

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

.

.

.

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

Elements:

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

$\text{ncidence-specs} \ = \ \{\ \text{point}_1\text{'}\ \}\ \ \{\ \text{'point}_2\text{'}\ \}.\ .\ .\ \ \{\ \text{'point}_p\text{'}$

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:

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

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

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.

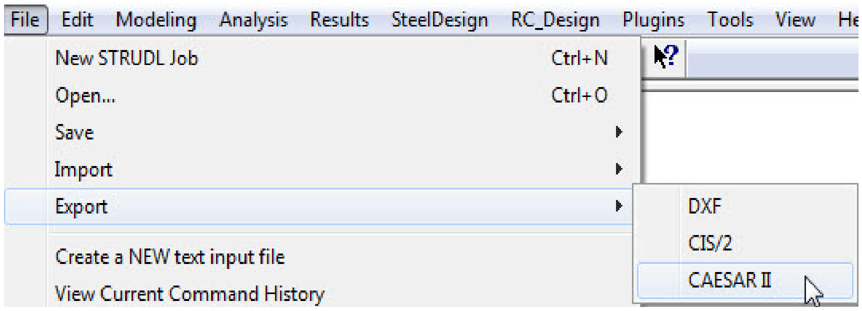
Examples:

```
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'
```

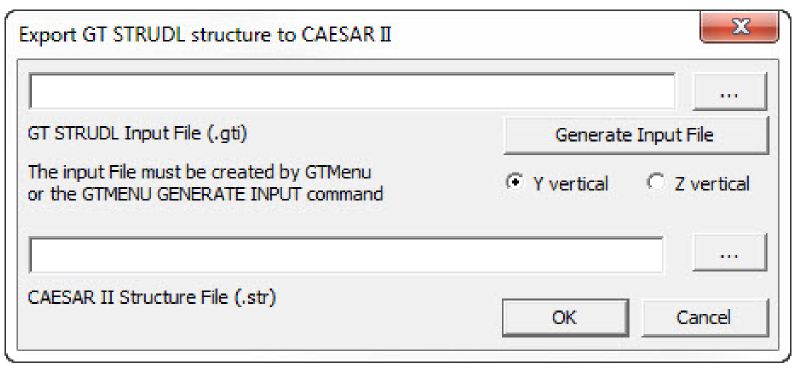
```
'S3' SURFACE OF REVOLUTION SPACING 10 20 -
    REVOLUTION ANGLE 360 -
    AXIS COORDINATES START  10.0 0.0 10.0 -
                     END    20.0 0.0  0.0 -
    U CURVE 'C2'
```

5.4.5 Export to CAESAR II

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

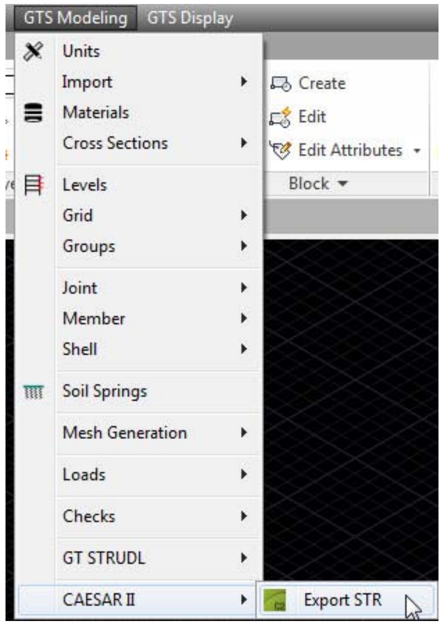


After selecting CAESAR II, the following dialog will pop-up:



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

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



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

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

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

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

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