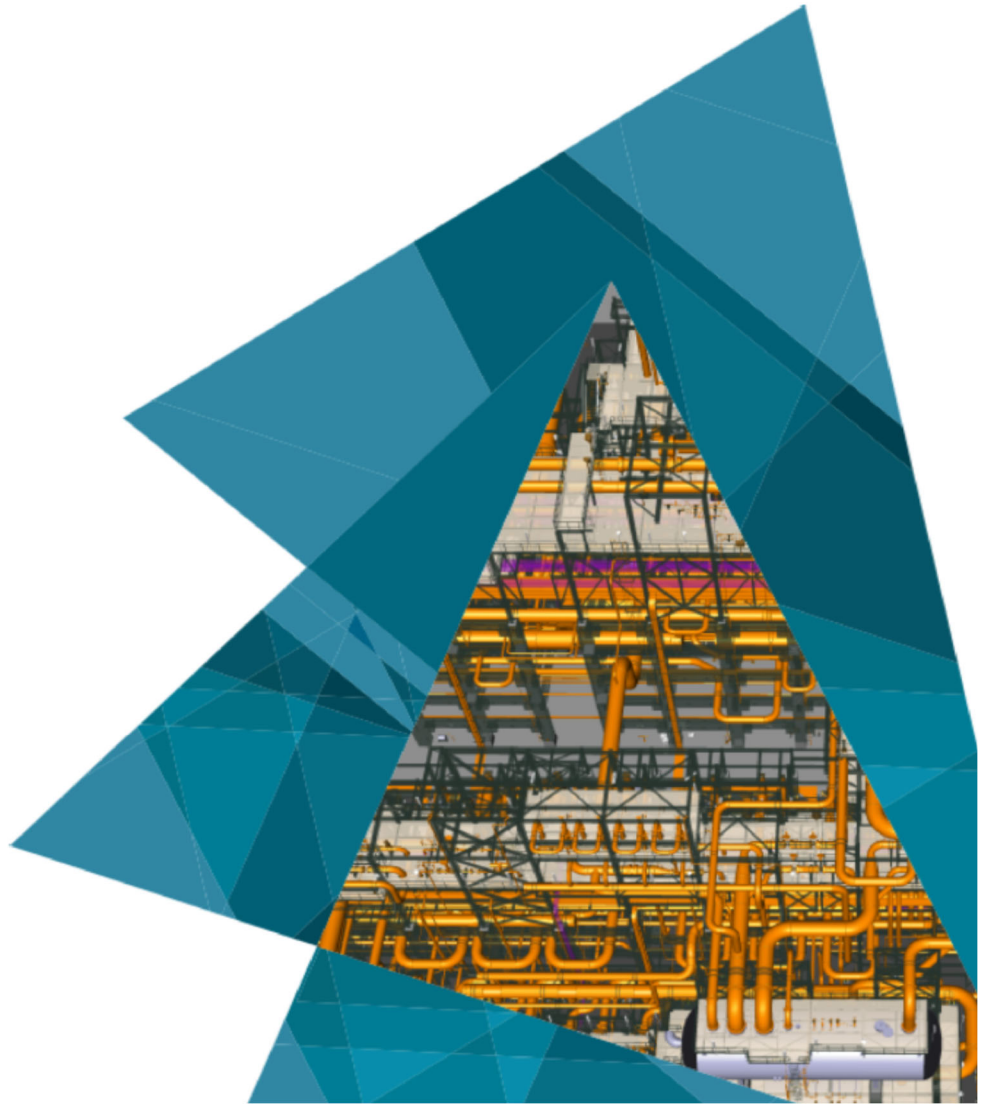


GT STRUDL[®] Version 43

User Guide



CAD Modeler
Getting Started Guide

Release Date: November 2024



Notice

This CAD Modeler Getting Started Guide is applicable to GT STRUDL Version 43 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

Copyright © 2024 Hexagon and/or its subsidiaries and affiliates. All rights reserved.

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

U.S. Government Restricted Rights Legend

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- Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States.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

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
305 Intergraph Way
Madison, AL 35758

Documentation

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

Other Documentation

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

Terms of Use

- a. Use of a software product and Documentation is subject to the Software License Agreement ("SLA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.
- b. For use of Documentation or Other Documentation where end user does not receive a SLA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial use. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

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

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

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

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

Limitation of Damages

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

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

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

Export Controls

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

- a. to Cuba, Iran, North Korea, the Crimean region of Ukraine, or Syria, or any national of these countries or territories.
- b. to any person or entity listed on any United States government denial list, including, but not limited to, the United States Department of Commerce Denied Persons, Entities, and Unverified Lists, the United States Department of Treasury Specially Designated Nationals List, and the United States Department of State Debarred List. Visit www.export.gov for more information or follow this link for the screening tool: <https://legacy.export.gov/csl-search>.
- c. to any entity if Customer knows, or has reason to know, the end use of the software product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.
- d. to any entity when Customer knows, or has reason to know, that an illegal reshipment will take place.

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

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

Trademarks

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

NOTICES	iii
Table of Contents	v
1. Getting Started	10
1.1. Introduction.....	10
1.2. Installing CAD Modeler under Windows 11	11
2. Using CAD Modeler	17
2.1. Overview of Using CAD Modeler and configuring AutoCAD/BricsCAD	17
2.2. Running CAD Modeler	17
2.3. Menu Bar and Ribbon Area	18
2.4. AutoCAD/BricsCAD Commands.....	19
2.5. AutoCAD/BricsCAD Drawing Units	20
2.6. CAD Modeler Commands	21
2.6.1. Units.....	21
2.6.2. Materials.....	22
2.6.3. Levels	22
2.6.4. Grid	24
2.6.5. Creating Joints	25
2.6.6. Finding Joints	26
2.6.7. Joint Supports.....	26
2.6.8. Joint Properties.....	26
2.6.9. Sections	27
2.6.10. Creating Members.....	30
2.6.11. Finding Members.....	31
2.6.12. Splitting Members	32
2.6.13. Splitting to Crossing Members	32
2.6.14. Merging Members	32
2.6.15. Member Properties	32
2.6.16. Member Filters	36
2.6.17. Creating Shell Finite Elements.....	37
2.6.18. Reverse Incidence Order	38
2.6.19. Finding Shells	38

2.6.20.	Shell Properties.....	38
2.6.21.	Joints Duplicates.....	39
2.6.22.	Joints Floatings	40
2.6.23.	Joints Interference.....	40
2.6.24.	Members Duplicates.....	41
2.6.25.	Members Zero Length	41
2.6.26.	Physical Members.....	41
2.6.27.	Shells Duplicates.....	42
2.6.28.	Names Duplicates	43
2.6.29.	Renumber Names	43
2.6.30.	Database Integrity	43
2.6.31.	Meshing along a curve.....	44
2.6.32.	Meshing between two lines	46
2.6.33.	Meshing between four lines.....	46
2.6.34.	Meshing inside a polyline	46
2.6.35.	Meshing by extruding a polyline	48
2.6.36.	Meshing using 3 curves	48
2.6.37.	Array 3D Advanced	48
2.6.38.	Soil Springs.....	49
2.6.39.	Export to CAESAR II.....	50
2.6.40.	Convert Lines/Polylines to Members/Shells	50
2.6.41.	Model Wizard	51
2.6.42.	Groups	52
2.6.43.	Self - Weight	53
2.6.44.	Load Cases	53
2.6.45.	Joint Loads	54
2.6.46.	Member Loads.....	55
2.6.47.	Shell Loads	57
2.6.48.	Area Load.....	57
2.6.49.	Wind Load ASCE 705.....	59
2.6.50.	Wind Load ASCE 710.....	61
2.6.51.	Wind Load ASCE 716.....	63
2.6.52.	Wind Load ASCE 722.....	65

2.6.53.	Seismic Load	67
2.6.54.	Load Combinations	68
2.6.55.	Standardized Combinations	69
2.6.56.	Steel Design Parameters.....	70
2.6.57.	Create GTI	71
2.6.58.	Edit GTI	72
2.6.59.	Execute GT STRUDL	72
2.6.60.	Read Analysis Results	73
2.6.61.	Import GTI.....	74
2.6.62.	Set Views	74
2.6.63.	3D or Wireframe View of the Structure	74
2.6.64.	Analytical/Physical Member View	75
2.6.65.	Colors and Visible Elements	75
2.6.66.	Display Options.....	77
2.6.67.	Annotate	78
2.6.68.	Select CAD Modeler's entities	79
2.6.69.	Display Member Local Axes.....	80
2.6.70.	Display Member Releases	80
2.6.71.	Display Shell Planar Axes	80
2.6.72.	Display Joint Supports	80
2.6.73.	Display Joint Loads	81
2.6.74.	Display Member Loads	81
2.6.75.	Display Shell Loads	82
2.6.76.	Display Area Loads.....	83
2.6.77.	Display Deformed Structure	83
2.6.78.	Annotate Joint Displacements.....	84
2.6.79.	Display Displacements.....	84
2.6.80.	Display Member Diagrams	85
2.6.81.	Display Finite Element Results.....	86
2.6.82.	Display Finite Element Selection Results.....	88
2.6.83.	Display Member Code Check Results	88
2.6.84.	Results Datasheets	89
2.6.85.	Report Builder	90

2.6.86.	Clear Results Layer	90
2.6.87.	Version.....	90
3.	Tutorial Example #1.....	91
3.1.	Introduction.....	91
3.2.	Open CAD Modeler and start working	91
3.3.	Define the basic geometry of the model.....	92
3.4.	Create the 1 st floor.....	97
3.5.	Create the 2 nd floor.....	107
3.6.	Create the 3 rd floor	109
3.7.	Create bracing	112
3.8.	Create girders	117
3.9.	Create an opening	122
3.10.	Create Supports.....	123
3.11.	Check the model.....	124
3.12.	Define Groups.....	125
3.13.	Define Loads	128
3.14.	GT STRUDL Input File.....	141
3.15.	Display Results.....	144
3.16.	Results Datasheets	152
3.17.	Report Builder	154
4.	Tutorial Example #2.....	156
4.1.	Introduction.....	156
4.2.	Open CAD Modeler and start working	157
4.3.	Define the basic geometry of the model.....	157
4.4.	Create the bottom of the tank	159
4.5.	Create the walls of the tank	161
4.6.	Create Supports.....	168
4.7.	Check the model.....	170
4.8.	Define Groups.....	171
4.9.	Define Loads	174
4.10.	Create GT STRUDL Input File	179
4.11.	Display Results.....	183
4.12.	Results Datasheets	189

4.13.	Report Builder	190
5.	Tutorial Example #3.....	191
5.1.	Introduction.....	191
5.2.	Open CAD Modeler and start working	191
5.3.	Define the basic geometry of the model.....	192
5.4.	Create Columns	196
5.5.	Create beams and girders.....	198
5.6.	Create girders	204
5.7.	Define supports	224
5.8.	Define Loads	225
5.9.	Perform analysis	226
5.10.	Read analysis results.....	233
5.11.	Display analysis results	233
6.	Appendix – List of Commands	243

GT STRUDL® CAD MODELER

Getting Started Guide

1. Getting Started

CAD Modeler is an add-on to BricsCAD® or AutoCAD®, which allows you to create GT STRUDL Input Files (GTI) graphically using their powerful CAD tools and graphical display capabilities and also graphically review GT STRUDL results from an analysis and steel code check. AutoCAD® or BricsCAD® must be installed in your computer before installing and running CAD Modeler. It is highly recommended that you have AutoCAD/BricsCAD experience before using CAD Modeler.

1.1. Introduction

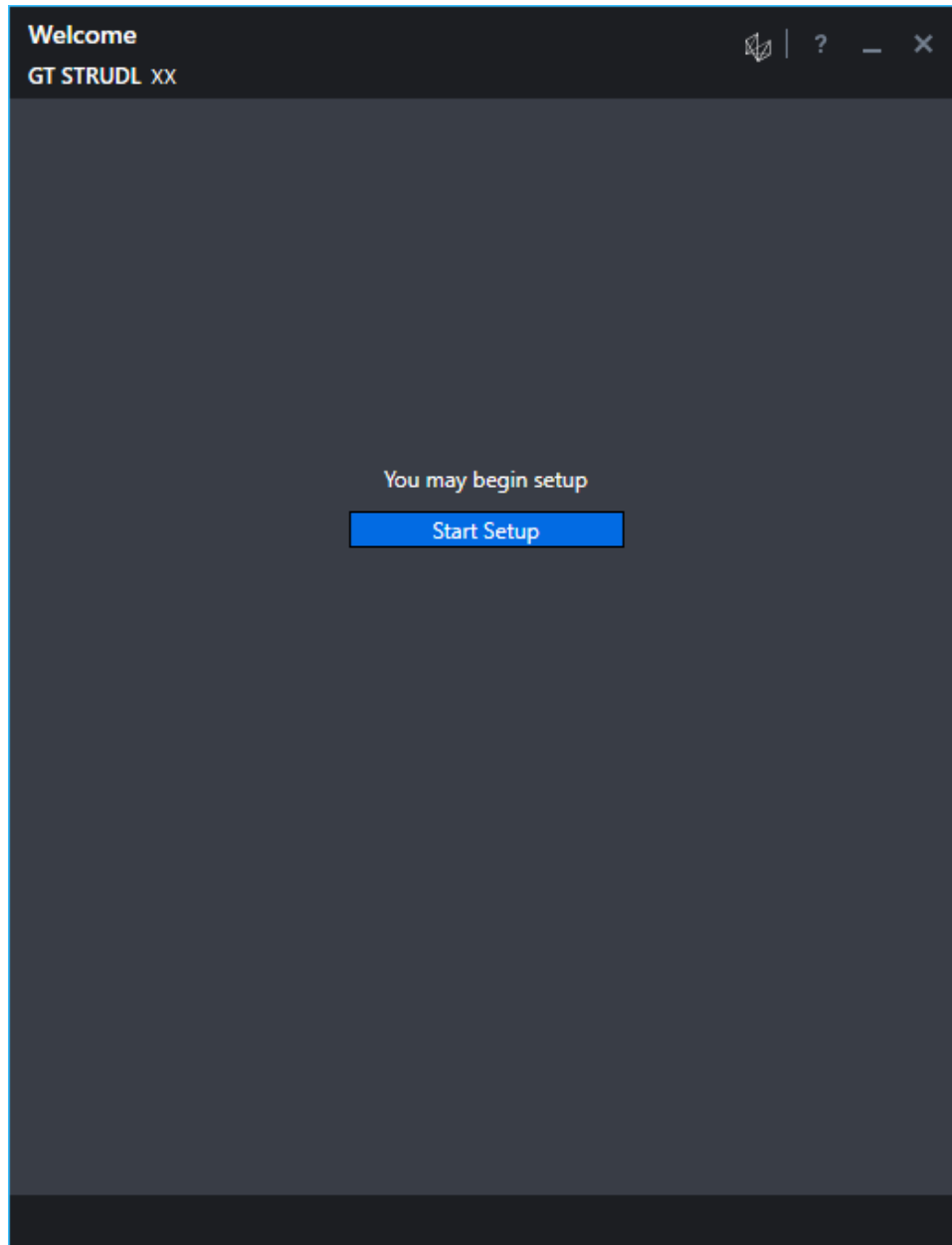
This document contains information about:

- Installing CAD Modeler
- Configuring AutoCAD/BricsCAD and running CAD Modeler for first time
- List of CAD Modeler Functions with detailed explanation for each one
- Tutorial examples
- Summary of CAD Modeler commands

1.2. Installing CAD Modeler

CAD Modeler is installed during GT STRUDL installation. Double-click the GTS_Setup icon to begin installation. Click the “Start Setup” button on the Welcome page.

Note: “XX” in the following screen captures is replaced by the version number being installed in the installation dialogs you will see.



Expand the “CAD Modeler” menu on the form shown below during the GT STRUDL main installation procedure by clicking the triangle to the left of “CAD Modeler”. The versions of BricsCAD® and AutoCAD® that CAD Modeler supports will be displayed.

Details and Features
GT STRUDL xx

User Name *
hexagon user

Company *
Unknown Company

Select Features to Install

☒ All Features

- ☒ GT STRUDL
- ☐ CAD Modeler
- ☐ Seismic Load Data
- ☐ Videos for GTMenu Example Models

Install Path *
C:\Program Files\

Requires 2.56 GB | Available Space 11.33 GB

Cancel | Back | Next

Select one or more AutoCAD or BricsCAD versions to use with CAD Modeler and then press the “Next” button. The supported versions of AutoCAD or BricsCAD usually change with each version of GT STRUDL.

At least one version of AutoCAD (2020-2025) or BricsCAD (Platinum or Pro 19.x, 20.x, 21.x, 22.x, or 23.x) needs to be installed in the computer prior to CAD Modeler installation.

Details and Features
GT STRUDL 43

User Name *
hexagon user

Company *
Unknown Company

Select Features to Install

☒ All Features

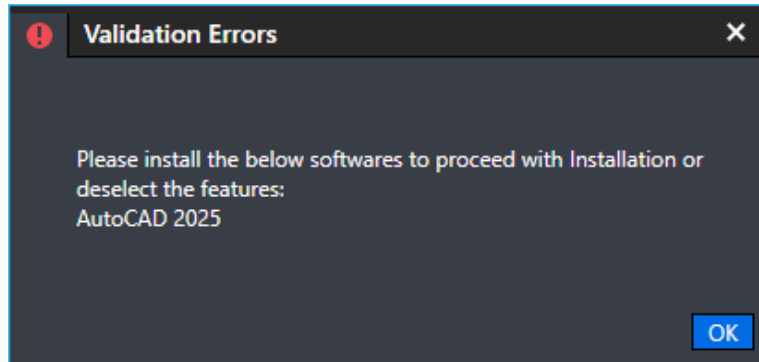
- ☒ GT STRUDL
- ☒ CAD Modeler
 - ☐ AutoCAD 2020
 - ☐ AutoCAD 2021
 - ☐ AutoCAD 2022
 - ☐ AutoCAD 2023
 - ☐ AutoCAD 2024
 - ☒ AutoCAD 2025
 - ☐ BricsCAD 2019
 - ☒ BricsCAD 2020

Install Path *
C:\Program Files\ ...

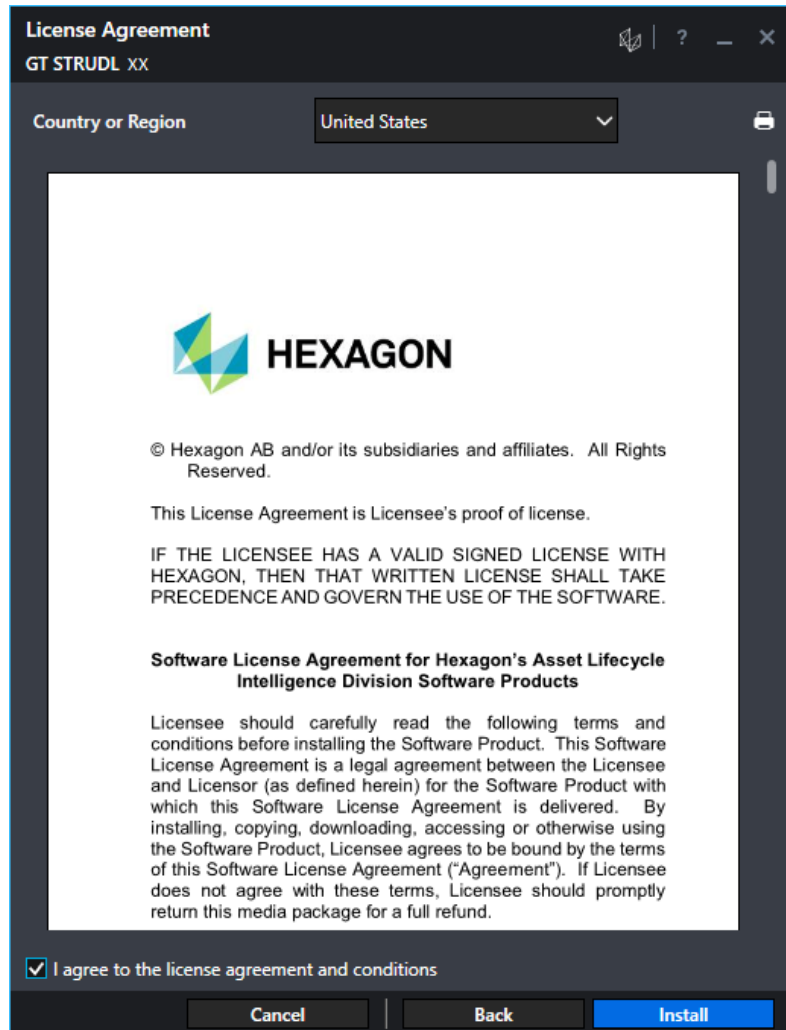
Requires 2.59 GB | Available Space 11.12 GB

Cancel | Back | Next

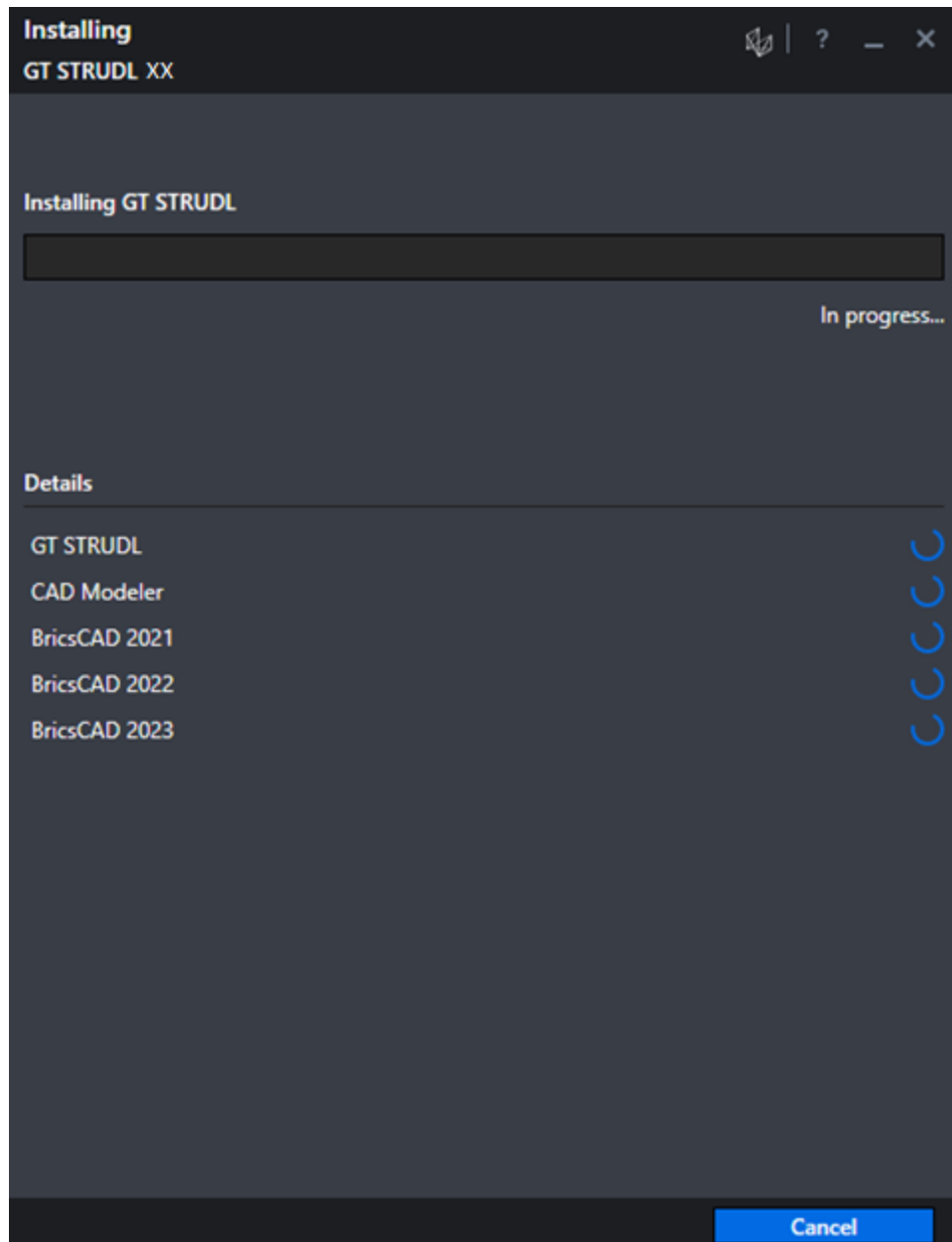
If you select a CAD version that is not installed on your computer, an error message will be displayed:



Otherwise, check the license agreement box and press the button "Install".

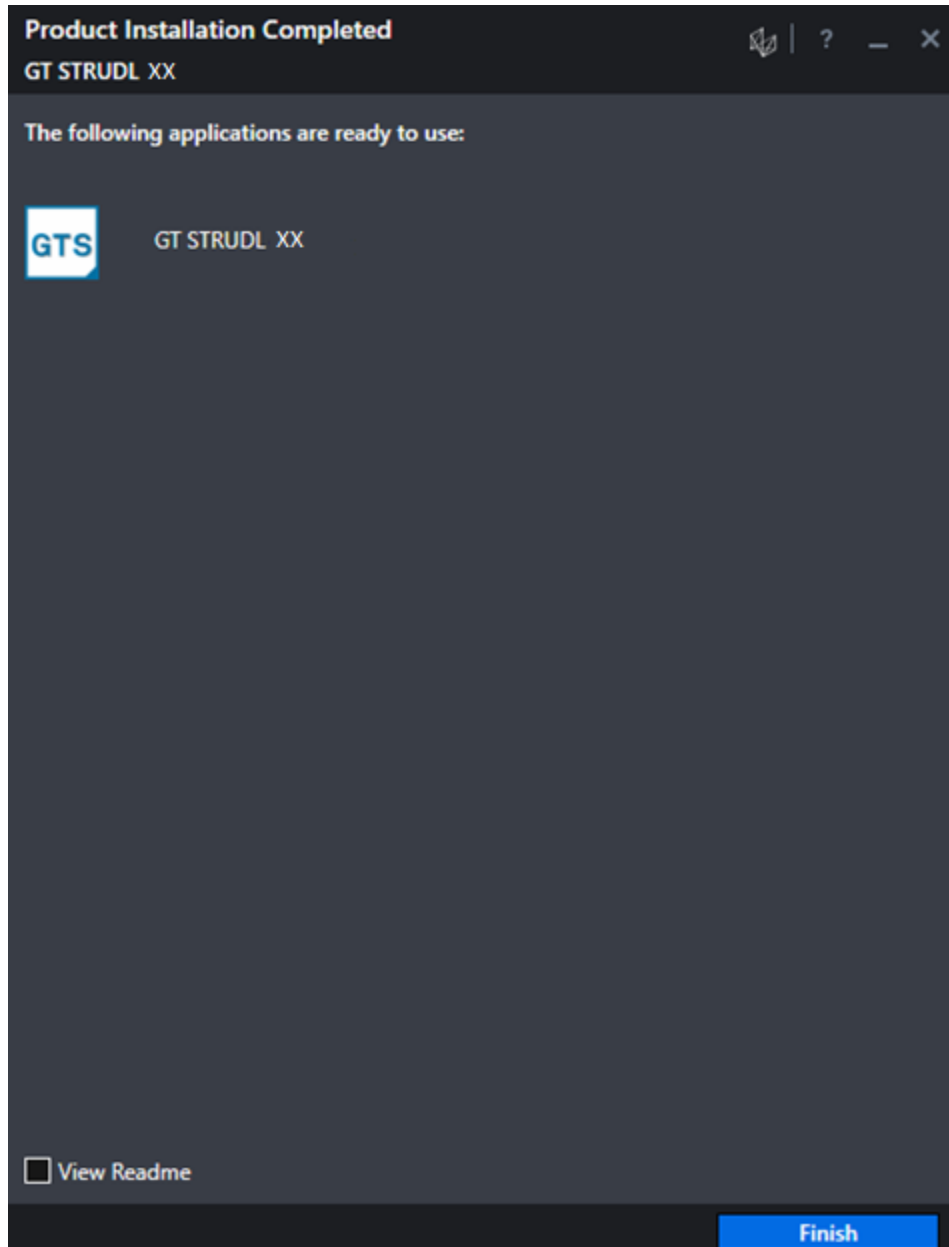


The next screen summarizes your selection.



Press “Finish” to exit the installation process.

CAD Modeler is installed in the same installation directory with GT STRUDL, under the sub-directory “CADModeler”. For example, the default CAD Modeler installation folder is “C:\Program Files\GTSTRUDL\XX\CADModeler”, where “XX” is the version number being installed.



2. Using CAD Modeler

2.1. Overview of Using CAD Modeler and configuring AutoCAD/BricsCAD

CAD Modeler can only be launched directly by GT STRUDL; by initializing an instance of AutoCAD or BricsCAD and automatically loading CAD Modeler ARX/BRX Application (CADModeler) in the same instance. CAD Modeler creates a separate AutoCAD/BricsCAD user profile, named "CADModeler", so the CAD Modeler menus, icons and ribbons do not affect your standard AutoCAD or BricsCAD environment, or other applications running on the top of them.

CAD Modeler commands can be accessed from the menu, from the ribbon area, or by typing the specific command in the command prompt.

- If AutoCAD/BricsCAD Menu is not turned ON, you have to type "MENUBAR" in the command prompt, and then enter 1.
- If AutoCAD/BricsCAD Ribbon is not turned ON, you have to type "RIBBON" in the command prompt.

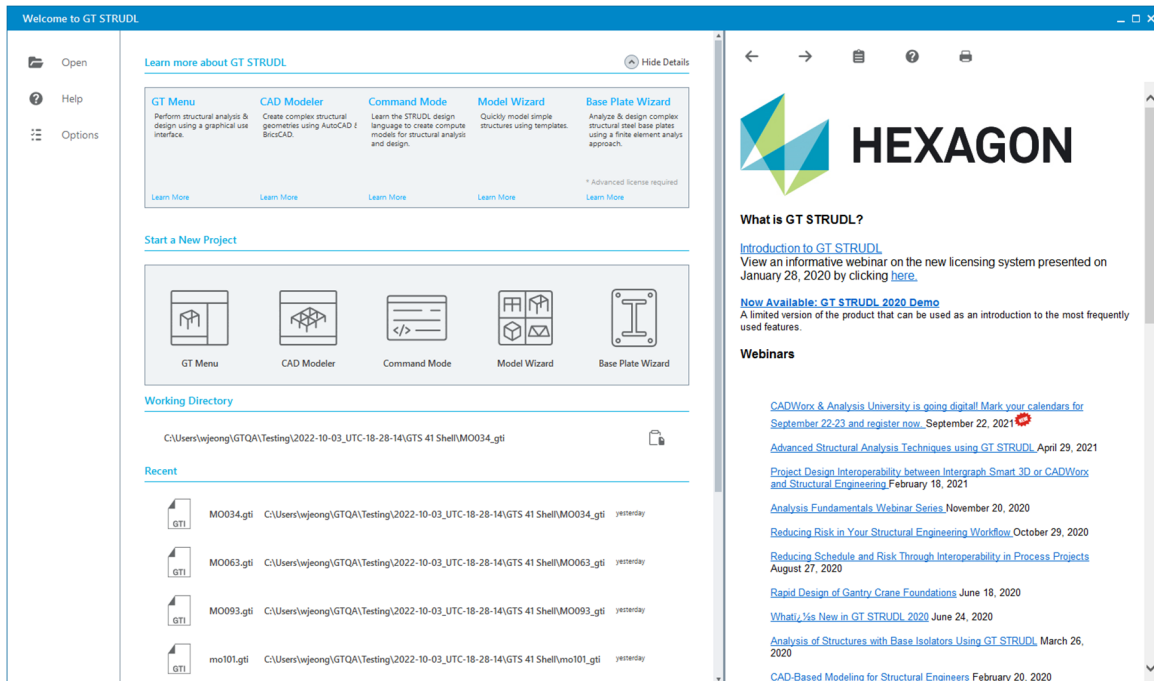
All GT STRUDL structural data, which are created using CAD Modeler, are stored in two files: the .dwg file which contains the AutoCAD/BricsCAD information, and the .db file which contains the structural data. The two files have the same filename (only the extension is different) and they are linked together by CAD Modeler.

CAD Modeler creates a GT STRUDL Input File (.gti) and is able to send this .gti file to the main instance of GT STRUDL, which sits on the background. When analysis is performed in GT STRUDL and results are available for reading, CAD Modeler is able to load them from GT STRUDL DBX (data base exchange) files.

Since CAD Modeler is launched and licensed by the main GT STRUDL application you are not allowed to close the main GT STRUDL window. If GT STRUDL is closed, CAD Modeler outputs a warning and CAD Modeler commands are no longer functional.

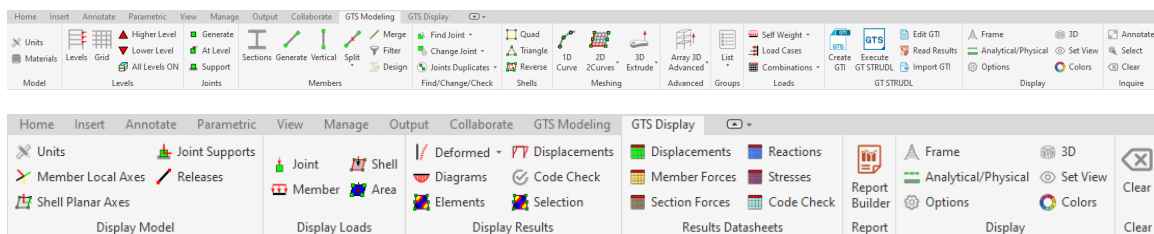
2.2. Running CAD Modeler

CAD Modeler is launched from the GT STRUDL Welcome screen by selecting the "CAD Modeler" icon in the Start a New Project group. A new instance of AutoCAD or BricsCAD, having CAD Modeler automatically loaded, is created. You will be able to identify that CAD Modeler is loaded successfully by having two additional menus ("GTS Modeling" and "GTS Display") next to the AutoCAD or BricsCAD main menus and the "GTS CAD Modeler" and "GTS Display" tabs in the ribbon area.

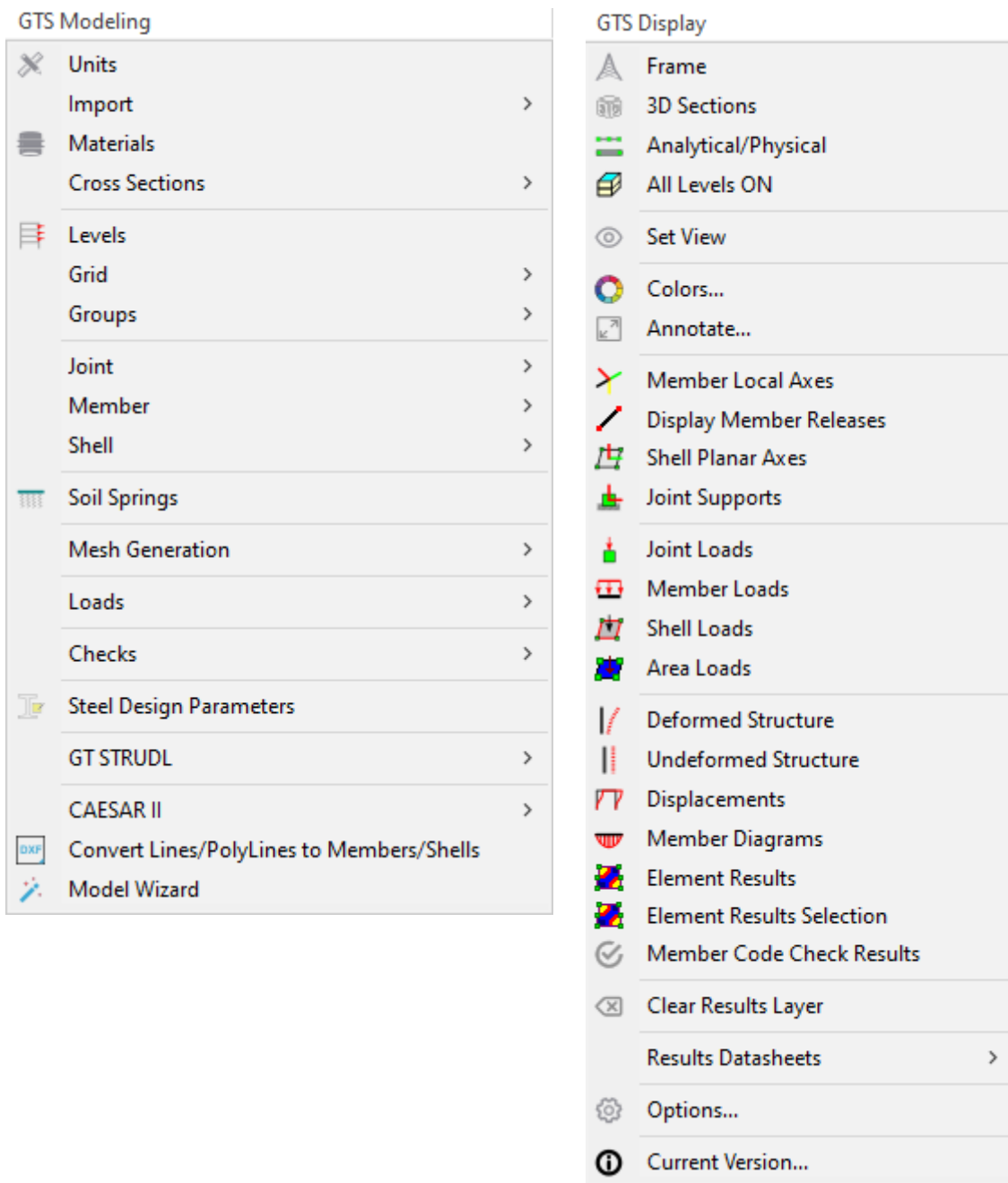


2.3. Menu Bar and Ribbon Area

CAD Modeler commands can be accessed from AutoCAD's or BricsCAD's Ribbon Area at the top of the window by selecting the two tabs at the right: GTS CAD Modeler and GTS Display. CAD Modeler now support BricsCAD version 23 including all new functionality and the dark scheme.



If the AutoCAD or BricsCAD menu is visible, then CAD Modeler commands can be accessed from the Menu Area at the top of the window by selecting GTS Modeling or GTS Display.



2.4. AutoCAD/BricsCAD Commands

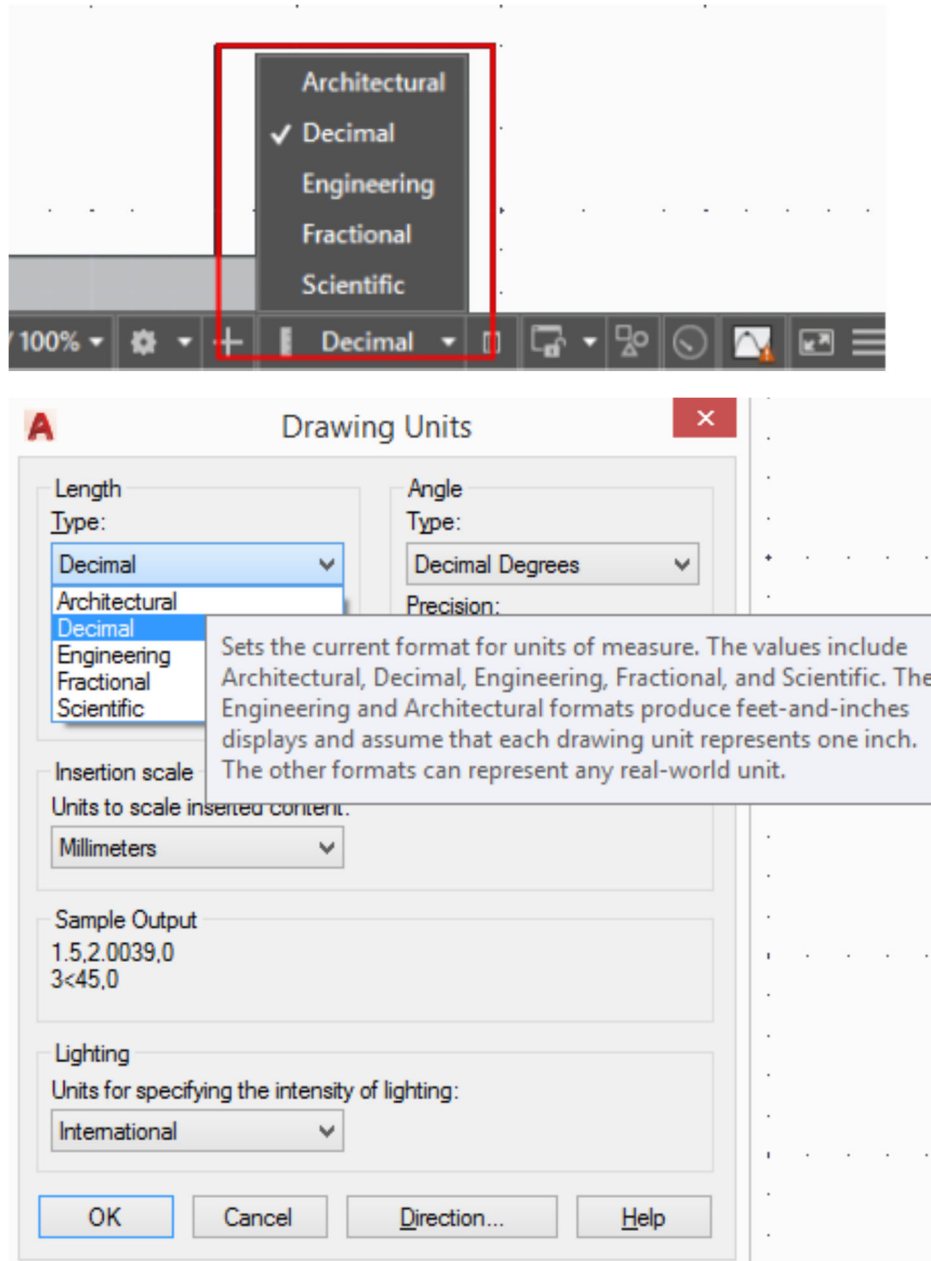
You can use AutoCAD or BricsCAD commands, such as Move, Copy, Rotate, Mirror and Delete to generate your model faster. The assumptions made in the use of each command are:

- Move: By moving a joint, the members and finite elements connected to the joint “follow” this movement
- Copy: Joint, Member and Element Loads and Supports are not copied
- Mirror: Joint, Member and Element Loads and Supports are not copied or mirrored. The Beta Angle of members is not mirrored. Element incidence order is mirrored so that element’s orientation, that defines the Z Planar Axis, remains the same.

- Delete: If a joint is deleted, there is a prompt that asks for confirmation since members and elements connected to this joint will automatically be deleted as well.

2.5. AutoCAD/BricsCAD Drawing Units

It is recommended that you set the AutoCAD/BricsCAD drawing units to **Decimal**. This can be done from the lower right corner of the main application window, as displayed in the next image, or by typing the command UNITS.

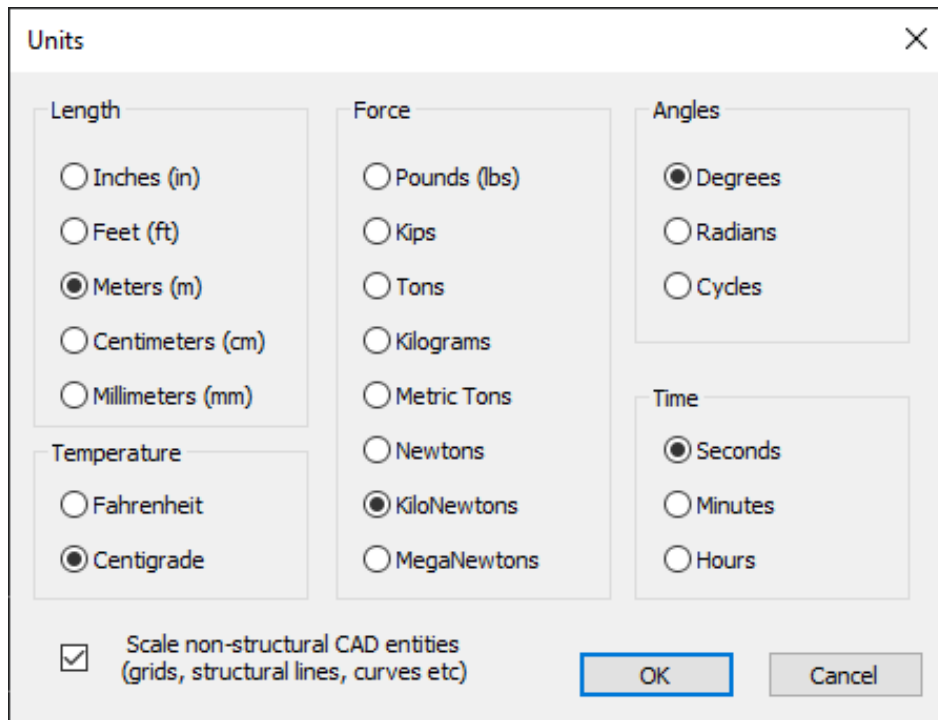


As written in AutoCADs' documentation "*The **Engineering** and **Architectural** formats produce feet-and-inches displays and assume that each drawing unit **represents one inch**. The other formats can represent any real-world unit*". Therefore, if you select **Engineering** or **Architectural** formats you have to set CAD Modeler Length Units to **Inches**. (see [Units](#) Command)

2.6. CAD Modeler Commands

2.6.1. Units

The following form is used to define the active units, either from ribbon command  **Units** or from the menu “*GTS Modeling>Units*” or by typing `GTSUnits` at the command prompt.



The image shows a dialog box titled "Units" with a close button (X) in the top right corner. The dialog is divided into four sections: Length, Force, Angles, and Time. Each section contains a list of units with radio buttons for selection. The "Length" section has options: Inches (in), Feet (ft), Meters (m) (selected), Centimeters (cm), and Millimeters (mm). The "Force" section has options: Pounds (lbs), Kips, Tons, Kilograms, Metric Tons, Newtons, KiloNewtons (selected), and MegaNewtons. The "Angles" section has options: Degrees (selected), Radians, and Cycles. The "Time" section has options: Seconds (selected), Minutes, and Hours. At the bottom left, there is a checked checkbox labeled "Scale non-structural CAD entities (grids, structural lines, curves etc)". At the bottom right, there are "OK" and "Cancel" buttons.

You can change the Units any time during working in CAD Modeler. Moreover, you can choose if the non-structural AutoCAD/BricsCAD entities, such as grids, structural lines, curves, polylines, etc will be scaled together with the structure whenever you change the length units.

As noted in [2.5](#), if the Drawing Units are in Engineering or Architectural format then you have to set the CAD Modelers' Length Units to Inches (in) using this Dialog.

The current units appear at the top of the main CAD Modeler window:

GTS CAD Modeler | M KN DEG FAH SEC | Drawing1.dwg

Note: Some Edit Boxes appear in yellow background and green fonts, like the one at the picture below. You can use mixed units in the yellow edit boxes. For more information about Mixed Units and the valid syntax, please read GT STRUDL GT Menu Guide.

Joint Coordinates and Offsets

Coordinates

X : 2487.85

Y : 1644.03

Z : 0

2.6.2. Materials

The following form is used to modify existing material properties or create new materials. You can select this command either from the ribbon command  **Materials** or from the menu “GTS Modeling>Materials” or by typing `GTSMaterials` at the command prompt.

Material

Material Properties

ID	Name	E	G	Density	Poisson	CTE
1	Steel	1.9994800E+08	7.5842000E+07	7.6901000E+01	3.0000000E-01	1.1700000E-05
2	Concrete	2.4821100E+07	9.9284600E+06	2.3561600E+01	1.7000000E-01	9.9000000E-06
3	Aluminum	6.8947600E+07	2.5855400E+07	2.6601800E+01	3.3000000E-01	2.3400000E-05

Add Material Delete Material OK Cancel

2.6.3. Levels

It is optional to define Levels (stories) in your structure. However, it is recommended that you do so when modeling industrial or other building-like structures as this will save you time during the creation of your model. You can access the level properties dialog from the ribbon icon



Levels or from the menu “GTS Modeling>Levels” or by typing `GTSLevels` at the command prompt.

Using the *Level Properties* form you can:

- Set the *Height* for each level, in current length units
- Define the visible status of each level: if *Visible* or not
- *Add Levels* to the model

- *Delete Levels* from the model
- *Detect Levels Automatically* using an algorithm to detect levels along the height of the structure by identifying locations having at least four horizontal members.
- *Merge Levels*, by selecting two or more levels and merging them to one.
- Define if the *Vertical Axis* is the global Z or global Y.
- *Update Levels for All Entities*, in order to assign the correct Level to each entity (joint, member or finite element) depending on its coordinates along the height of the structure.

The dialog box is titled "Level Properties" and contains a table of levels and a set of options.

Levels	Height	Elevation	Visible
1	4	4.000000	☒
2	3.000000	7.000000	☒
3	3.000000	10.000000	☒

Options:

- Add Level
- Delete Level
- Detect Levels Automatically
- Merge Levels
- Base Elevation: 0.0000
- ☒ Z Vertical Axis (else Y)
- ☐ Update Levels for All Entities

Buttons: OK, Cancel

After defining Levels, you can switch between levels by either using the "Visible" check boxes from the Level Properties form, or using the ▲ Higher Level and ▼ Lower Level icons in the ribbon area. You can also type `GTSLevelUp` and `GTSLevelDown` at the command prompt.

Moreover, you can define a grid system and/or generate vertical members (Columns) with a single click. These commands will be explained below.

Finally, the current level, if defined, appears at the top of CAD Modeler window, next to the Current Units.

GTS CAD Modeler | M KN DEG CEN SEC | Level: 2

2.6.4. Grid

A Grid system can be defined and used as a pattern for entering beams and columns. In order to be able to enter a Grid, you must first specify Levels in your structure (see the Levels command above). You can access the Grid dialog by expanding the “Levels” tab from the ribbon icon

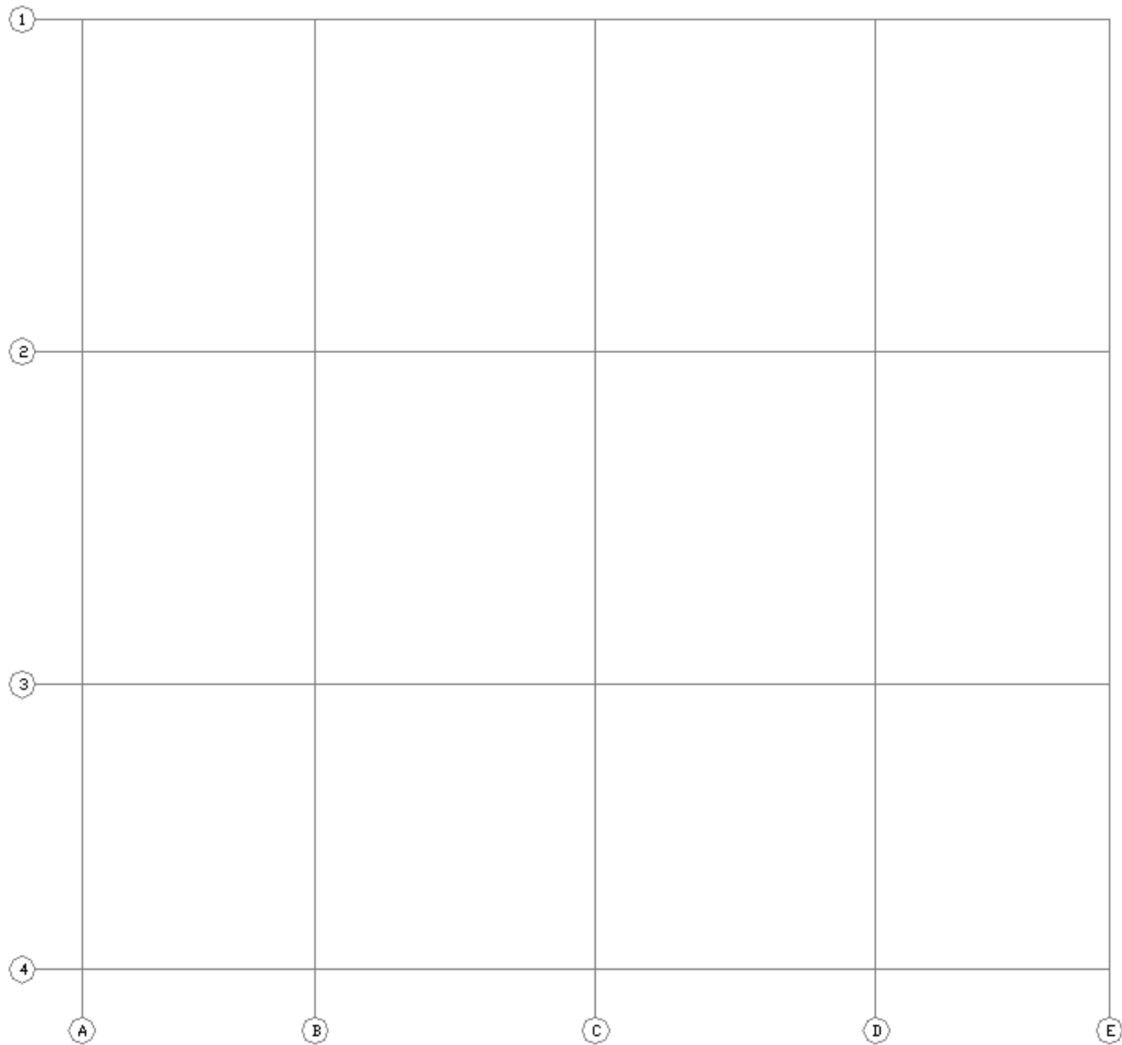


Grid, or from the menu “GTS Modeling>Grid>Create” or by typing `GTSGrid` at the command prompt.

Using the *Grid* form you can:

- Set different parameters for the *Horizontal* and *Sidelong* directions of the grid.
- Define and control the spacing in each direction, by entering the desired spacing – *Distance* of the new grid line and pressing *Add* button. Later on you can edit a specified spacing or delete it, using the corresponding buttons *Edit* and *Delete*.
- Define the *Angle* between the *Grid X-Axis* and the global X-axis
- The *Angle* between the *Horizontal* and *Sidelong* lines (default equal to 90 degrees)
- Control the *Height* of fonts
- Control the *Position* of the labels
- Control the *Type* of identification to be either Number or Letters
- Control the *Starting From* item, which can be a number or letter depending on the Type.
- Select the levels that this grid will be applied to. You can apply the grid to more than one levels and/or have multiple grids per level.

By pressing OK, you are prompted to enter the *Insert Point* of the grid, meaning the coordinates of the lower left corner of the grid. The grid lines are then created as shown in the figure on the next page.



You can also change the properties of an existing grid from the menu “*GTS Modeling>Grid>Change*” or by typing `GTSGridChange` at the command prompt, and then selecting the Grid to be edited.

2.6.5. Creating Joints

You can generate individual joints from the ribbon command  **Generate** or from the menu “*GTS Modeling>Joint>Generate Joint*” or by typing `GTSJoint` at the command prompt. You then must enter the X,Y,Z coordinates (separated by comma) or click at the corresponding point at the screen. However, for frame structures, it is recommended to start generating members (and joints will be automatically generated at their ends).

If you have already defined Levels at the structure, you can generate individual joints at the current level from the ribbon command  **At Level** or from the menu “*GTS Modeling>Joint>Generate Joint at Level*” or by typing `GTSJointLevel` at the command prompt. You then have to enter only X and Y coordinate (Z will be calculated using the current Level’s Elevation).

2.6.6. Finding Joints

You can find an individual joint from the ribbon command  **Find Joint** or from the menu “*GTS Modeling>Joint>Find*” or by typing `GTSFJID` at the command prompt and enter the name of the Joint. If the joint name exists, the joint will be selected (by clicking on “change”, you can modify it without making a new selection).

2.6.7. Joint Supports

You can find an individual joint from the ribbon command  **Support** or from the menu “*GTS Modeling>Joint>Support*” or by typing `GTSJointSupport` at the command prompt and select the joint or the joints to be supported. The Joint Properties form then appears, where you can define which degrees of freedom are fixed and also enter a *spring* value in case of elastic supports. Using the *Quick Selection*, you can quickly define a Fixed, Pinned or Free Joint (by default all joints are free)

Restraints & Spring values

Quick Selection : Pin

Restraint	Spring	Restraint	Spring
☒ Fx	0	☐ Mx	0
☒ Fy	0	☐ My	0
☒ Fz	0	☐ Mz	0

2.6.8. Joint Properties

You can change the properties of a joint from the ribbon command  **Change Joint** or from the menu “*GTS Modeling>Joint>Change*” or by typing `GTSJointChange` at the command prompt and select the joint or the joints to be edited or by double-clicking on an existing joint.

Joint Properties

Model | Joint Generalized Loads

General
 Name :
 Level :

Rotation
 theta X :
 theta Y :
 theta Z :

Groups
☐ Inactive
 Groups that joint belongs

Joint Coordinates and Offsets
 Coordinates
 X :
 Y :
 Z :

Restraints & Spring values
 Quick Selection :

Restraint	Spring	Restraint	Spring
☐ Fx	0	☐ Mx	0
☐ Fy	0	☐ My	0
☐ Fz	0	☐ Mz	0

The “*Joint Properties*” form appears, and at the “*Model*” tab you can enter the *Name* of the Joint (up to 8 characters) the *Level* that the joint belongs (optional), the *theta rotation* angles for rotated support joints, the *Groups* that the joints belongs to, the *coordinates* of the joint in the current unit system, the *restraints* of the joint and the *spring* values.

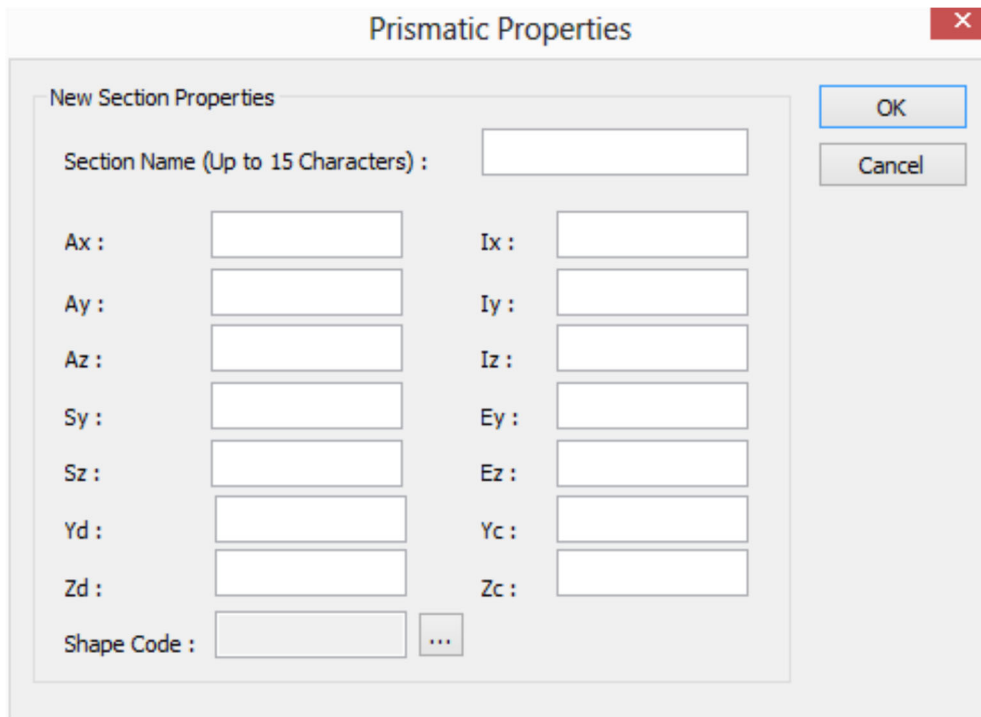
If you select more than one joint, then “*Multiple Selection*” appears at the top of the Joint Properties form, and all data entered in the form will be applied to all selected joints.

2.6.9. Sections

For models which contain frame members, you should select the cross sections to be added to your project either from the default cross section library or by creating user defined prismatic cross-sections or by importing user defined cross sections that have been created in GT STRUDL.

Prismatic cross sections can be created from the Menu “*GTS Modeling >Cross Sections>Prismatic*” or by typing `GTSPismatic` at the command prompt.

In the dialog shown below, you enter the cross-section properties in the current unit system.



The image shows a software dialog box titled "Prismatic Properties" with a close button (X) in the top right corner. Inside the dialog, there is a section titled "New Section Properties". Below this title, there is a text input field for "Section Name (Up to 15 Characters) :". To the right of the input fields are two buttons: "OK" and "Cancel". Below the section name field, there are two columns of input fields. The left column contains: Ax :, Ay :, Az :, Sy :, Sz :, Yd :, Zd :, and Shape Code :. The right column contains: Ix :, Iy :, Iz :, Ey :, Ez :, Yc :, and Zc :. Each of these labels is followed by an empty rectangular input box. At the bottom right of the "Shape Code" field, there is a small button with three dots "...".

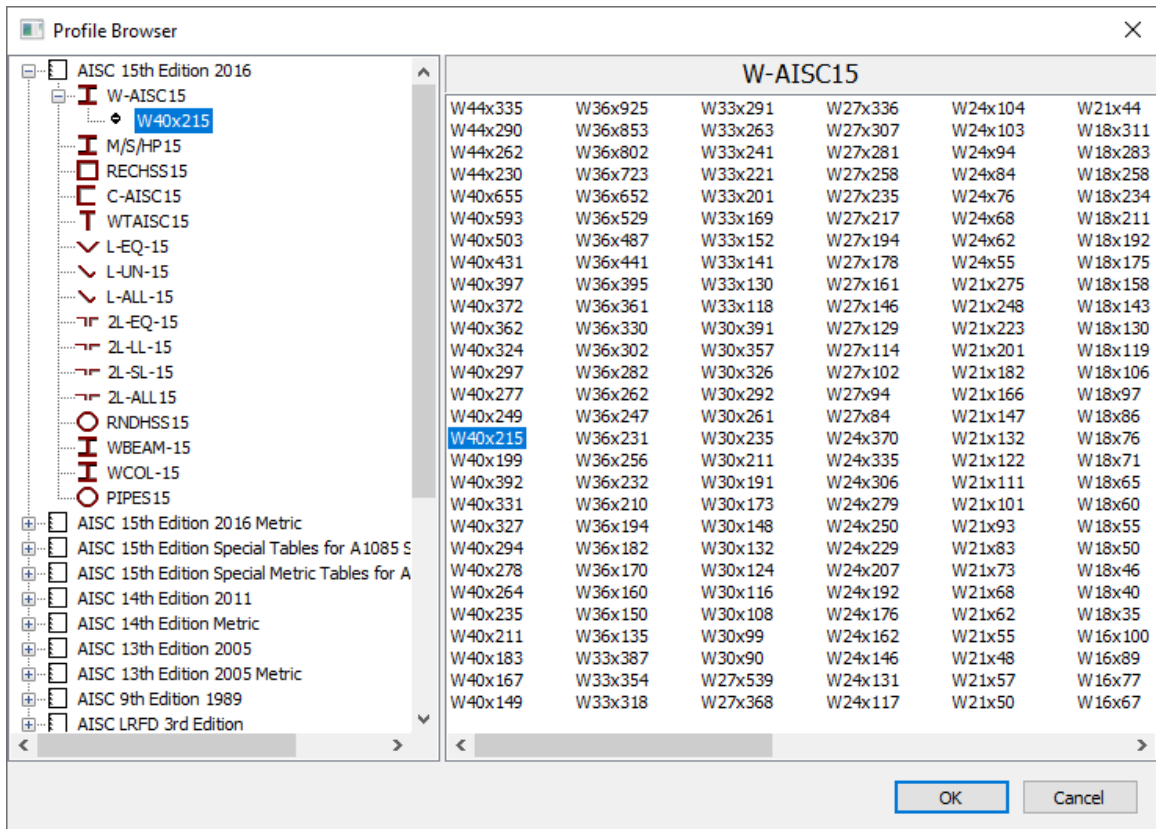
You can access the existing cross section library (GT STRUDL tables) from the ribbon command



Sections, or from the Menu "*GTS Modeling>Cross Sections>Table*" or by typing `GTSPParams` at the command prompt.

Using the following form, where all GT STRUDL built-in TABLES are available, you select the cross sections for your project by double clicking on them, at the right part of the screen. The list of selected cross-section profiles appears at the left part of the screen and the selected profiles have a large black dot in front of them. Check the Release Guide for your version to see the newest built-in tables.

User defined cross sections, that have been created in GT STRUDL, can be imported in CAD Modeler from the Menu "*GTS Modeling >Cross Sections>Import from User Dataset*" or by typing `GTSPOpenDS` at the command prompt. Using the "Select User Dataset DS File" dialog you can select the dataset file (*.ds) that includes the user defined cross sections to be imported. By pressing "Open", the cross sections of the selected .ds file are added to the available built-in TABLES, presented in the previous paragraph.

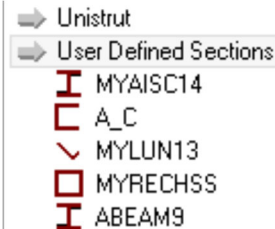


Moreover, you get a notification at the command prompt, regarding the number of tables that exist in the dataset. For example: "Number of tables in ds = 5".



Sections

Next time you select the command you will be able to see the user defined sections at the bottom of the list.



Note: Whenever you import a TABLE of sections that has been previously imported in CAD Modeler, all new data will overwrite previous TABLE section data.

Note: Whenever you use a User Defined Section in CAD Modeler, you must manually open the corresponding .ds file in GT STRUDL prior to the "Execute GTSTRUDL" Command.

2.6.10. Creating Members



You can generate individual members from the ribbon command **Generate** or from the menu “*GTS Modeling>Members>Generate Beam Members*” or by typing `GTSCoordinate` at the command prompt. You must then enter the X,Y,Z coordinates (separated by commas or click at the corresponding point at the screen) of the member start and then of the member end. Joints are automatically generated at both member ends, unless a joint already exists at the specific point. If so, the member is connected to the existing joint(s).

If you have already defined Levels at the structure, you can generate vertical members



(columns) at the current level from the ribbon command **Vertical** or from the menu “*GTS Modeling>Member>Generate Vertical Member*” or by typing `GTSCoordinate` at the command prompt. You then have to enter only one point (starting top point) in the floor plan. The ending bottom point will be automatically calculated, having the same X and Y coordinates, and Z coordinate will be calculated by the current level’s height.

After giving the command the “*Place Member*” form appears, where you define the properties of the member.

Note: Each time you create a member, the orientation of the cross section will appear in the middle of the element, unless you clear it with command “Clear” (see 2.6.86).

Cross Section

Table Section:
 <Select Table Section> ▼ ...

or Member Dimensions:
 Rectangle Concrete ▼

or Same as Member:
 <Select Existing Member> ▼

Material
 Steel ▼ ...

Releases
 00 Pin-Pin ▼

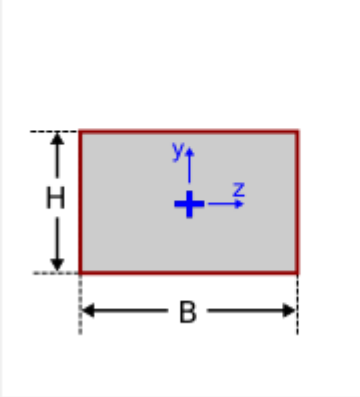
Beta (o)
 0 ▼

☒ Split Intersecting Members ☐ Physical Member

☒ Split Ending Members

Place Member(s) >>

Section Properties



B :
 H :

You can select:

- A Table Section from the list of available sections in the project, **or** give the dimensions of the typical shapes available **or** match the section properties of one existing member to save time typing the values.
- The Material from the list of available materials in the project
- Common Member Releases configuration
- Beta angle (in degrees)
- To split intersecting members, along new member's length, including the new member
- To split ending members, if the member starting and/or ending joints are placed along existing members.
- To place a physical member together with the analytical members. If the new physical member is crossing other physical members, the new physical member will be split into more parts.

Then, you must click on the "Place Member(s) >>" button and start placing members.

You can change the properties while the command is active, and the next member(s) will be placed using the new values.

When you are done, press ESC to exit from the command. The form is hidden automatically.

2.6.11. Finding Members

You can find an individual member from the ribbon command  **Find Member** or from the menu "GTS Modeling>Member>Find" or by typing `GTSFMID` at the command prompt and enter the name of the Member. If member name exists, the member will be selected (by clicking on "change" you can modify it without making a new selection).

2.6.12. Splitting Members



You can split a member into two or more parts from the ribbon command **Split** or from the menu “*GTS Modeling>Member>Split Member*” or by typing `GTSSplitMember` at the command prompt and select the Member to be split. You then define “*Distance for splitting the member or the number of equal parts (negative number)*”, entering:

- the position of the split, meaning the length of the 1st part starting from starting joints, or
- the number of equal parts that will be generated after the split, by typing a negative number. For instance, entering -3 means to split the original member into 3 equal parts.

2.6.13. Splitting to Crossing Members

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



Split to Members

or from the menu “*GTS Modeling>Member>Split to Crossing Members*” or by typing `GTSSplitToMembers` at the command prompt and select the Member to be split.

2.6.14. Merging Members



You can merge two members to one member from the ribbon command **Merge** or from the menu “*GTS Modeling>Member>Merge Members*” or by typing `GTSMergeMembers` at the command prompt and select two members. The two members must have a common joint (middle). After merging the middle joint is NOT deleted and you have to delete it manually. This joint can be removed manually, using AutoCAD’s/BricsCAD’s erase command, or by using CAD Modeler’s “Remove Floating Joints” Command.

2.6.15. Member Properties



You can change the properties of a member from the ribbon command **Change Member** or from the menu “*GTS Modeling>Member>Change*” or by typing `GTSBeamChange` at the command prompt and select the member or the members to be edited or by double-clicking on an existing member.

The “*Member Properties*” form appears, and at the “*Model*” tab you can enter the *Name* of the Member (up to 8 characters), the *Level* that the member belongs to (optional), the *Type* of the Member (Space Frame or Space Truss), *Starting* and *Ending* Joints, *Beta Angle*, the *Groups* that the member belongs to, the *Cross-Section* applied to this member and the corresponding section properties, the *Material* of the member, member *releases* and *elastic end connection* spring values, *End Sizes* and global *Member Eccentricities*.

Member Properties

Model Section Properties Member Loads Member Temperature Loads Member Distortions

General

Name : 1

Level : 1

Type - Incidences

Type : SPACE FRAME

Start : 1

End : 2

Beta Angle : 0

Physical Member

Groups

☐ Inactive ... Remove

Groups that Member belongs

Section Properties

Section :

Ax : Ix :

Ay : Iy :

Az : Iz :

Sy : Ey :

Sz : Ez :

Yd : Yc :

Zd : Zc :

Shape Code :

Material Properties

Material : Steel

E : 1.99948e+08 Density : 76.901

G : 7.5842e+07 CTE : 1.17e-05

Poisson : 0.3

Releases & Elastic Connection spring values

Quick Selection :

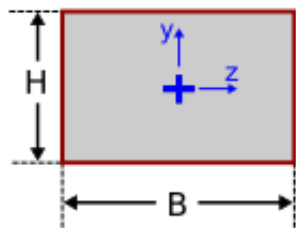
Start	Spring	End	Spring
☐ Fx	0	☐ Fx	0
☐ Fy	0	☐ Fy	0
☐ Fz	0	☐ Fz	0
☐ Mx	0	☐ Mx	0
☐ My	0	☐ My	0
☐ Mz	0	☐ Mz	0

End Sizes OR Member Eccentricities (Offsets)

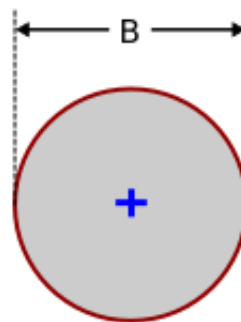
Start	End
Sizes : 0	0
X : 0	X : 0
Y : 0	Y : 0
Z : 0	Z : 0

OK Cancel Apply Help

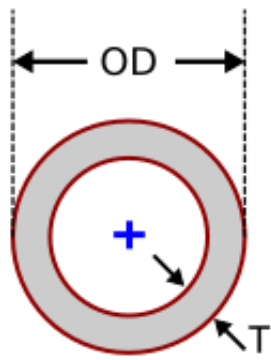
Using the “Section Properties” tab, you can define typical concrete shapes or other shapes that appear in the next image. Shapes having the identifier “Concrete” should be used for thick sections.



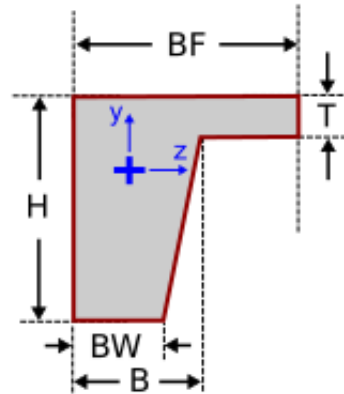
Rectangle (Concrete)



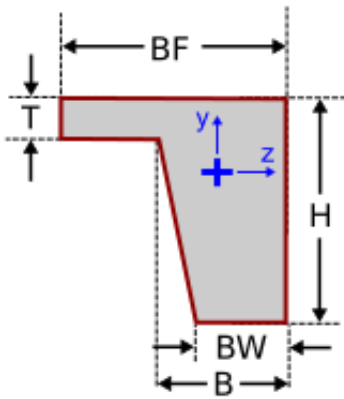
Circle (Concrete)



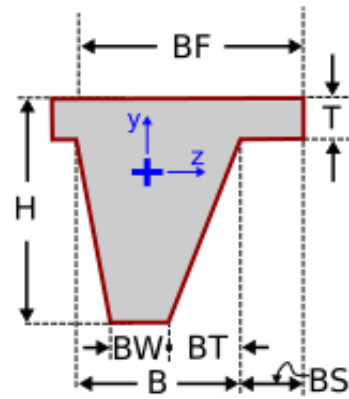
Pipe



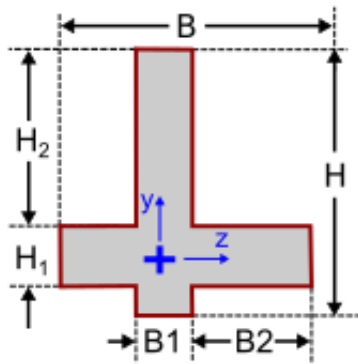
Right L (Concrete)



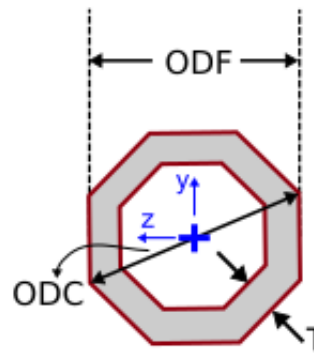
Left L (Concrete)



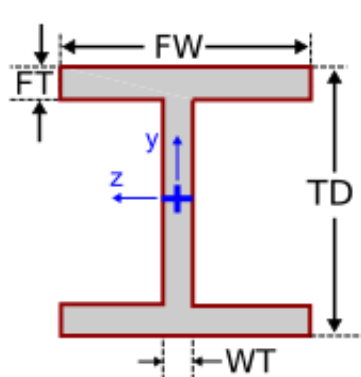
T Shape (Concrete)



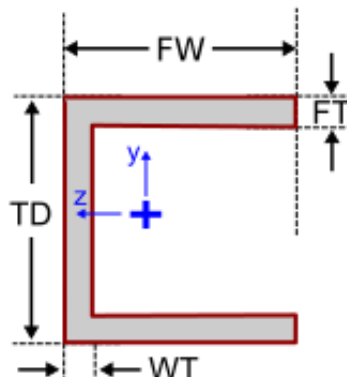
Cross (Concrete)



Polygon



I Shape



Channel

If you select more than one member, then “Multiple Selection” appears at the top of the Member Properties form, and all data entered in the form will be applied to all selected members.

Physical Member Properties

Model

Section Properties

Member Loads

General

Name : PM000003

Level : 1

Type - Incidences

Type : SPACE FRAME

Start : 6

End : 18

Beta Angle : 90

Physical Member : PM000003

Groups

Inactive ... Remove

Analytical Members

14

29

15

21

Section Properties

Section : IPE270 IPE European

Ax : 0.00459

Ay : 0.00164736

Az : 0.001836

Sy : 6.2222e-05

Sz : 0.00042889

Yd : 0.27

Zd : 0.135

Ix : 1.6e-07

Iy : 4.2e-06

Iz : 5.79e-05

Ey : 0

Ez : 0

Yc : 0.135

Zc : 0.0675

Shape Code : 1.0

Material Properties

Material : Steel

E : 1.99948e+08

G : 7.5842e+07

Poisson : 0.3

Density : 76.901

CTE : 6.49865e-06

Releases & Elastic Connection spring values

Quick Selection :

Start	Spring	End	Spring
☐ Fx	0	☐ Fx	0
☐ Fy	0	☐ Fy	0
☐ Fz	0	☐ Fz	0
☐ Mx	0	☐ Mx	0
☐ My	0	☐ My	0
☐ Mz	0	☐ Mz	0

End Sizes OR Member Eccentricities (Offsets)

Start	End
Sizes : 0	0
X : 0	X : 0
Y : 0	Y : 0
Z : 0	Z : 0

OK Cancel Apply Help

If you select a physical member, then “Physical Member Properties” appears at the top of the Member Properties form, and you can modify the properties of the Physical Member. You can only modify the Name, Beta Angle, Section Properties, Releases and Eccentricities. Any modification applies to all analytical members, except Releases and Eccentricities that apply only to the start of the first analytical member and to the end of the last analytical member.

Moreover, you can see the list of labels of the Analytical members that are part of this Physical Member.

If you apply any loads to the physical member, the load parameters are copied to each analytical member belonging to it.

2.6.16. Member Filters

You can select members of the structure, that fulfill several criteria, using the icon  **Filter** or from the menu "*GTS Modeling>Member>Filter*" or by typing `GTSFilterMembers` at the command prompt.

Members can be filtered forming queries of three different categories:

- **Their Properties**, that can be: Name, Level, Section, Material, Beta Angle, Group, Release Statues, Kf values, Eccentricities and End Sizes
- **Their Loading Data**, that can be: Load Case, Load Type, Load Direction, Load Values and Location.
- **Their Analysis Results**, that can be: Load Case, Member Forces $F_x - F_y - F_z - M_x - M_y - M_z$ for both ends and section forces $F_x - F_y - F_z - M_x - M_y - M_z$.

You can set multiple (up to 5) conditions of the same category using logical expressions (AND, OR). For example, filter members that their section is IPE330 AND they belong to level < 3 AND their beta angle is greater than or equal to 90.

After the query is formed, your press "Execute >>" and the IDs of the members fulfilling the criteria appear in the "Results" list.

Filtered members may be:

- Added to any Group
- Selected as AutoCAD's/BricsCAD's selection (to be edited, moved, copied, moved etc), using the option "*Keep Selected after closing form*"
- Made the only visible entities of the structure, by hiding all other entities, using the option "*Make them the only visible*"

X

Filter Members

OK

Cancel

SELECT Members

Section

AND Level

AND Beta Angle

=

<

>=

=

=

IPE330 IPE European

3

90

Member Loads (Units: kN, m)

Member Results / Section Forces

SQL Query (WHERE)

(nSection = '2')
 AND (nLevel < '3')
 AND (betaAngle >= '90')

Execute >>

Clear

Action

Items Selected: 70

ADD To Group >>

...

☐ Keep Selected after closing form

☐ Make them the only Members Visible

Result

9

10

11

12

13

14

15

16

17

18

19

20

21

22

23

24

25

26

33

24

2.6.17. Creating Shell Finite Elements

Shell finite elements are generated automatically using the meshing functions described below. However, you can generate individual quadrilateral or triangle shell elements one by one.

Quadrilateral elements can be created using the icon  **Quad** or from the menu “GTS Modeling>Shell>Generate quad at joints” or by typing `GTSShell` at the command prompt. You must then enter the X,Y,Z coordinates (separated by commas or click at the corresponding point at the screen) of the four corners of the quad element. Joints are automatically generated unless a joint already exists at the specific point. If so, the element is connected to the existing joint(s).

Triangular elements can be created using the icon  **Triangle** or from the menu “GTS Modeling>Shell>Generate triangle at joints” or by typing `GTSShellT` at the command prompt. You must then enter the X,Y,Z coordinates (separated by commas or click at the corresponding point at the screen) of the three corners of the triangular element. Joints are

automatically generated unless a joint already exists at the specific point. If so, the element is connected to the existing joint(s).

2.6.18. Reverse Incidence Order

The Incidence Order (clockwise or counterclockwise) of selected shell elements can be reversed using the icon  **Reverse** or from the menu “*GTS Modeling>Shell>Reverse Incidence Order*” or by typing `GTSShellReverse` at the command prompt. The Incidence Order defines the orientation of the Element’s Planar Z and Local Z Axes which then also affects the Local and Planar X and Y Axes.

2.6.19. Finding Shells

You can find an individual shell element from the icon  **Find Shell** or the menu “*GTS Modeling>Shell>Find*” or by typing `GTSFEID` at the command prompt and enter the name of the element. If the element name exists, the shell element will be selected (by clicking on “change” you can modify it without making a new selection).

2.6.20. Shell Properties

You can change the properties of a shell finite element from the icon  **Change Shell** or the menu “*GTS Modeling>Shell>Change*” or by typing `GTSShellChange` at the command prompt and select the shell or the shells to be edited or by double-clicking on an existing shell element.

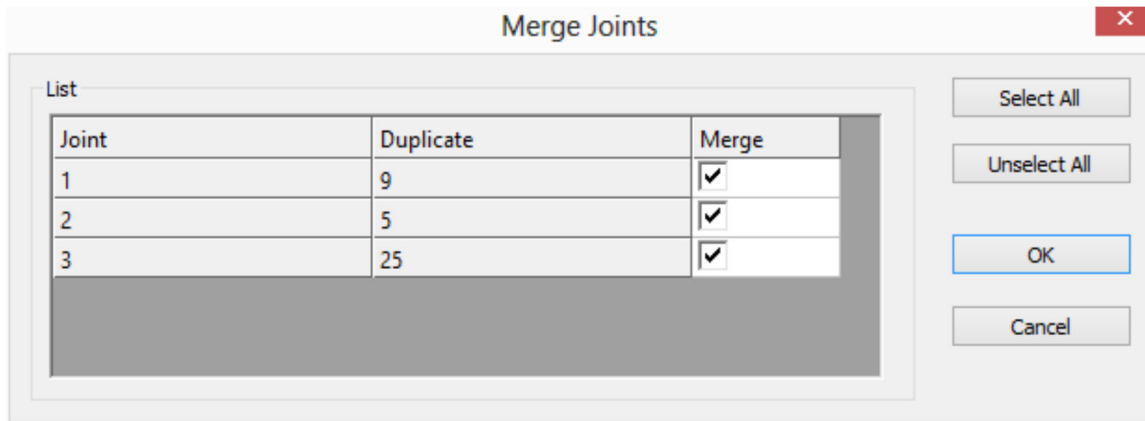
The “*Element Properties*” form appears as shown below, and from the “*Model*” tab, you can enter the *Name* of the Element (up to 8 characters), the *Level* that the element belongs to (optional), the *Type* of the Element, *Joint Incidences*, the *Thickness* of the shell, the *Groups* that the element belongs to and the *Material* of the element.

If you select more than one element, then “*Multiple Selection*” appears at the top of the Element Properties form, and all data entered in the form will be applied to all selected elements.

2.6.21. Joints Duplicates

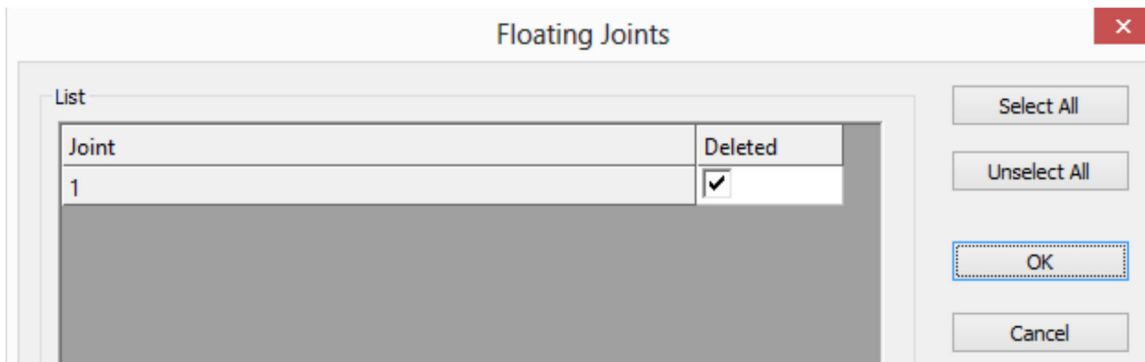
In order to erase joints that have the same coordinates (one on the top of the other) that may have been generated by mesh generation functions, you have to check the model for duplicate

joints from the ribbon command  **Joints Duplicates** in the “Find/Change/Check” panel or select “*GTS Modeling>Checks>Joint Duplicates*” from the menu or type `GTSCheckDuplicateJoints` at the command prompt. You then must enter the desired merge accuracy (*Enter Merge Tolerance <0.001000>*). If duplicate joints exist in the structure, a new dialog appears having the full set of duplicate pairs, where you can select the joints to be merged or not as shown on the next page:



2.6.22. Joints Floatings

Floating Joints are the joints that are not connected to any member or finite element, therefore they may cause instability in the solution of the mathematical model. Use the ribbon command  **Joints Floatings** in the “Find/Change/Check” panel or select “*GTS Modeling>Checks>Joint Floatings*” from the Menu or type `GTSCheckFloatingJoints` at the command prompt, floating joints are automatically identified, and using the corresponding dialog as shown below, they can be deleted.



2.6.23. Joints Interference

In order to check in order to search for joints close to a member, not connected to it, you can use the ribbon command  **Joints Interference** in the “Find/Change/Check” panel or select “*GTS Modeling>Checks>Joints Interference*” from the Menu or type `GTSCheckInteferenceJoints` at the command prompt. You then must enter the desired tolerance (Enter Check Tolerance <0.050800>).

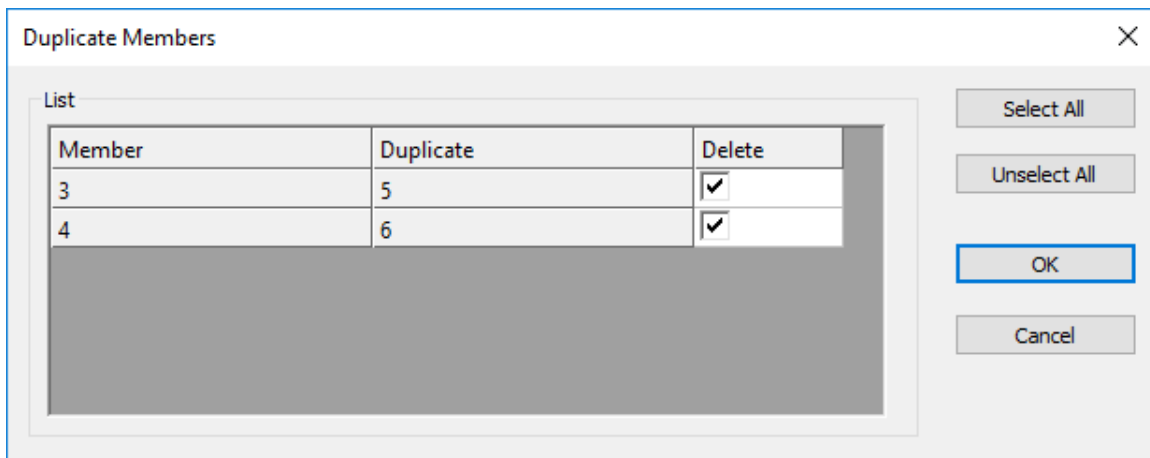
2.6.24. Members Duplicates

In order to check if two members are one on the top of the other (having common joints in any order), you must check the model for duplicate members from the ribbon command



Members Duplicates

in the “Find/Change/Check” panel or select “GTS Modeling>Checks>Members Duplicates” from the menu or type `GTSCheckDuplicateMembers` at the command prompt. If duplicate members exist in the structure, a new dialog appears having the full set of duplicate pairs, where you can select the members to be deleted or not as shown in the following figure. For all the selected pairs, one member is deleted (the member first created is kept).



2.6.25. Members Zero Length

In order to check if a member has zero length, you can use the ribbon command



Members Zero Length

in the “Find/Change/Check” panel or select “GTS Modeling>Checks>Members Zero Length” from the Menu or type `GTSCheckMembersZeroLength` at the command prompt. Following the command execution, you must enter the desired tolerance (Enter Check Tolerance <0.001000>) and then a list of zero length members appears from where you can select which to delete.

2.6.26. Physical Members

Physical members in a frame model are consisted using two or more analytical members. In order to check if physical members exist in the structural entities, you can use the ribbon



Physical Members

command in the “Find/Change/Check” panel or select “GTS Modeling>Checks>Physical Members” from the Menu or type `GTSCheckPhysicalMembers` at the command prompt. The five (5) checks that are performed with their corresponding errors are as follows:

#1. Check that all members have the same Beta angle. If not, there is an error message:

Physical Member %s: Member %s has inconsistent BETA angle

#2. Check that all members have the same cross section. If not, there is an error message:

Physical Member %s: Member %s has inconsistent CROSS SECTION

#3. Checks that all member do not have internal releases. If not, there is an error messages:

Physical Member %s: Member %s has internal RELEASES

Physical Member %s: Starting Member %s has internal RELEASES at its end

Physical Member %s: Ending Member %s has internal RELEASES at its start

#4. Checks that there is a sequential order of members. If not, there is an error message:

Physical Member %s: Members %s (end) and %s (start) do not have common joint (are not SEQUENTIAL)

#5. Check that all members form a straight line. . If not, there is an error message:

Physical Member %s: Member %s is not LINEAR with the physical member

If all 5 checks are successful an informative message appears:

Physical Member %s: OK

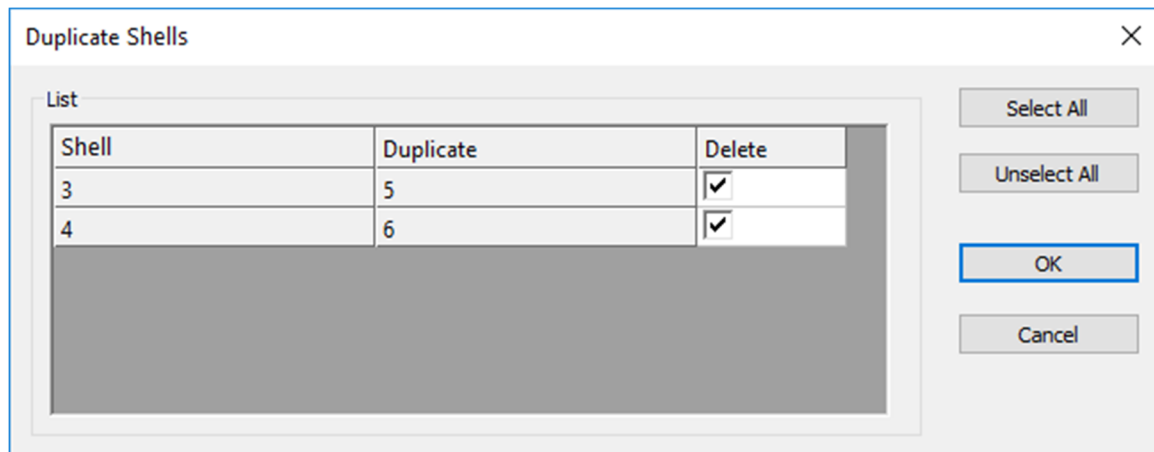
2.6.27. Shells Duplicates

In order to erase shells that coincide (one on the top of the other) that may have been generated by mesh generation functions, you have to check the model for duplicate shells from



Shells Duplicates

the ribbon command in the “Find/Change/Check” panel or select “GTS Modeling>Checks>Shells Duplicates” from the Menu or type `GTSCheckDuplicateShells` at the command prompt. If duplicate shells exist in the structure, a new dialog appears having the full set of duplicate pairs, where you can select the shells to be deleted or not as shown in the following figure. For all the selected pairs, one shell is deleted (the shell first created is kept).



2.6.28. Names Duplicates

In order to check if two joints have the same name or two members have the same name or a member and a shell have the same name, you can use the ribbon command



Names Duplicates

in the “Find/Change/Check” panel or select “*GTS Modeling>Checks>Names Duplicates*” from the Menu or type `GTSCheckNames` at the command prompt. If duplicate names exist in the structural elements, a message appears in the command prompt as follows:

```
Joints 4 and 4 have the same Name
Joints 5 and 5 have the same Name
2 Duplicate Names Found
```

2.6.29. Renumber Names

It is often convenient to have continuous numbering (labeling) of joints members and elements. In order to renumber the names of structural entities in ascending order (only if their name is an



Renumber Names

integer), you can use the ribbon command in the “Find/Change/Check” panel or select “*GTS Modeling>Checks>Names Duplicates*” from the Menu or type `GTSRenumber` at the command prompt. Following the command execution, you must decide whether or not you want to renumber joints and members/elements by typing yes or no (y or n) to the following questions:

```
Renumber Joints? (Yes/No)
Renumber Members and Elements? (Yes/No)
```

2.6.30. Database Integrity

In order to check that all CADM entities in a DWG have a unique database record, you can use



Database Integrity

the ribbon command in the “Find/Change/Check” panel or select “*GTS Modeling>Checks>Database Integrity*” from the Menu or type `GTSCheckDatabase` at the command prompt. There may be database records, without a dwg entity, or the opposite, dwg entities without a database record. This may happen in very extreme cases (for example, a power supply failure while saving, or out of disk space, etc). Entities/records with a problem are deleted and you get an information message such as the one below:

```
Error Fixed: Joint %s deleted from DWG
Error: Joint %s does not exist in database
```

2.6.31. Meshing along a curve

You can create several members along any selected AutoCAD/BricsCAD linear entity, that can be

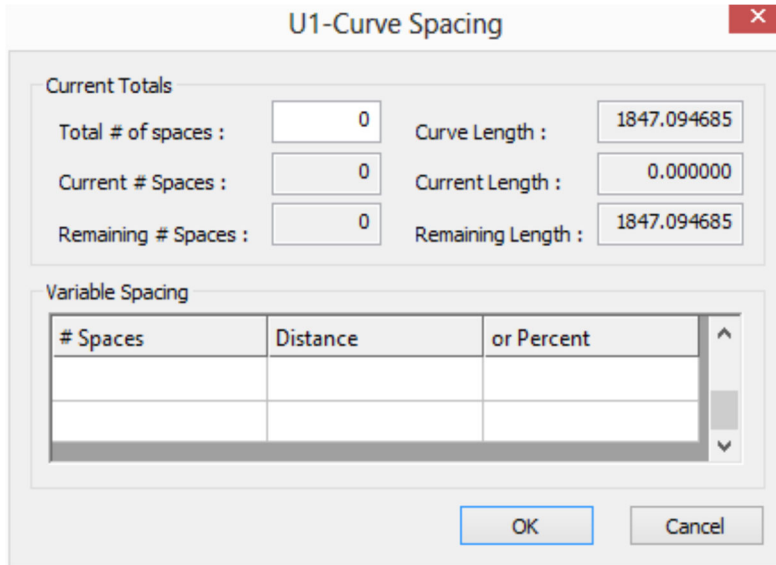


a Line, an Arc or a Circle, from the ribbon command **Curve** or from the menu “GTS Modeling>Mesh Generation>1D Along Line or Curve or Circle” or by typing `GTSMesh1D` at the command prompt.

After selecting the AutoCAD/BricsCAD linear entity the Mesh Properties dialog appears, where you can define:

- The *Material* of the members to be generated.
- The *Type* of the members (FRAME or TRUSS)
- The *Cross-Section* of the members from a list of previously selected project sections.
- The *Beta* angle that defines the orientation of the cross-section in 3D space.
- The number of members, to be generated, that can be equally spaced (*uniform*) or may have *variable* spacing.
- The *Labeling* (optional) of the joints and members to be generated (“Enter Labeling Rules” form), where you can set the First ID for joints and members and their labeling prefix.
- By clicking “*Preview*” you are able to preview the members to be generated (without creating any entities).
- By clicking “*Create*” the members are generated and the *Mesh Properties* form is closed.

By selecting Variable spacing, the “U1-Curve Spacing” form appears, where you can enter the *Total Number of Spaces*, and the *Length* of each part, either in absolute distance or as a percentage of the line or curve’s total length using the dialog shown below:



The "U1-Curve Spacing" dialog box is titled "U1-Curve Spacing" with a red close button. It contains two main sections: "Current Totals" and "Variable Spacing".

Current Totals:

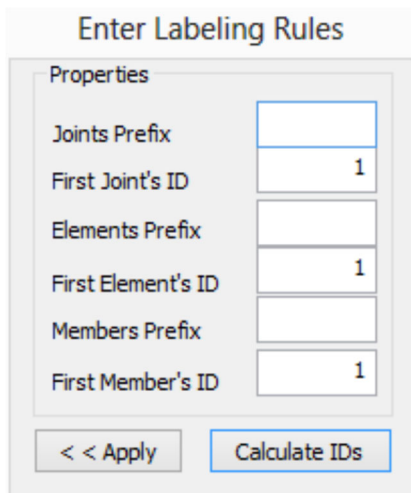
- Total # of spaces :
- Curve Length :
- Current # Spaces :
- Current Length :
- Remaining # Spaces :
- Remaining Length :

Variable Spacing:

# Spaces	Distance	or Percent

At the bottom are "OK" and "Cancel" buttons.

The “Enter Labeling Rules” form allows you to set the First ID for joints and members to be created and their labeling prefix. Note that the total length of the label cannot be more than 8 characters. Element and member labels (IDs) must be unique – you can’t have a member and element with the same label (ID).



The "Enter Labeling Rules" dialog box is titled "Enter Labeling Rules". It contains a "Properties" section with the following fields:

- Joints Prefix:
- First Joint's ID:
- Elements Prefix:
- First Element's ID:
- Members Prefix:
- First Member's ID:

At the bottom are "< < Apply" and "Calculate IDs" buttons.

2.6.32. Meshing between two lines

You can create Members or Finite Elements between two selected AutoCAD/BricsCAD linear



entities such as Lines or Arcs, from the ribbon command **2Curves** or from the menu “*GTS Modeling>Mesh Generation>2D Between 2 Lines or Curves*” or by typing `GTSMesh2D2L` at the command prompt. You are then asked to select two AutoCAD/BricsCAD curves that will define the U and V boundaries of the Mesh.

The dialog has the same options as in the 1D mesh command and in addition you can also define:

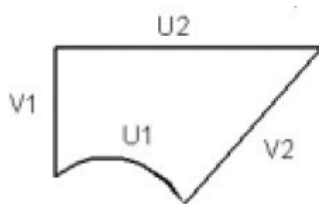
- Members or Elements to be generated (for Members the options are the same as in 1D)
- *Type* of Finite Elements, from the available GT STRUDL Finite Element library
- *Thickness* of Finite Elements
- Spacing in both the U and V directions

2.6.33. Meshing between four lines

You can create Members OR Finite Elements between four selected AutoCAD/BricsCAD linear



entities, that can be Lines or Arcs, from the ribbon command **4Curves** or from the menu “*GTS Modeling>Mesh Generation>2D Between 4 Lines or Curves*” or by typing `GTSMesh2D4L` at the command prompt. You are then asked to select four AutoCAD/BricsCAD curves that will define the U1, U2, V1 and V2 boundaries of the Mesh as shown in the figure below with U2 opposite U1 and V2 opposite V1:



The dialog has the same options as in the 2D mesh between two curves command.

2.6.34. Meshing inside a polyline

You can create Finite Elements inside an AutoCAD/BricsCAD closed curve, that can be a Polyline



or a Circle, from the ribbon command **Area** or from the menu “*GTS Modeling>Mesh Generation>Inside Polyline*” or by typing `GTSMesh2DPoly` at the command prompt. You are then prompted to select the closed AutoCAD/BricsCAD polyline or circular curve.

After selecting the AutoCAD/BricsCAD entity the *Mesh Properties* dialog appears, where you define:

Select Mesh Properties

Generate

Material Steel ...

Element Attributes

Type SBHT6 Thickness 0.20

Mesh Geometry

External Boundary obj-581 ...

External Boundary Edge Size 0.000000

Maximum Element Edge Size 0.50

Minimum Element Edge Size 0.000000

Mesh Quality Very Low

Internal Boundaries

Internal Joints

Add + Remove -

Multi+ Multi -

Add Remove

Spacing Extrude Direction

☒ Uniform 4 ...

☐ Variable ...

☐ Defined by Curve, Size: 3.242418

☐ Sweep Function

Labeling

More >>

Preview Clear Create Close

- The *Material* of the elements to be generated
- *Type* of Finite Elements from the available GT STRUDL Finite Element library
- *Thickness* of Finite Elements
- The *Maximum Element Edge Size* along the Boundary. CAD Modeler will generate additional joints along the boundaries so that there is no finite element edge, along the boundary curve, longer than the entered value.
- You can have the boundary curve to be split in smaller parts than the Max, or not. Additional splitting may be required if you try to increase the quality of the finite element mesh.
- The quality of the triangles that are going to be generated.
- Add one or multiple (Multi+) internal closed boundaries (polylines or circles), or open boundaries (arcs or lines). If a closed internal boundary is selected there is a question asking if you want the elements inside the boundary to be removed (treat it as a hole) or not. Moreover, you will be asked a question regarding the size of elements along the internal boundary curve, that can be 0, so as to follow the current value of the "*Maximum Element Edge Size*", or it can be a positive number which defines the maximum length along the internal boundary, or it can be a negative integer which defines the number of equal parts that the internal boundary will be split.

NOTE: Internal boundaries may intersect the external mesh curves, but it is recommended that you manually split the external curve at this point

- Add internal joints (points) that will be additional corners of the finite element mesh.
- Labeling, Preview and Create functions are identical to the ones of the previously described meshing forms.
-

2.6.35. Meshing by extruding a polyline

You can create Finite Elements by extruding an AutoCAD/BricsCAD closed curve, that can be a



Polyline or a Circle, from the ribbon command **3D Extrude** or from the menu “*GTS Modeling>Mesh Generation>3D Extrude PolyLine*” or by typing `GTSExtrudePoly` at the command prompt. You are then prompted to select AutoCAD/BricsCAD curves, first the extruded curve, and then the curve which defines the extrude direction which can be either a line or polyline. The finite elements will be generated on the extruded surface.

The “*Mesh Properties*” form is similar to the “*Meshing inside a polyline*” properties form but you must also define the “*Spacing Extrude Direction*”, meaning the parameters that control the size of elements along the extrude direction. *Uniform* and *Variable* options are suitable if the extrude entity is a line. “*Defined by Curve, size*” is suitable if the extrude entity is a polyline, so that it is enforced that joints will be generated at the intermediate points of the polyline. Labeling, Preview and Create functions are identical to the ones of the previously described meshing forms.

2.6.36. Meshing using 3 curves

You can create Members OR Finite Elements between three selected AutoCAD/BricsCAD linear



entities, that can be Lines or Arcs, from the ribbon command **3Curves** or from the menu “*GTS Modeling>Mesh Generation>3D Between 3 Lines or Curves*” or by typing `GTSMesh3D3L` at the command prompt. You are then asked to select three AutoCAD/BricsCAD curves that will define the U, V and W boundaries of the Mesh.

The dialog has the same options as in the 2D mesh command, between 2 lines, with the extra parameters for the meshing in the W direction.

2.6.37. Array 3D Advanced

You can create copies along the vertical direction, similar to AutoCAD's and BricsCAD's ARRAY



commands from the ribbon command **Array 3D Advanced** or by typing `GTSMesh3D3` at the command prompt. This optimized command is useful for large models, or for multiple copies, when the default AutoCAD's and BricsCAD's command take a lot amount of time. It produces the same result in significantly reduced time.

2.6.38. Soil Springs



You can create soil springs on individual/multiple shell element(s) from the icon **Soil Springs** or by typing `GTSTFoundationSprings` at the command prompt and select the corresponding shell element(s). After selection the "Soil Springs" dialog appears, where you can:

- Define in-plane spring ratio of Ks value if the "in-plane (orthogonal) springs" option is checked. The in-plane spring ratio will be applied to the two remaining directions, other than the selected.
- Define the direction of springs (in global system).
- Append new spring values in case of keeping the existing ones and adding the new ones. Else, the new values replace the existing ones
- Define the value of modulus of subgrade reaction Ks.

After pressing "Place Springs", a legend appears showing the Ks values distribution on the shell elements. By double clicking on each node, you can see the value of Ks in "Restraints and Spring Values" section of the Model tab in the "Joint Properties" dialog box. Note that if the spring's degree of freedom is not restrained by the user then it is restrained automatically.

General Information

☐ In-plane (orthogonal) springs

In-plane Spring Ratio :

Direction

☐ Global X ☐ Global Y ☒ Global Z

☒ Append Existing Spring Values

Modulus of Subgrade Reaction

Ks :

Place Springs >>

Ks Values

-0.00 ... 0.08
0.08 ... 0.15
0.15 ... 0.23
0.23 ... 0.31
0.31 ... 0.38
0.38 ... 0.46
0.46 ... 0.54
0.54 ... 0.62
0.62 ... 0.69
0.69 ... 0.77
0.77 ... 0.85
0.85 ... 0.92
0.92 ... 1.00

2.6.39. Export to CAESAR II

The current drawing can be exported into CAESAR II Modeler (str file) from the menu “*GTS Modeling>CAESAR II> Export STR*” or by typing `GTSEXPORSTR` at the command prompt. 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
```

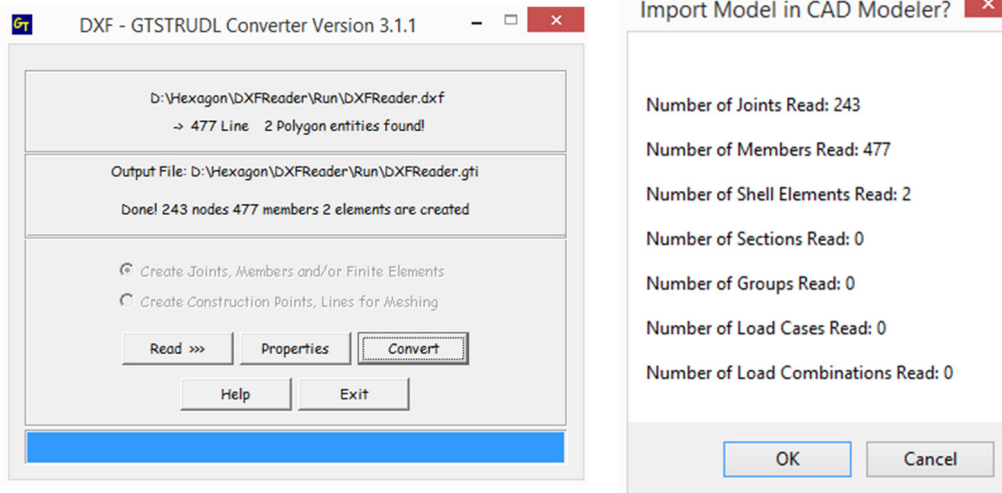
If the cross sections used in CAD Modeler (and GT STRUDL) are not available in CAESAR’s section library you get a warning like this:

```
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
```

2.6.40. Convert Lines/Polylines to Members/Shells

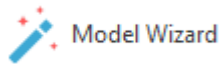
Convert lines and polylines to members and shell elements through the utility “DXF converter”,

from the ribbon command  **Convert Lines to Members** or by typing `GTSDXFRead` at the command prompt. This command is not limited to DXF Files, but you can create new lines and polylines on the DWG that is currently open and convert them to structural members. Or open an existing DXF or DWG, select some lines and polylines, and convert them to members and elements.



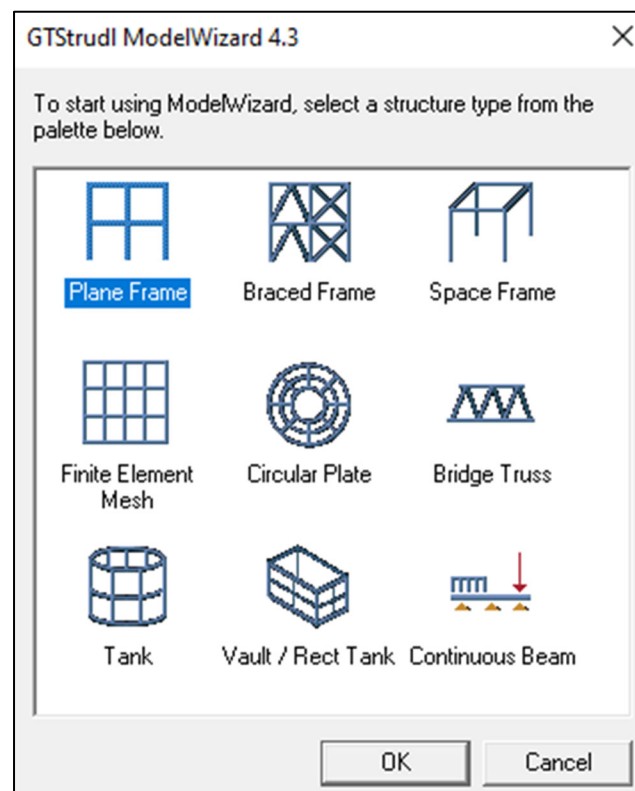
2.6.41. Model Wizard

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



Model Wizard

or by typing `GTSMModelWizard` at the command prompt.



2.6.42. Groups

It is optional to define Groups in your model, but it is strongly advised to do so, since it will be easier to control the display and modeling of parts of your structure. Each Group is defined as a set of joints, members and finite elements. Each structural entity can belong in more than one group. Moreover, Groups defined in CAD Modeler are exported to the GT STRUDL Input file (.gti), meaning you can also use them in GT STRUDL analysis and design commands.



You have to first define the name of each group from the ribbon icon **List** or from the menu “GTS Modeling>Groups>Manage” or by typing `GTSGroups` at the command prompt.

Using the Groups Form you can:

- Set the *Name* of each Group (NOTE: not larger than 8 characters, due to a GT STRUDL limitation)
- *Add Groups* to the model
- *Delete Groups* from the model
- Define if this Group is corresponding in a “Physical” member definition (used in Steel Design commands).

ID	Name	Physical
1	Columns	☐
2	Beams	☐
3	Girders	☐
4	Bracing	☐
5	Slabs	☐

After defining a group you can add joints, members and shell elements to it using the commands:

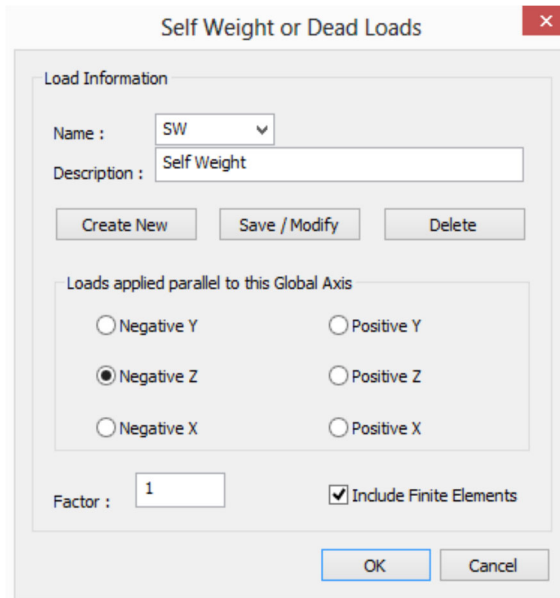
 **+Joints** ribbon icon, or “GTS Modeling>Groups>Add Joints” or by typing `GTSGroupJoints` at the command prompt

 **+Members** ribbon icon, or “GTS Modeling>Groups>Add Members” or by typing `GTSGroupMembers` at the command prompt

 **+Shells** ribbon icon, or “GTS Modeling>Groups>Add Shells” or by typing `GTSGroupShells` at the command prompt

2.6.43. Self - Weight

The Self-weight load of the structure can be created from the ribbon command  **Self Weight** or from the menu “GTS Modeling>Loads>Self Weight” or by typing `GTSSelfWeight` at the command prompt.



The dialog box titled "Self Weight or Dead Loads" contains the following fields and controls:

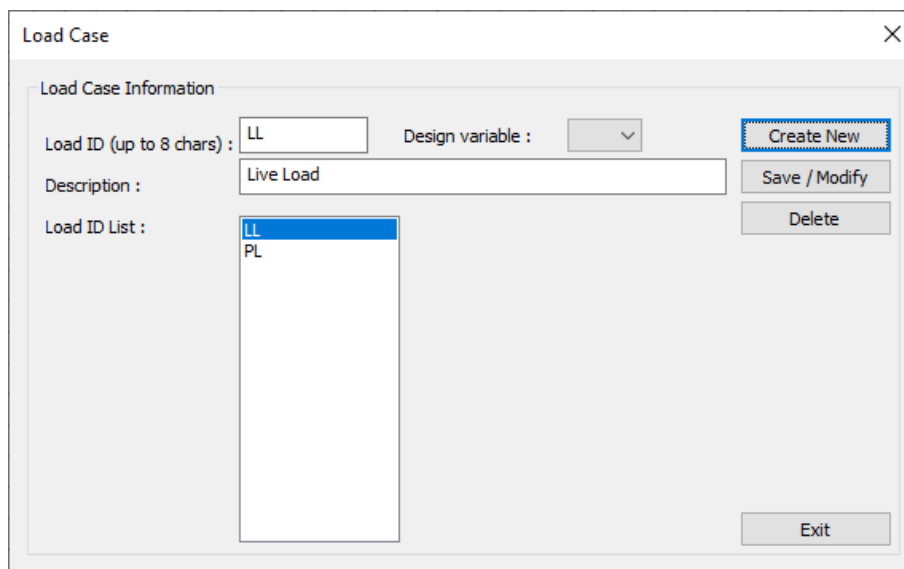
- Load Information** section:
 - Name :** A dropdown menu with "SW" selected.
 - Description :** A text field containing "Self Weight".
 - Buttons: "Create New", "Save / Modify", and "Delete".
- Loads applied parallel to this Global Axis** section:
 - Radio buttons for "Negative Y", "Positive Y", "Negative Z", "Positive Z", "Negative X", and "Positive X". "Negative Z" is selected.
- Factor :** A text field containing "1".
- ☒ **Include Finite Elements**
- Buttons: "OK" and "Cancel".

The “Self-Weight” form appears where you can define:

- the global direction of the self-weight
- the load factor (default = 1.0) for the self-weight
- if the self-weight of finite elements will be taken into account or not

2.6.44. Load Cases

A new load case can be created from the ribbon command  **Load Cases** or from the menu “GTS Modeling>Loads>Load Cases” or by typing `GTSNewLoadCase` at the command prompt. The “Load Case” form appears as shown below where you can enter new load cases, modify existing ones, or delete them.



The dialog box titled "Load Case" contains the following fields and controls:

- Load Case Information** section:
 - Load ID (up to 8 chars) :** A text field containing "LL".
 - Design variable :** A dropdown menu.
 - Buttons: "Create New", "Save / Modify", and "Delete".
- Description :** A text field containing "Live Load".
- Load ID List :** A list box containing "LL" and "PL", with "LL" selected.
- Buttons: "Exit".

Design variable associates the Load Case with the Design Load Variable. This information can be used later on, when creating the standardized load combinations.

▼ Create New

- D Dead load
- DI Weight of ice
- E Earthquake load
- F Fluid loads having well - defined pressures and maximum heights or depths
- FA1 Flood loads F a in V - Zones or Coastal A - Zones
- FA2 Flood loads F a in noncoastal A - Zones
- HA Lateral pressure load, H, due to earth, ground water, or bulk materials, when H adds to the primary variable(live) load effect
- HR Lateral pressure load, H, due to earth, ground water, or bulk materials, when H resists the primary variable(live) load effect
- L Live load
- LR Roof live load
- R Rain load
- S Snow load
- T Self straining load, such as a temperature load that causes thermal stresses
- W Wind load
- WI Wind - on - ice load
- AK Extra ordinary event load, such as blast, fire, etc.
- N Notional load
- L100 Live load that is permitted to be less than or equal to 100 psf.in a given load combination
- LEX Loading conditions associated with LEX are live loads L that are treated as Exclusive loads in the creation of the standardized loading combinations

2.6.45. Joint Loads

A Joint Load can be entered from the ribbon command  **Joint** or from the menu “GTS Modeling>Loads>Joint Load” or by typing `GTSJointLoad` at the command prompt. You then have to select the joint or the joints that the load will be applied to.

At the “Joint Generalized Loads” tab you can apply joint loads or displacements. On the left part of the form, you can see a list of all available load cases: Load cases having loads already applied to the specific joint appear at the top list box. Load cases that do not have any loads applied to the specific joint appear at the bottom list box. Next to the name of each load case there is a \$ symbol followed by the total number of joints that are already loaded in the specific load case.

Joint Properties

Model Joint Generalized Loads

Applied Load Cases

- PL \$ Point Load \$[1]

Empty Load Cases

- LL \$ Live Load \$[0]

Loading

Load Case : PL Create New

Description : Point Load Save / Modify

Loads in this Load Case : 1 Delete

Applied Load Values

Force X : 0 Force Y : 0 Force Z : -3

Moment X : 0 Moment Y : 0 Moment Z : 0

Applied Displacement Values

Tran X : Tran Y : Tran Z :

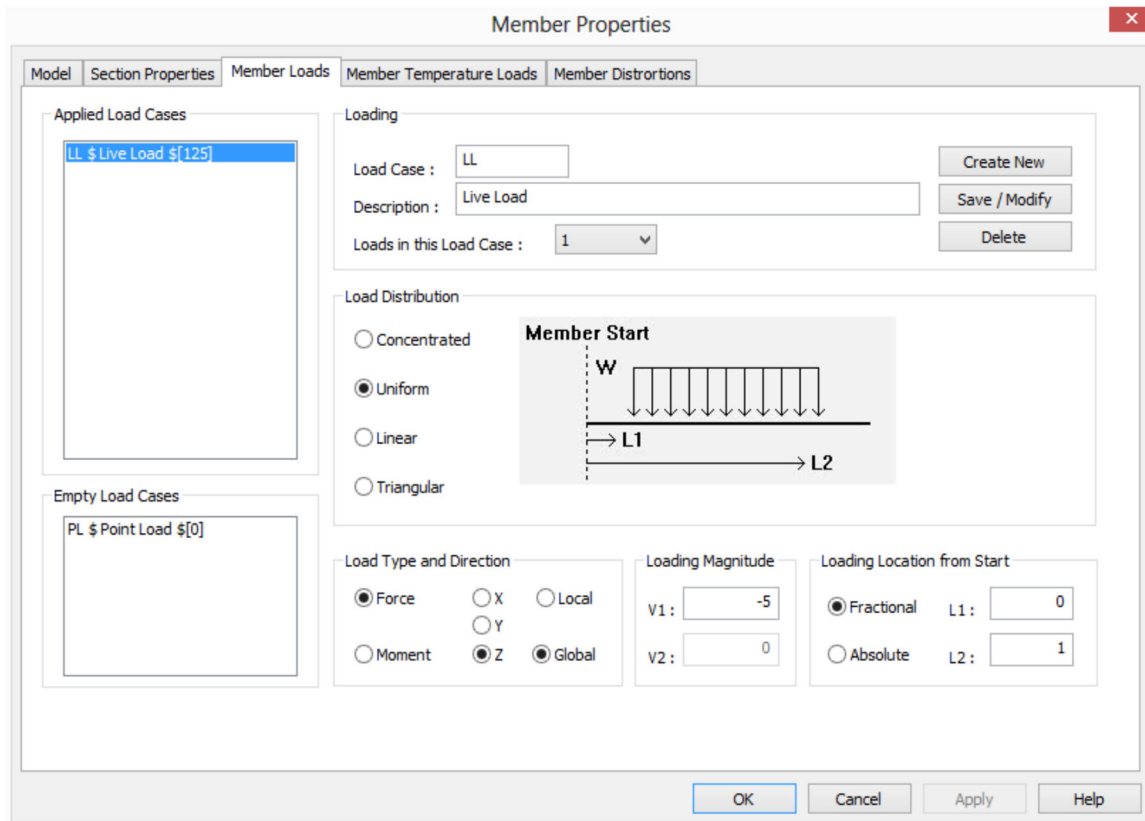
Rot X : Rot Y : Rot Z :

OK Cancel Apply Help

2.6.46. Member Loads

A Member Load can be entered from the ribbon command  **Member** or from the menu “GTS Modeling>Loads>Member Load” or by typing `GTSBeamLoad` at the command prompt. You then have to select the member or the members that the load will be applied to.

At the “Member Loads” tab or the Member Properties form as shown below, you can apply member loads. In the left part of the form you can see a list of all available load cases: Load cases having loads already applied to the specific member appear in the top list box. Load cases that do not have any loads applied to the specific member appear in the bottom list box. Next to the name of each load case there is a \$ symbol followed by the total number of members that are already loaded in the specific load case. The load distribution can be Concentrated, Uniform, Linear or Triangular and it can be applied in any local member or global direction. The Location can be entered in fractional terms (0: start, 1: end) or absolute terms in current length units measured from the start of the member.



The screenshot shows the "Member Properties" dialog box with the "Member Loads" tab selected. The dialog has four tabs: "Model", "Section Properties", "Member Loads", "Member Temperature Loads", and "Member Distortions".

Applied Load Cases: A list box containing "LL \$ Live Load \$[125]".

Empty Load Cases: A list box containing "PL \$ Point Load \$[0]".

Loading:

- Load Case:
- Description:
- Loads in this Load Case:
- Buttons: "Create New", "Save / Modify", "Delete"

Load Distribution:

- ☐ Concentrated
- ☒ Uniform
- ☐ Linear
- ☐ Triangular

Member Start: A diagram showing a horizontal member with a vertical dashed line at the start. A uniform load is represented by a series of downward arrows. The member is labeled with "W" at the start, "L1" at the end of the load, and "L2" at the far right end.

Load Type and Direction:

- ☒ Force
- ☐ Moment
- ☐ X
- ☐ Y
- ☒ Z
- ☐ Local
- ☒ Global

Loading Magnitude:

- v1:
- v2:

Loading Location from Start:

- ☒ Fractional
- ☐ Absolute
- L1:
- L2:

Buttons at the bottom: "OK", "Cancel", "Apply", "Help".

Using the “Member Temperature Loads” tab, you can define Axial or Bending temperature change along a part of the member, similar to the “Member Loads” tab as shown on the next page.

Finally, using the “Member Distortions” tab, you can define concentrated or uniform distortion of the member in any direction along the member as shown on the next page.

Member Properties

Model Section Properties Member Loads **Member Temperature Loads** Member Distortions

Applied Load Cases

Empty Load Cases

LL \$ Live Load \${0}
 PL \$ Point Load \${0}

Loading

Load Case :

Description :

Loads in this Load Case :

Temperature Change

☒ X Axial Uniform through cross-section
☐ Y Bending Vary through depth Change :
☐ Z Bending Vary through width

Location: Length experiencing temperature change

☒ Fractional (% 0-1) Starting Location :
☐ Absolute (len) Ending Location :

Member Properties

Model Section Properties Member Loads Member Temperature Loads **Member Distortions**

Applied Load Cases

Empty Load Cases

LL \$ Live Load \${0}
 PL \$ Point Load \${0}

Loading

Load Case :

Description :

Loads in this Load Case :

Distortion Type

☐ Concentrated
☒ Uniform

Distortion Direction - Value

☒ Displacement ☒ X
☐ Rotation ☐ Y Value :
☐ ☐ Z

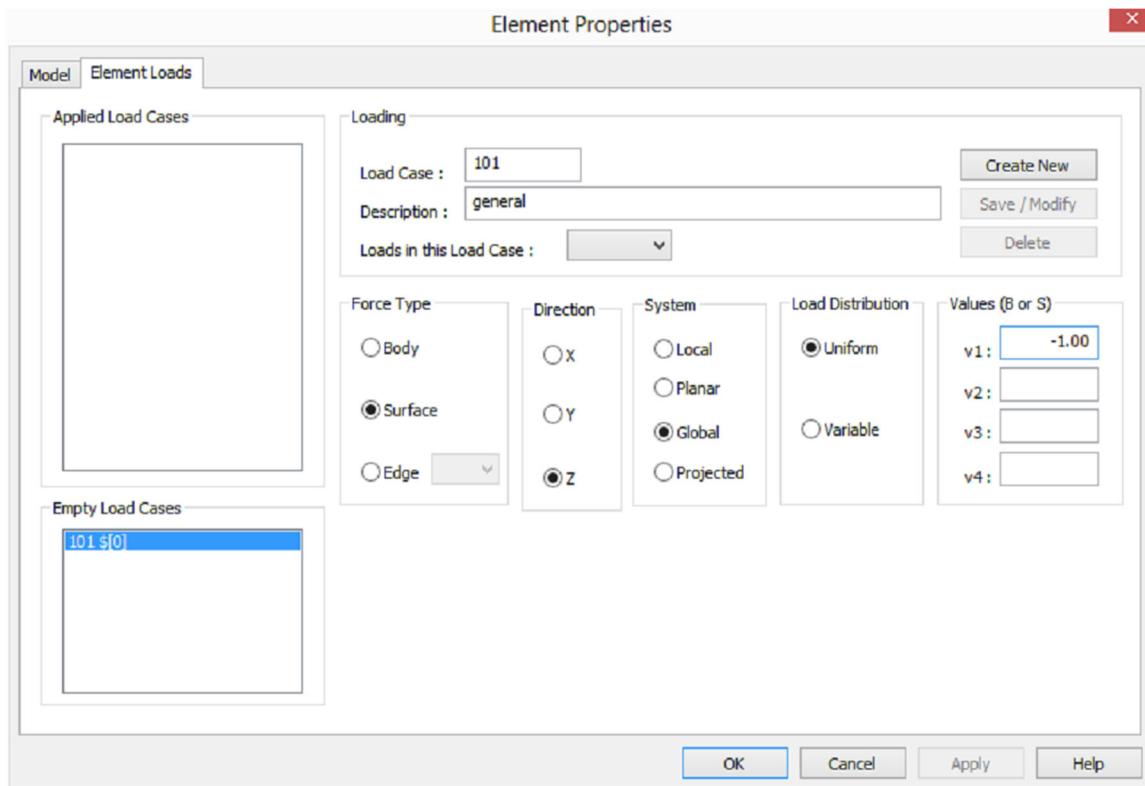
Distortion Location

☒ Fractional (% 0-1) LA :
☐ Absolute (Len) LB :

2.6.47. Shell Loads

A Shell Load can be entered from the ribbon command  **Shell** or from the menu “*GTS Modeling>Loads>Shell Load*” or by typing `GTSShellLoad` at the command prompt. You then have to select the shell or the shells that the load will be applied to.

Using the “*Element Loads*” tab of the Element Properties form, you can apply element loads. In the left part of the form, you can see a list of all available load cases: Load cases having loads already applied to the specific element appear in the top list box. Load cases that do not have any loads applied to the specific element appear in the bottom list box. Next to the name of each load case there is a \$ symbol followed by the total number of elements that are already loaded in the specific load case. The Force type can be Body, Surface or Edge, in any Local, Planar, Global or Projected direction with uniform or variable values.



Element Properties

Model Element Loads

Applied Load Cases

Empty Load Cases

101 \$[0]

Loading

Load Case : 101

Description : general

Loads in this Load Case : [dropdown]

Create New

Save / Modify

Delete

Force Type

☐ Body

☒ Surface

☐ Edge [dropdown]

Direction

☐ X

☐ Y

☒ Z

System

☐ Local

☐ Planar

☒ Global

☐ Projected

Load Distribution

☒ Uniform

☐ Variable

Values (B or S)

v1 : -1.00

v2 : [input]

v3 : [input]

v4 : [input]

OK Cancel Apply Help

2.6.48. Area Load

An Area Load can be entered from the ribbon command  **Area** or from the menu “*GTS Modeling>Loads>Area Load*” or by typing `GTSAreaLoad` at the command prompt.

Area Load ✕

Generate

Name : ▼

Description :

Load - Direction

Load Value :

Global Direction Perpendicular to the Loading Plane :

☐ X
 ☐ Y
 ☒ Z

Plane Tolerance :

Elevation

Plane Perpendicular at :

☒ Value (coordinate)

☐ Joint ▼

Distribution

☒ Two way
 ☐ X
 ☐ One way
 ☐ Y

☐ Custom
 X
Y

Advanced Features

Using the Area Load form you can define:

- The Name of the Load (up to 8 characters)
- The description of the load
- The Loading Value in current units (force/length²). A positive value is applied in the negative global DIRECTION.
- The Global direction of the loading plane (X, Y or Z) and the tolerance
- The position (Elevation) of the plane, defined by a coordinate or an existing Joint. All members belonging to the plane having this elevation are located and loaded by GT STRUDL.
- The Distribution of the load (one way or two way)

Advanced Optional Features:

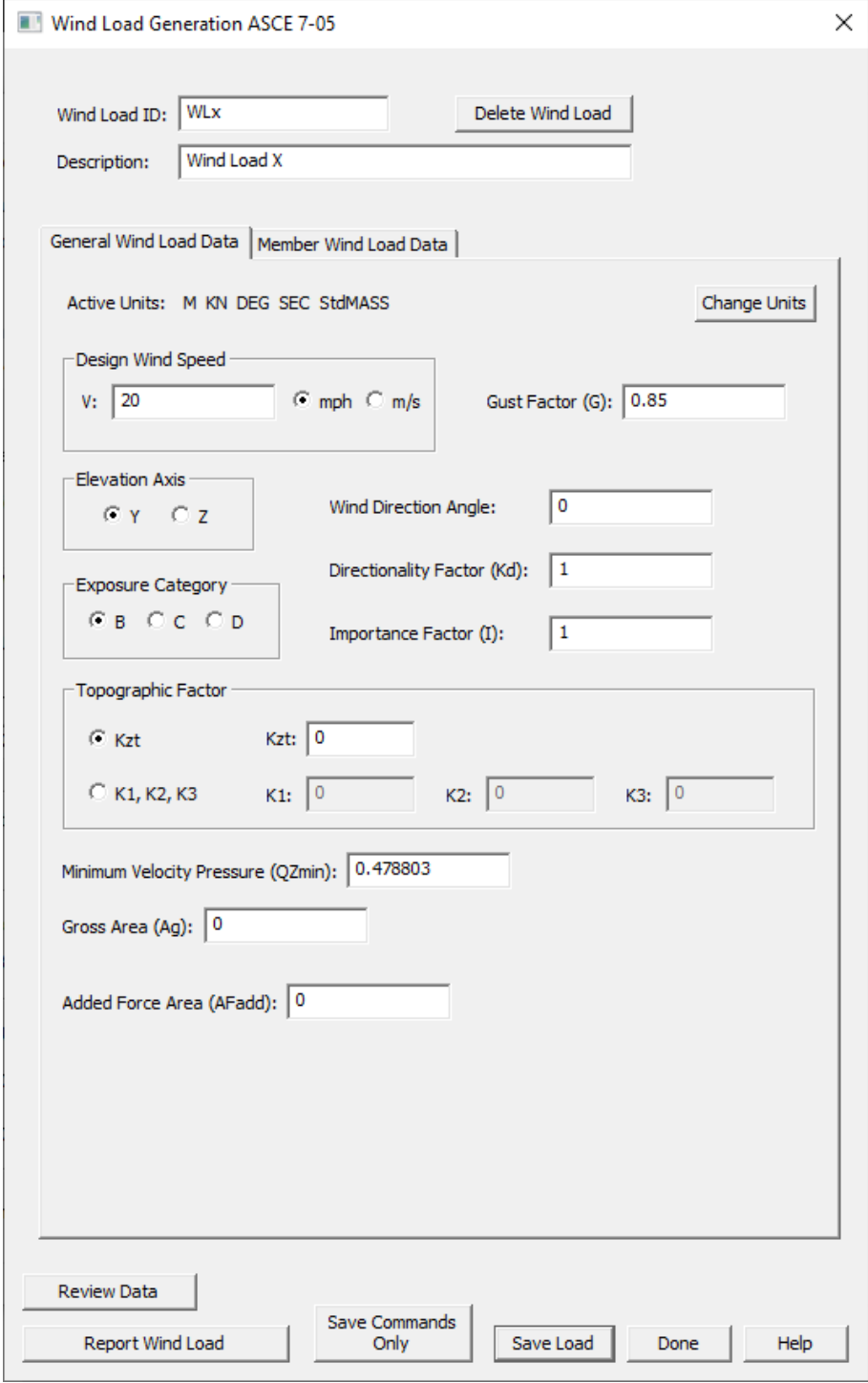
- Outline Region: Select the members that form the closed perimeter of the loaded area. If an Outline Region is defined then the area load is applied **ONLY** to the selected area and **NOT** to the whole plane having the elevation defined above.
- Exclude Area: Internal openings or islands not being loaded may be specified by selecting the members that define the perimeter of the excluded area.

- Ignore Members: Select members that you do not want to be loaded (eg bracing members)

By selecting "Display >>" you are able to graphically view the loaded area, marked with a yellow hatch pattern.

2.6.49. Wind Load ASCE 705

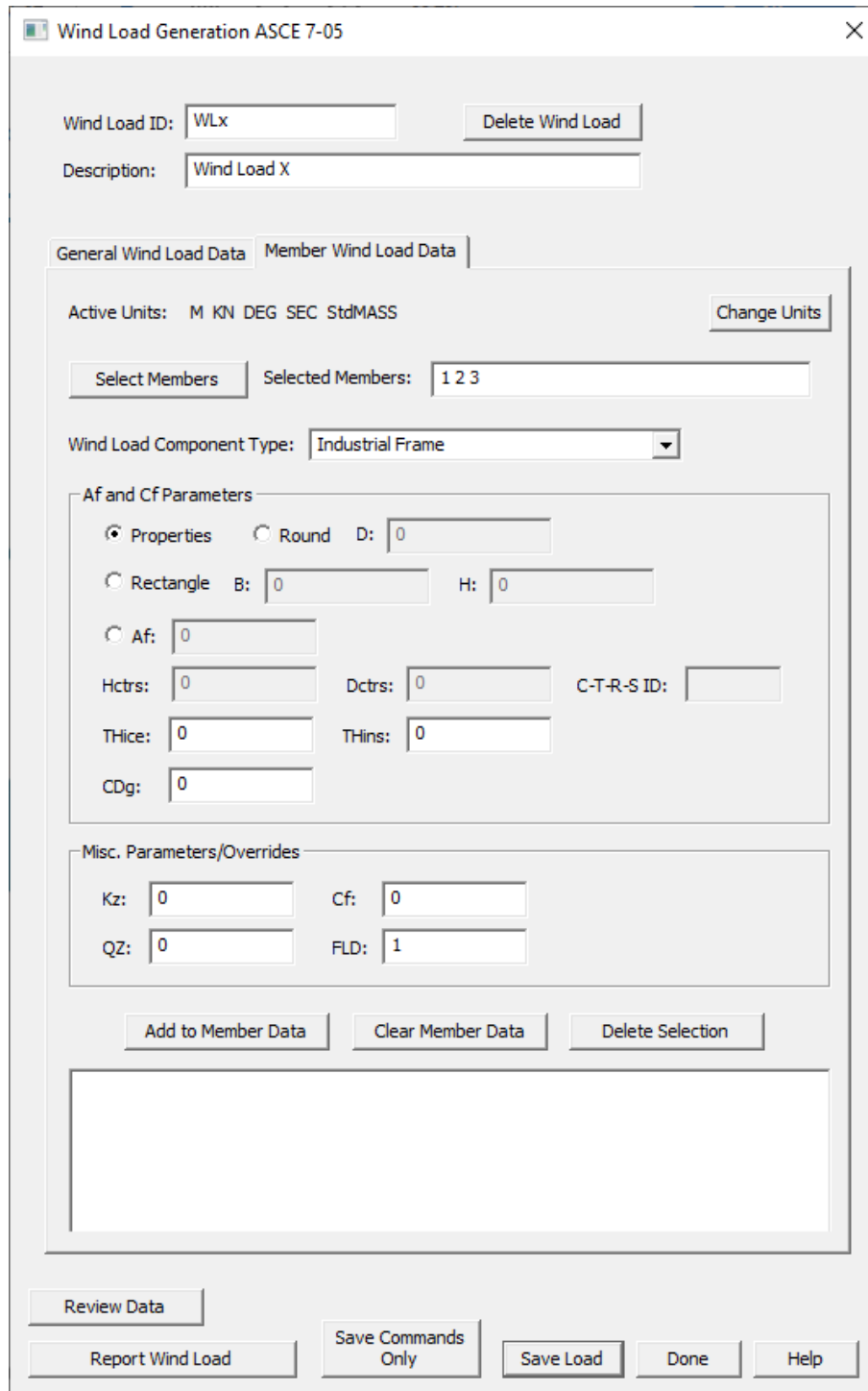
You can define Wind Loads using a similar form with GT Shell. A Wind Load can be entered from the ribbon command  **Wind Load ASCE 7-05** or from the menu “*GTS Modeling>Loads>Wind Load ASCE 705*” or by typing `GTSWindLoadsASCE705` at the command prompt.



The dialog box is titled "Wind Load Generation ASCE 7-05". It contains the following fields and controls:

- Wind Load ID:** Text box containing "WLx".
- Delete Wind Load:** Button.
- Description:** Text box containing "Wind Load X".
- General Wind Load Data / Member Wind Load Data:** Tabbed interface with "General Wind Load Data" selected.
- Active Units:** Text box containing "M KN DEG SEC StdMASS".
- Change Units:** Button.
- Design Wind Speed:** Section header.
- V:** Text box containing "20".
- mph / m/s:** Radio buttons, with "mph" selected.
- Gust Factor (G):** Text box containing "0.85".
- Elevation Axis:** Section header.
- Y / Z:** Radio buttons, with "Y" selected.
- Wind Direction Angle:** Text box containing "0".
- Exposure Category:** Section header.
- B / C / D:** Radio buttons, with "B" selected.
- Directionality Factor (Kd):** Text box containing "1".
- Importance Factor (I):** Text box containing "1".
- Topographic Factor:** Section header.
- Kzt:** Radio button, selected.
- Kzt:** Text box containing "0".
- K1, K2, K3:** Radio button, unselected.
- K1:** Text box containing "0".
- K2:** Text box containing "0".
- K3:** Text box containing "0".
- Minimum Velocity Pressure (QZmin):** Text box containing "0.478803".
- Gross Area (Ag):** Text box containing "0".
- Added Force Area (AFadd):** Text box containing "0".
- Review Data:** Button.
- Report Wind Load:** Button.
- Save Commands Only:** Button.
- Save Load:** Button.
- Done:** Button.
- Help:** Button.

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



The image shows a software dialog box titled "Wind Load Generation ASCE 7-05". It contains several input fields and buttons for configuring wind loads. At the top, there are fields for "Wind Load ID:" (containing "WLx") and "Description:" (containing "Wind Load X"), with a "Delete Wind Load" button. Below these are two tabs: "General Wind Load Data" and "Member Wind Load Data". The "General" tab is active, showing "Active Units:" as "M KN DEG SEC StdMASS" and a "Change Units" button. A "Select Members" button is next to a "Selected Members:" field containing "1 2 3". The "Wind Load Component Type:" is set to "Industrial Frame" in a dropdown menu. The "Af and Cf Parameters" section includes radio buttons for "Properties" (selected), "Round", and "Rectangle", with various input fields for dimensions and coefficients. The "Misc. Parameters/Overrides" section includes input fields for Kz, Cf, QZ, and FLD. At the bottom of the main area are three buttons: "Add to Member Data", "Clear Member Data", and "Delete Selection". A large empty rectangular box is located below these buttons. The footer contains five buttons: "Review Data", "Report Wind Load", "Save Commands Only", "Save Load", and "Done", along with a "Help" button.

Wind Load ID:

Description:

General Wind Load Data | Member Wind Load Data

Active Units: M KN DEG SEC StdMASS

Selected Members:

Wind Load Component Type:

Af and Cf Parameters

☒ Properties ☐ Round D:

☐ Rectangle B: H:

☐ Af:

Hctrs: Dctrs: C-T-R-S ID:

THice: THins:

CDg:

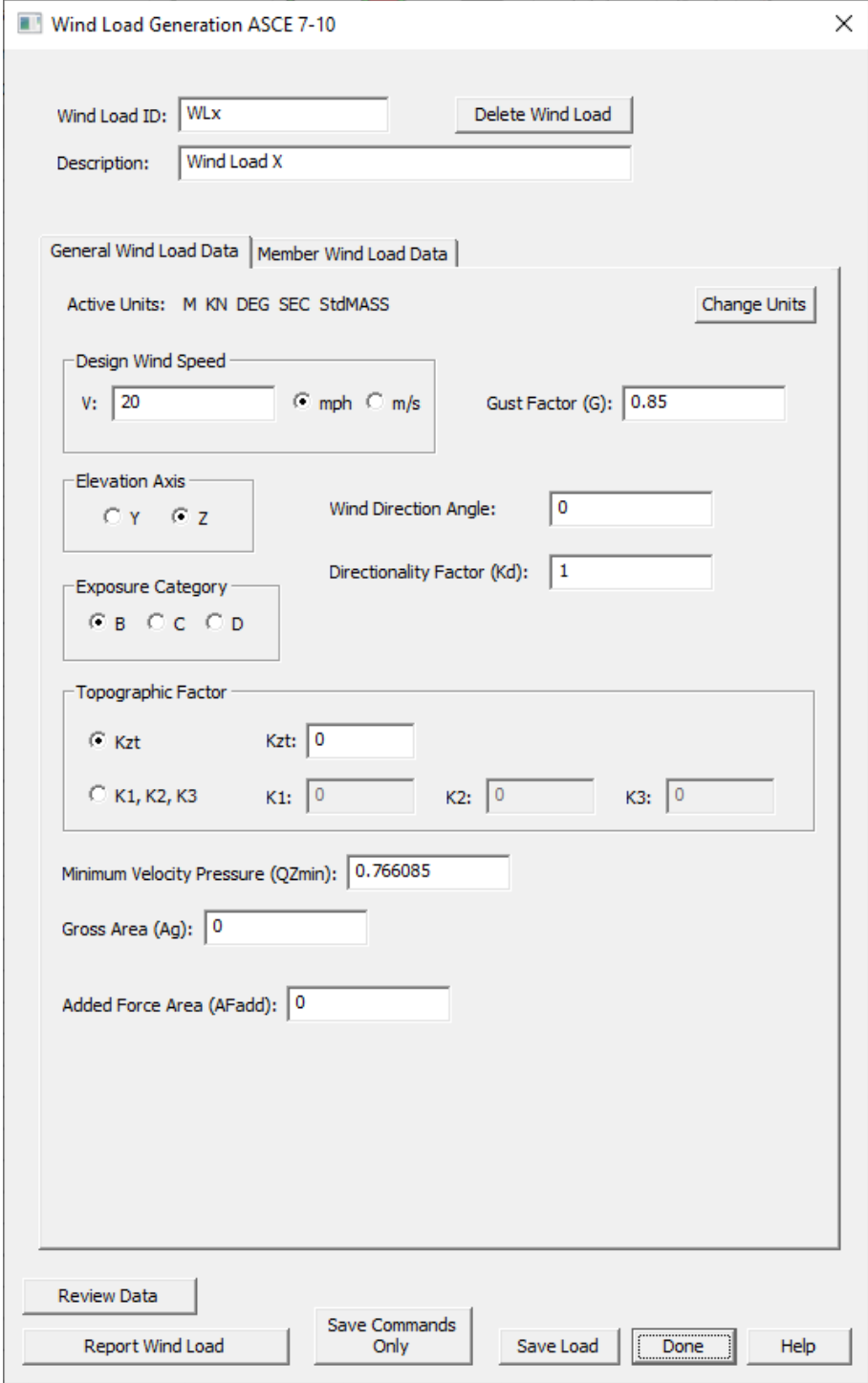
Misc. Parameters/Overrides

Kz: Cf:

QZ: FLD:

2.6.50. Wind Load ASCE 710

A Wind Load can be entered from the ribbon command  **Wind Load ASCE 7-10** or from the menu “*GTS Modeling>Loads>Wind Load ASCE 710*” or by typing `GTSWindLoadsASCE710` at the command prompt.



The dialog box is titled "Wind Load Generation ASCE 7-10" and contains the following fields and controls:

- Wind Load ID:**
- Description:**
- General Wind Load Data | Member Wind Load Data** (tabbed interface, currently on General)
- Active Units:** M KN DEG SEC StdMASS
- Design Wind Speed:**
 - V: ☒ mph ☐ m/s
 - Gust Factor (G):
- Elevation Axis:** ☐ Y ☒ Z
- Wind Direction Angle:**
- Exposure Category:** ☒ B ☐ C ☐ D
- Directionality Factor (Kd):**
- Topographic Factor:**
 - ☒ Kzt
 - ☐ K1, K2, K3
- Minimum Velocity Pressure (QZmin):**
- Gross Area (Ag):**
- Added Force Area (AFadd):**
- Buttons at the bottom:** Review Data, Report Wind Load, Save Commands Only, Save Load, Done, Help

Wind Load Generation ASCE 7-10
✕

Wind Load ID:
Delete Wind Load

Description:

General Wind Load Data
Member Wind Load Data

Active Units:
M KN DEG SEC StdMASS
Change Units

Select Members
Selected Members:

Wind Load Component Type:
Industrial Frame

Af and Cf Parameters

☒ Properties
☐ Round
D:

☐ Rectangle
B:
H:

☐ Af:

Hctrs:
Dctrs:
C-T-R-S ID:

THice:
THins:

CDg:

Misc. Parameters/Overrides

Kz:
Cf:

QZ:
FLD:

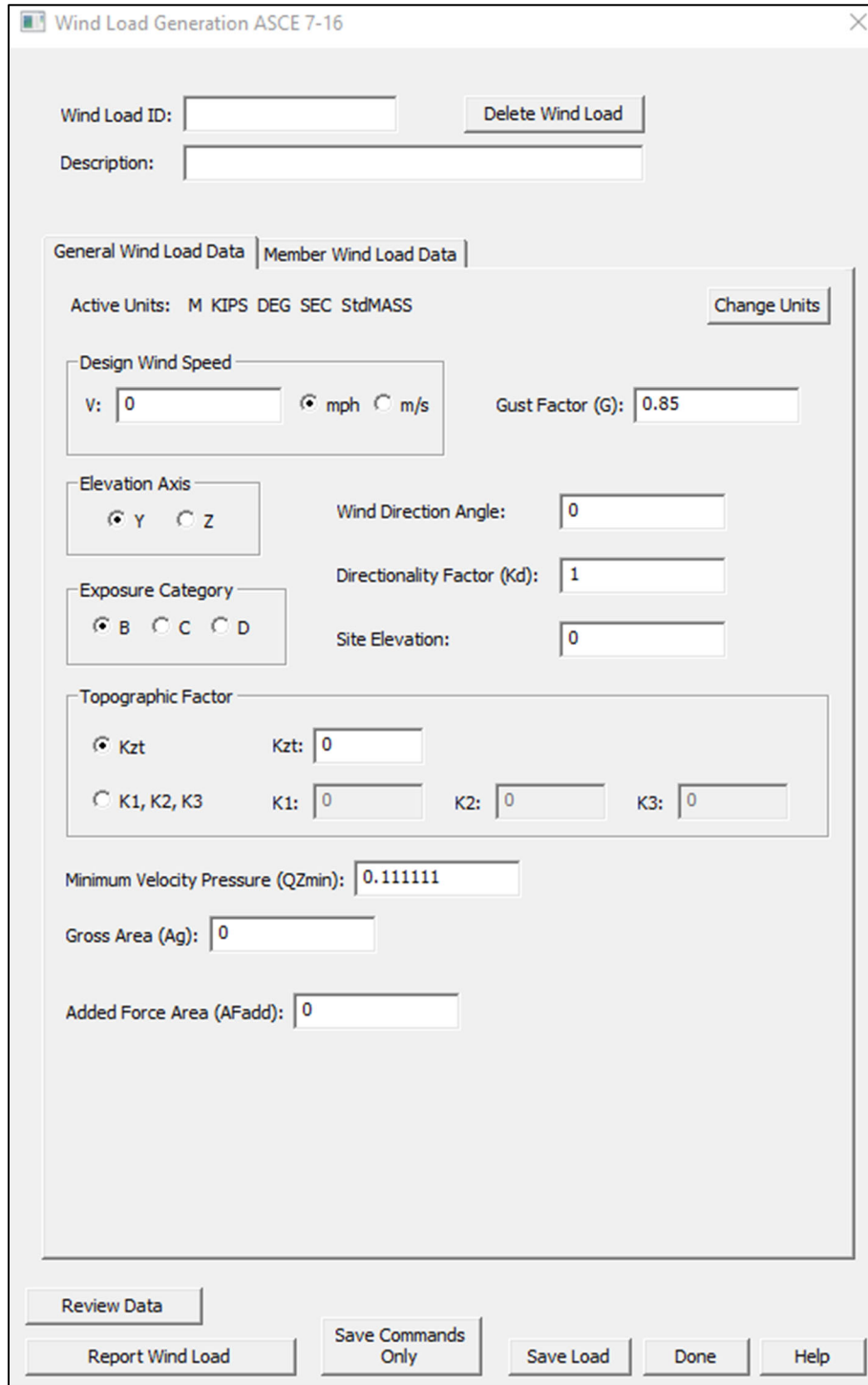
Add to Member Data
Clear Member Data
Delete Selection

Review Data

Report Wind Load
Save Commands Only
Save Load
Done
Help

2.6.51. Wind Load ASCE 716

A Wind Load can be entered from the ribbon command  **Wind Load ASCE 7-16** or from the menu “*GTS Modeling>Loads>Wind Load ASCE 716*” or by typing `GTSWindLoadsASCE716` at the command prompt.



Wind Load ID: Delete Wind Load

Description:

General Wind Load Data | Member Wind Load Data

Active Units: M KIPS DEG SEC StdMASS Change Units

Design Wind Speed

V: ☒ mph ☐ m/s Gust Factor (G):

Elevation Axis

☒ Y ☐ Z Wind Direction Angle:

Exposure Category

☒ B ☐ C ☐ D Directionality Factor (Kd):

Site Elevation:

Topographic Factor

☒ Kzt Kzt:

☐ K1, K2, K3 K1: K2: K3:

Minimum Velocity Pressure (QZmin):

Gross Area (Ag):

Added Force Area (AFadd):

Review Data Report Wind Load Save Commands Only Save Load Done Help

Wind Load ID:

Description:

General Wind Load Data | **Member Wind Load Data**

Active Units: M KIPS DEG SEC StdMASS

Selected Members:

Wind Load Component Type:

Af and Cf Parameters

☒ Properties ☐ Round D:

☐ Rectangle B: H:

☐ Af:

Hctrs: Dctrs: C-T-R-S ID:

THice: THins:

CDg:

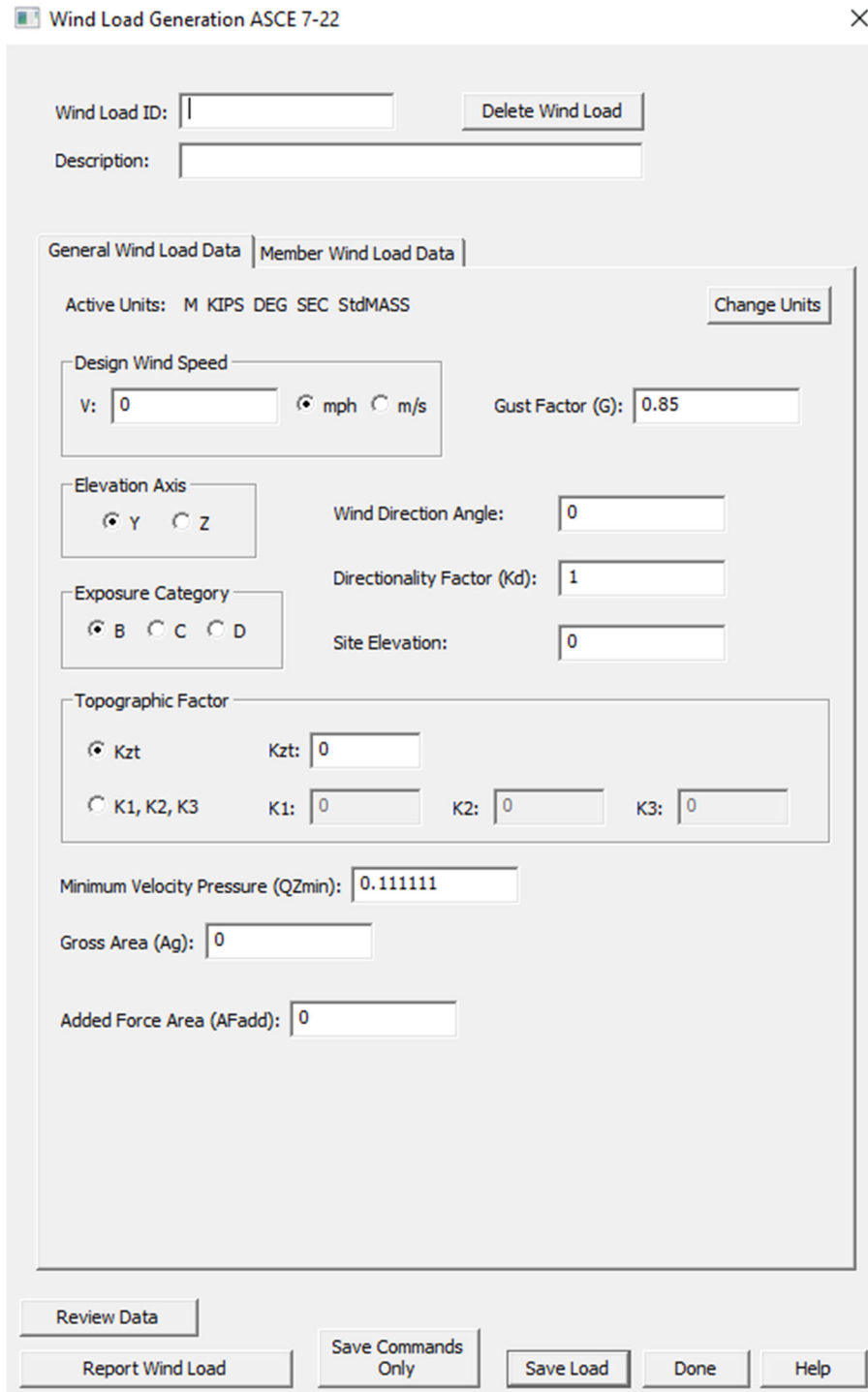
Misc. Parameters/Overrides

Kz: Cf:

QZ: FLD:

2.6.52. Wind Load ASCE 722

A Wind Load can be entered from the ribbon command  **Wind Load ASCE 7-22** or from the menu “*GTS Modeling>Loads>Wind Load ASCE 722*” or by typing `GTSWindLoadsASCE722` at the command prompt.



Wind Load ID:

Description:

General Wind Load Data | Member Wind Load Data

Active Units: M KIPS DEG SEC StdMASS

Design Wind Speed

V: ☒ mph ☐ m/s Gust Factor (G):

Elevation Axis

☒ Y ☐ Z Wind Direction Angle:

Exposure Category

☒ B ☐ C ☐ D Directionality Factor (Kd):

Site Elevation:

Topographic Factor

☒ Kzt Kzt:

☐ K1, K2, K3 K1: K2: K3:

Minimum Velocity Pressure (QZmin):

Gross Area (Ag):

Added Force Area (AFadd):

Wind Load ID:

Description:

General Wind Load Data | Member Wind Load Data

Active Units: M KIPS DEG SEC StdMASS

Selected Members:

Wind Load Component Type:

Af and Cf Parameters

☒ Properties ☐ Round D:

☐ Rectangle B: H:

☐ Af:

Hctrs: Dctrs: C-T-R-S ID:

THice: THins:

CDg:

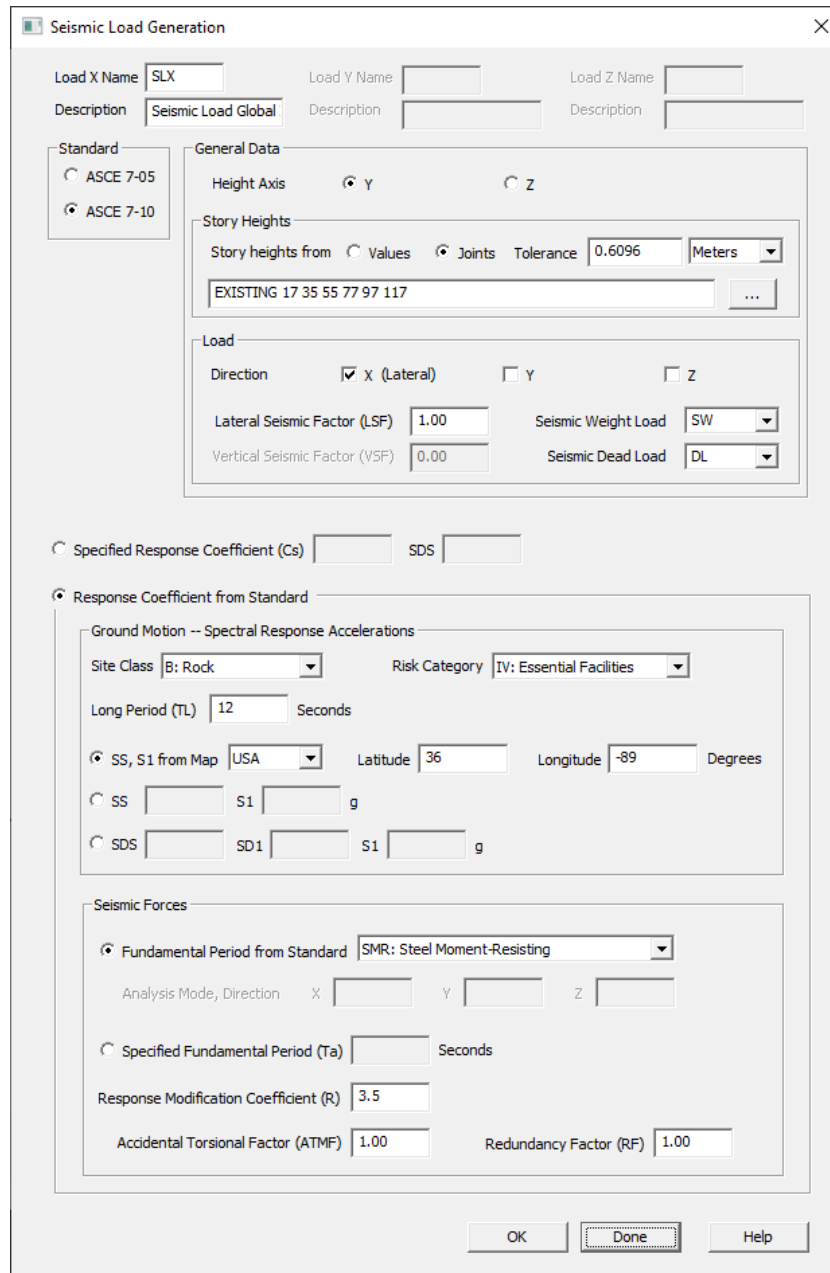
Misc. Parameters/Overrides

Kz: Cf:

QZ: FLD:

2.6.53. Seismic Load

You can define Seismic Loads using a similar form with GT Shell. Seismic Loads can be entered from the ribbon command  **Seismic Load** or from the menu “*GTS Modeling>Loads>Seismic Load*” or by typing `GTSSeismicLoading` at the command prompt. In addition you can select joints (to define story heights) interactively with mouse picks in the CAD environment.



The **Seismic Load Generation** dialog box is used to define seismic loads. It contains the following sections:

- General Data:**
 - Load X Name: Load Y Name: Load Z Name:
 - Description: Description: Description:
 - Standard: ☐ ASCE 7-05 ☒ ASCE 7-10
 - Height Axis: ☒ Y ☐ Z
 - Story Heights: Story heights from ☐ Values ☒ Joints Tolerance: Meters:
 - Load: Direction ☒ X (Lateral) ☐ Y ☐ Z Lateral Seismic Factor (LSF): Seismic Weight Load: Vertical Seismic Factor (VSF): Seismic Dead Load:
- Response Coefficient from Standard:**
 - Ground Motion -- Spectral Response Accelerations: Site Class: Risk Category: Long Period (TL): Seconds
 - ☒ SS, S1 from Map: Latitude: Longitude: Degrees
 - ☐ SS: S1: g
 - ☐ SDS: SD1: S1: g
- Seismic Forces:**
 - ☒ Fundamental Period from Standard: Analysis Mode, Direction: X: Y: Z:
 - ☐ Specified Fundamental Period (Ta): Seconds
 - Response Modification Coefficient (R):
 - Accidental Torsional Factor (ATMF): Redundancy Factor (RF):

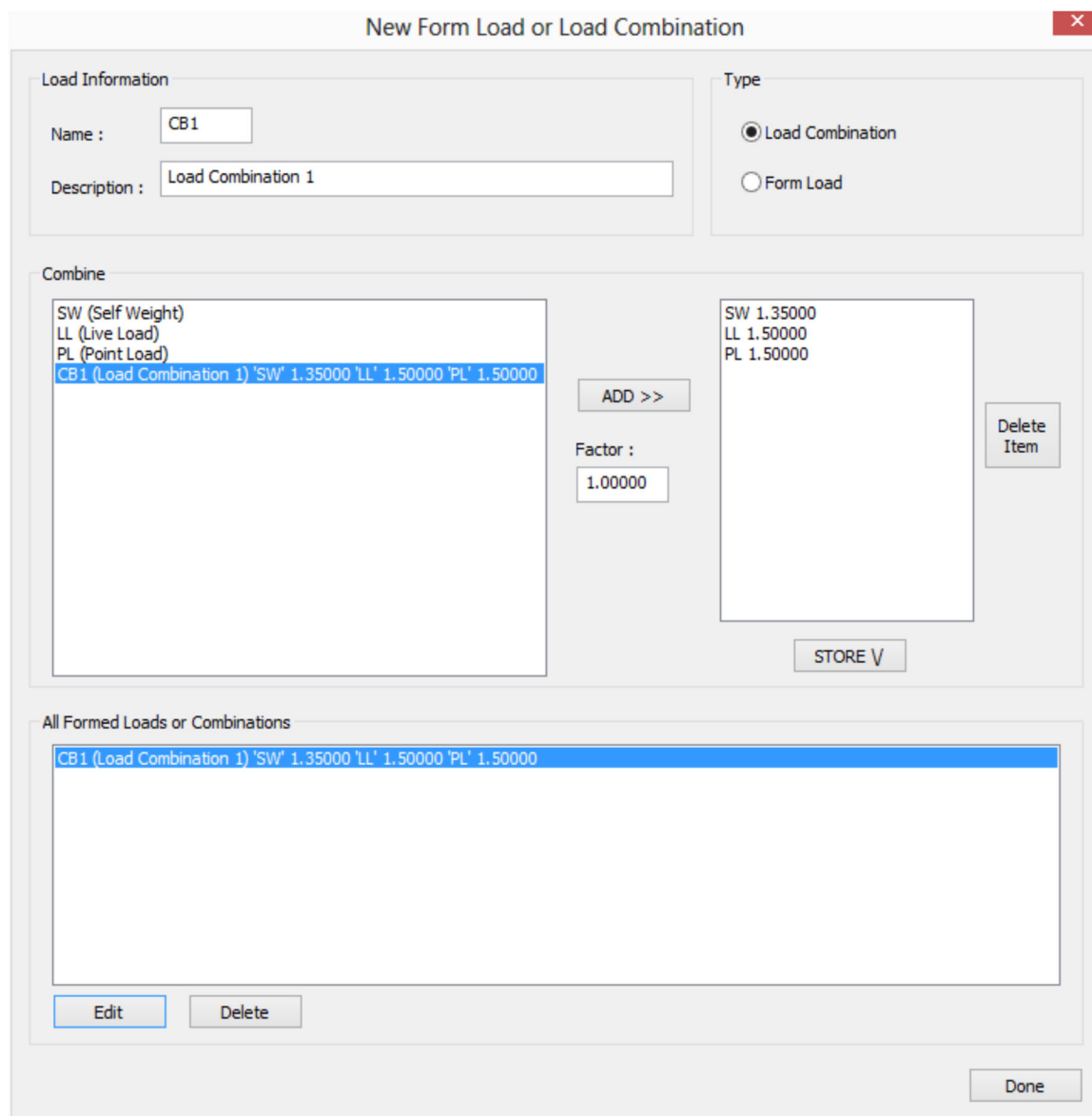
Buttons:

2.6.54. Load Combinations

A new load combination can be created from the ribbon command  **Combinations** or from the menu “GTS Modeling>Loads>Load Combinations” or by typing `GTSLoadCombination` at the command prompt. The “New Load Combination” form appears where you can enter new load combinations. For each Load Case or Load Combination, that appears in the left list box, you define a factor and using the “ADD>>” button the selected load case is added in the combination.

When all load cases are added, press the “STORE” button to store the load combination.

Using the same form, you can also Edit an Existing Load Combination (remember to press “STORE” after you are done with the modifications). You can also Delete an Existing Load Combination using this form.



The dialog box is titled "New Form Load or Load Combination" and contains the following sections:

- Load Information:**
 - Name:
 - Description:
- Type:**
 - ☒ Load Combination
 - ☐ Form Load
- Combine:**
 - Left list box: SW (Self Weight), LL (Live Load), PL (Point Load), CB1 (Load Combination 1) 'SW' 1.35000 'LL' 1.50000 'PL' 1.50000 (selected).
 - ADD >> button
 - Factor:
 - Right list box: SW 1.35000, LL 1.50000, PL 1.50000
 - Delete Item button
 - STORE button
- All Formed Loads or Combinations:**
 - List box: CB1 (Load Combination 1) 'SW' 1.35000 'LL' 1.50000 'PL' 1.50000 (selected)
 - Edit button
 - Delete button

Done button at the bottom right.

2.6.55. Standardized Combinations

You can Create Standardized Combinations using exactly the same form with GT Shell. Standardized combinations can be entered from the ribbon command



Standardized Combinations

or from the menu "*GTS Modeling>Loads>Standardized Combinations*" or by typing `GTSLoadCombinationStandardized` at the command prompt.

[illegible]

Create Standardized Load Combinations

Standard (Code) **ASCE 7-10 LRFD**

Definitions of DLVs

Associations **Create Loads**

Combination Selection

☒ All Combinations

☒ Ice Loading Check

Load Creation Options

☒ Load Combinations

☐ Form Loads

Start ID

Load Group **ASCE710L**

☒ Print Load Combos Report

Selection of Basic Combinations

	Basic Combinations
☒	Cmb. 1 1.4D
☒	Cmb. 2 1.2D + 1.6L + 0.5(LR or S or R)
☒	Cmb. 3 1.2D + 1.6(LR or S or R) + (L or 0.5W)
☒	Cmb. 4 1.2D + W + L + 0.5(LR or S or R)
☒	Cmb. 5 1.2D + E + L + 0.2S
☒	Cmb. 6 0.9D + W
☒	Cmb. 7 0.9D + E
☒	Cmb. 8 1.2D + AK + 0.5L + 0.2S
☒	Cmb. 9 1.2D + N + L + 0.2S

Reference for All Derived Combinations

	Derived Combinations
Cmb. 1	1.4(D + F) + T
Cmb. 2	1.2(D + F) + 1.6L + 1.6LEX + 0.5(LR or S or R) + 0.9HR + 1 <Ice> 1.2(D + F) + 1.6L + 1.6LEX + 0.2DI + 0.5S + 0.9HR
Cmb. 3	1.2(D + F) + 1.6(LR or S or R) + L + 0.5L100 + LEX + 0.9HR
Cmb. 4	1.2(D + F) + W + L + 0.5L100 + LEX + 0.5(LR or S or R) + C 1.2(D + F) + W + L + 0.5L100 + LEX + 0.5(LR or S or R) + 2 1.2(D + F) + 0.5W + L + 0.5L100 + LEX + 0.5(LR or S or R) <Ice> 1.2(D + F) + DI + WI + L + 0.5L100 + LEX + 0.5S +
Cmb. 5	1.2(D + F) + E + L + 0.5L100 + LEX + 0.2S + 0.9HR + 1.6H
Cmb. 6	0.9D + W + 0.9HR + 1.6HA + T 0.9D + W + 2.0FA1 + 0.9HR + 1.6HA + T 0.9D + 0.5W + FA2 + 0.9HR + 1.6HA + T <Ice> 0.9D + DI + WI + 0.9HR + 1.6HA + T
Cmb. 7	0.9(D + F) + E + 0.9HR + 1.6HA + T

OK Done Help

2.6.56. Steel Design Parameters

You can specify steel design parameters for AISC16, AISC15, AISC14, EC3, IS800, CSA-2019, CSA-2014 and other codes as shown in the CODE Name dropdown. Steel design parameters can be defined from the ribbon command  **Design** or menu “GTS Modeling>Steel Design Parameters” or by typing `GTSSteelDesignParameters` at the command prompt. Parameters can be applied to ALL members or to specific members that can be selected interactively with mouse picks in the CAD environment.

G_T Steel Design PARAMETERS Command

CODE Name: **AISC16** Units: **METERS** KILONEWTONS

Commonly Used | All Parameters | Alphabetic

Parameter Name	Parameter Values (Default Values)
Method	LRFD
ASDRF	1.0
CombEqs	Computed
TBLNAM	W-AISC16
CODETOL	0.0
SUMMARY	NO
Shortcut to Deflection Parameters (click here)	
SECTYPE	ROLLED

Member List Processing

All Members

Member List Selection

Same as Previous Member List

OK

Cancel

Help

UNITS

Help

AISC16

Based on the Manual of Steel Construction, American Institute of Steel Construction, Inc., Sixteenth Edition, Chicago, Illinois, 2022.

This code is applicable to the following cross-sections:
 1. I shapes, Channels, Single Angles, Round HSS (Pipes), and Rectangular and Square HSS (Structural Tubes) subjected to

Selected Order | Alphabetic

Parameter Names	Parameter Values	Member, Joint or Load	UNITS

2.6.57. Create GTI



A GT STRUDL Text Input file can be generated from the ribbon command **GTS** or from the menu *"GTS Modeling>Create GT.STRUDL GTI"* or by typing `GTSExportGTI` at the command prompt. In the *"Create GTSTRUDL Input File"* dialog, you can enter the filename of the GTI File and add additional commands to your GTI file, such as the Stiffness Analysis command to perform a static analysis automatically and commands which control the analysis results data that can be imported into CAD Modeler immediately after the analysis is complete.

Moreover, you can append additional GTI Files or Macros at the end of the GTI file of the model. For example, additional GTI files may include static or dynamic analysis commands, result output commands or member design commands. If you check “Copy Commands from GTI Files/Macros (not CINPUT)” then the above mentioned files or macros will be copied in your GTI file. Otherwise, there will be a reference to them, with a CINPUT command.

2.6.58. Edit GTI

The GT STRUDL Text Input file can be edited from the ribbon command  **Edit GTI** or from the menu “*GTS Modeling>Edit GT.STRU DL GTI*” or by typing `GTSEditGTI` at the command prompt. The previously created GTI is opened for editing using the default text editor.

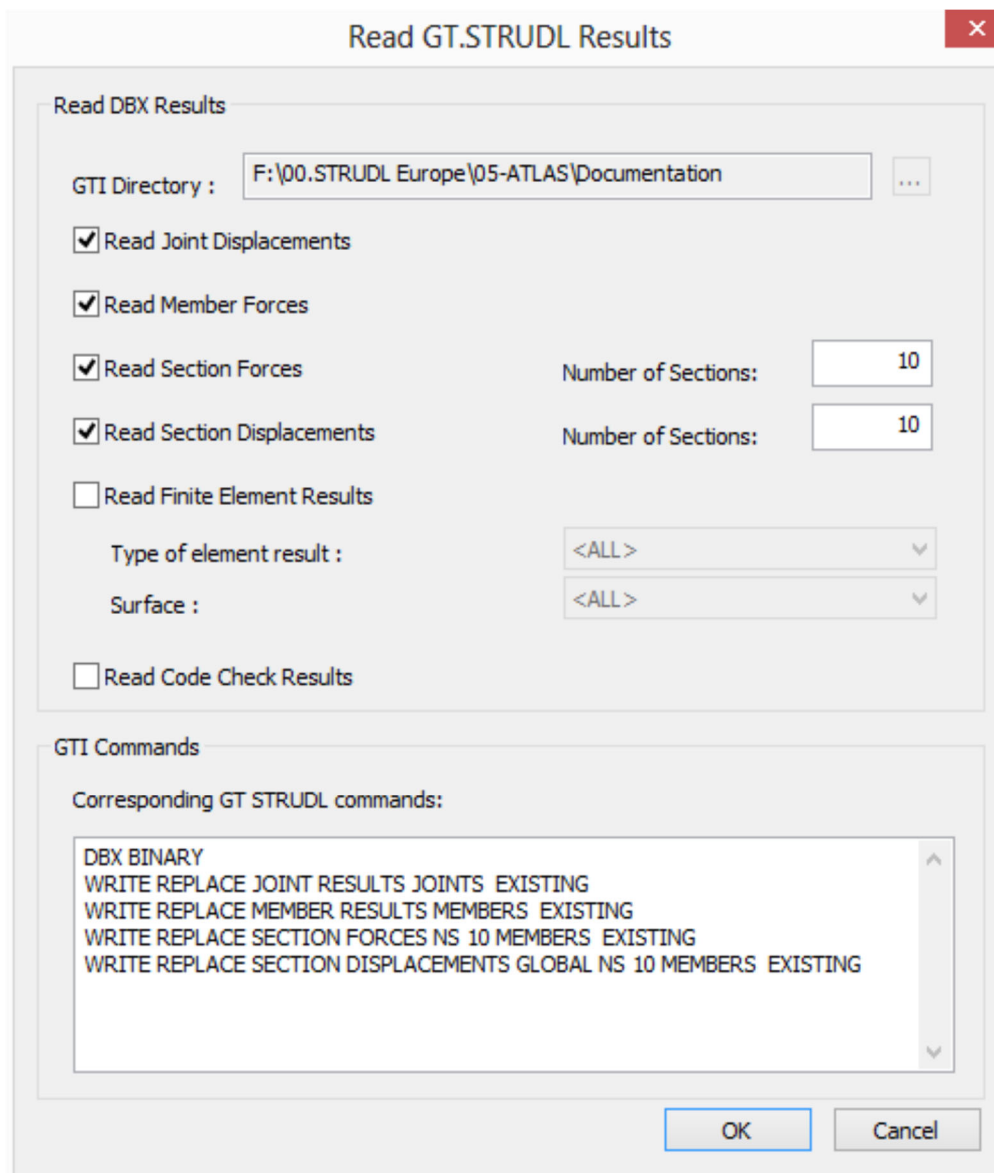
2.6.59. Execute GT STRUDL

GTSTRU DL can be launched, to process the previously created GTI, from the ribbon command  or from the menu “*GTS Modeling>Edit GT.STRU DL GTI*” or by typing `GTSExecuteGTI` at the command prompt.

2.6.60. Read Analysis Results

After performing the stiffness analysis in GT STRUDL, results can be read back to CAD Modeler, from the ribbon command  **Read Results** or from the menu “GTS Modeling>Read GTSTRUDL Results” or by typing `GTSResultsGTI` at the command prompt.

“Read GTSTRUDL Results” form appears, where you can choose to import Displacements, Member Forces, Section Forces, Section Displacements, Finite Element Results and Code Check Results. Depending on your selection a set of GTI DBX commands are created in the edit boxes shown below. If you have selected the same options in “Generate GTI” command, then the DBX commands are already included in your GTI file. Else, they should be copied and pasted into GT STRUDL main window. Do not press OK before the writing of the files in the GT STRUDL main window has completed.



The dialog box is titled "Read GT.STRUDL Results" and contains two main sections: "Read DBX Results" and "GTI Commands".

Read DBX Results

GTI Directory : ...

☒ Read Joint Displacements

☒ Read Member Forces

☒ Read Section Forces Number of Sections:

☒ Read Section Displacements Number of Sections:

☐ Read Finite Element Results

Type of element result :

Surface :

☐ Read Code Check Results

GTI Commands

Corresponding GT STRUDL commands:

DBX BINARY
WRITE REPLACE JOINT RESULTS JOINTS EXISTING
WRITE REPLACE MEMBER RESULTS MEMBERS EXISTING
WRITE REPLACE SECTION FORCES NS 10 MEMBERS EXISTING
WRITE REPLACE SECTION DISPLACEMENTS GLOBAL NS 10 MEMBERS EXISTING

OK Cancel

By pressing OK you will get the confirmation message “Results Loaded Successfully” at the command prompt. Else, you will get an error message informing you about the type of analysis results that are missing and the corresponding DBX full path file names.

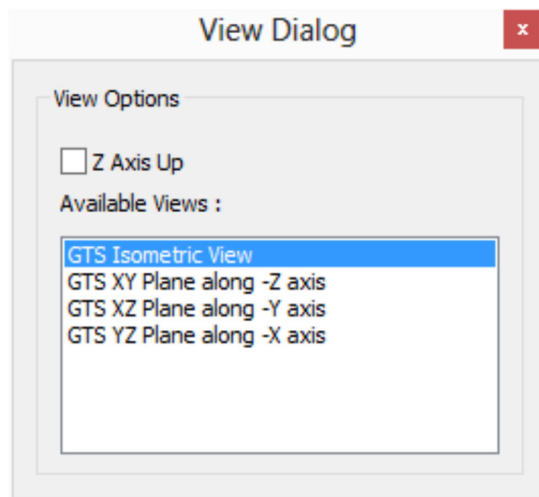
2.6.61. Import GTI

An existing GTI file can be imported it into CAD Modeler from the menu “*GTS Modeling>Import>GT.STRUDL GTI*” or by typing `GTSGTIRead` at the command prompt. Note that the GTI should be generated by the command “File>Save>Text Input File...” from GT STRUDL main menu.

2.6.62. Set Views

You can switch between different 2D or 3D views of the structure from the ribbon command  **Set View** or from the menu “*GTS Display>Set View*” or by typing `GTSSetView` at the command prompt.

It is strongly recommended to use Z as the vertical axis, so as to be able to use all built-in AutoCAD or BricsCAD functions for Views (Top, Bottom, Left, Right, Isometric, etc). However, if you use Y as the vertical axis, you can use this form to have identical 2D and 3D views, as in GTMenu.



2.6.63. 3D or Wireframe View of the Structure

You can switch between the 3D view or wireframe view of the structure.

You can view the 3D display of your model from the ribbon command  or from the menu “*GTS Display>3D Sections*” or by typing `GTSSet3D` at the command prompt. When 3D view is selected, all members appear as solid cross sections and shell elements are displayed in 3D view taking into account their thickness.

You can view the wireframe display of your model from the ribbon command  **Frame** or from the menu “*GTS Display>Frame*” or by typing `GTSSet1D` at the command prompt. When the wireframe view is selected, all members and shell edges are displayed as lines.

If some parts of the structure are hidden (i.e. using Level’s form) you can display the whole structure from the menu “*GTS Display>All Levels ON*”  **All Levels ON** or by typing `GTSSetAllVisible` at the command prompt.

2.6.64. Analytical/Physical Member View

You can turn ON or OFF the physical member view of the structure. When Physical Member view is turned ON, all analytical members belonging to a physical member are hidden and the physical members are displayed instead, as single objects. You can use copy, edit and move commands on the physical member objects and they apply to analytical members as well.

You can switch between the physical or analytical member view of your model from the ribbon command  **Analytical/Physical** or from the menu “*GTS Display>Analytical/Physical*” or by typing `GTSDisplayPhysicalMembers` at the command prompt.

2.6.65. Colors and Visible Elements

You can control the color of each member or element, and its visibility from the ribbon command  **Colors** or from the menu “*GTS Display>Colors*” or by typing `GTSColorView` at the command prompt.

Using the tab “*Sections*” in the “*Color Options*” form shown below, you can assign a different color for each cross-section profile and set its visibility to ON or OFF. By pressing “*Reset Colors*”, all colors are set to defaults. By clicking on the color value cell a pop-up dialog appears, where you can select the color that you want to be assigned to this section. If you select “BYLAYER” or 256 then the color is assigned by the color set at *GTS Display>Options*  **Options**, as explained in [2.6.66](#).

Color Options

Sections

Groups

Categories

Sections	Color	Visible
HE320B	161	☒
IPE330	191	☒
IPE120	1	☒
60x60x5	50	☒

Note: If you set a color to 256 (BYLAYER) then the color is defined by the color set at View > Options Dialog

Uncheck ALL

Check ALL

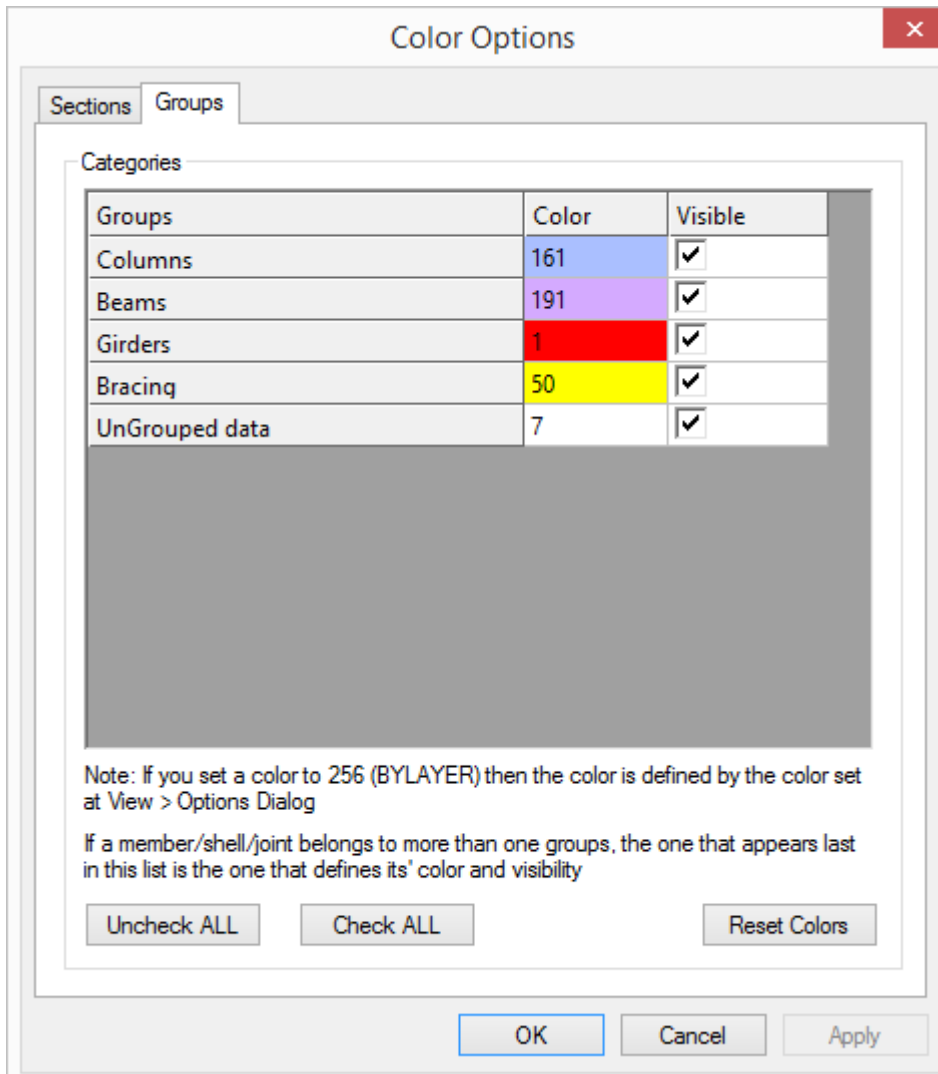
Reset Colors

OK

Cancel

Apply

Using the tab “Groups” in the “Color Options” form, you can assign a different color for each group and set its visibility to ON or OFF. Moreover, you can set a color for entities that do not belong to any group (*UnGrouped data*). For entities belonging to more than one groups, the one that appears last in the color list is the one that defines it’s color and visibility.



Note, that if the “Sections” tab is active when pressing “OK”, then the colors will be selected according to the “Sections” tab. If the “Groups” tab is active when pressing “OK”, then the colors will be selected according to the “Groups” tab.

2.6.66. Display Options

You can set the display options from the ribbon command  **Options** or from the menu “*GTS Display>Options*” or by typing `GTSDisplay` at the command prompt. Using the “*Display Options*” form shown below, you can:

- set which objects will be visible or not
- set object colors
- set which labels will be visible or not
- set font sizes for labels. NOTE: Font sizes are defined in length units, except Annotation fonts that are entered in Points.

- set type of Annotation Format: Decimal, Exponential or Generic (automatic) and the number of decimal places
- set object sizes
- set the shrink factor for finite elements and members. This option makes it is easier for you to detect members that lie along finite element edges.
- Do Not Display Thickness in 3D. If you check this option, elements will be displayed as being 2D instead of a 3D display which shows the thickness of the elements. This option may increase the display speed in very large finite element models.
- Display Members as Analytical or Physical (see [2.6.64](#))

Display Options

Visible Objects

- ☒ Joints
- ☒ Members
- ☒ 2D Elements
- ☒ 3D Elements

Visible Labels

- ☒ Joints
- ☒ Members
- ☒ 2D Elements
- ☒ 3D Elements

Label Settings - Font Sizes

Joints : 9.842520

Members : 9.842520

2D Elements : 9.842520

3D Elements : 9.842520

Annotation (pts) : 10.000000

Annotation Format: Decimal

Decimal Places : 2

Object Sizes

Joint : 1.968504

Load Arrowhead (pts): 10.00000

Display Members / Elements

Shrink Factor : 1.0

☐ Do Not Display Thickness in 3D

Members As: Analytical

Scale Factors

Concentrated Load (pts) : 72.000000

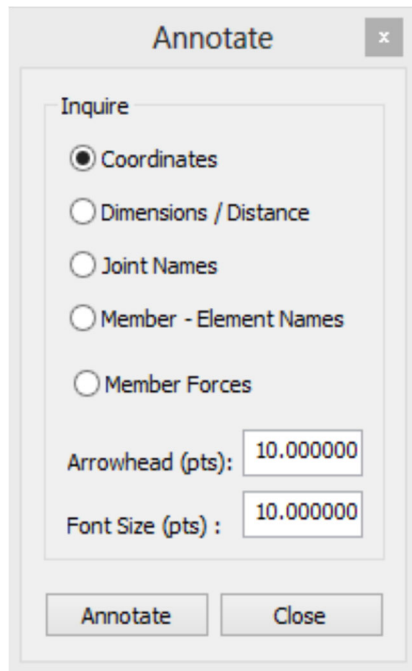
Distributed Load (pts) : 72.000000

OK Cancel

2.6.67. Annotate

You can display information related to your model from the ribbon command  **Annotate** or by typing `GTSAnnotate` at the command prompt. The “Annotate” form appears where you

choose the type information needed, press the “Annotate” button and select the corresponding entities.

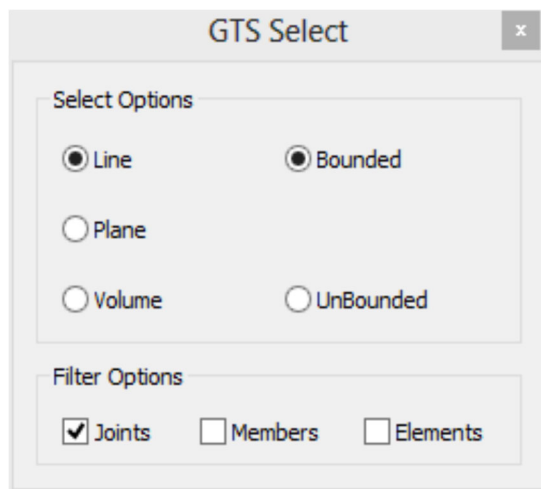


The available inquire options are:

- *Coordinates* of a specific Joint or AutoCAD/BricsCAD Point. Immediately after selecting this you have to select one Joint or Point.
- *Dimension/Distance* between two Joints or AutoCAD/BricsCAD points. Immediately after selecting this you have to select two Joints or Points.
- *Joint Names* to display the name of a specific joint. Immediately after selecting this you have to select one Joint.
- *Member or Element Names*. Immediately after selecting this you have to select one Member or Shell.
- You can also control the size of the fonts (in points) and the arrowhead.

2.6.68. Select CAD Modeler’s entities

You can use all AutoCAD’s/BricsCAD’s selection functions (window, crossing, pick, etc) to select CAD Modeler’s structural entities. In addition, there is a command to help you selecting entities, having functionality similar to GTMENU. You can access this command from the ribbon command  **Select** or by typing `GTSSelect` at the command prompt. The “GTS Select” form appears where you can set the selection options.



- Bounded Line Selection: All entities that lie on a Line
- UnBounded Line Selection: All entities that lie on a Line or its extension
- Bounded Plane Selection: All entities that lie on a Plane
- UnBounded Plane Selection: All entities that lie on a Plane or its extension
- Bounded Volume Selection: All entities that are located inside a Volume
- UnBounded Volume Selection: All entities that are located inside a

Moreover, you can choose to filter only Joint, Members and Elements during the selection.

2.6.69. Display Member Local Axes

You can view the local axes of all members from the icon  **Member Local Axes** (Ribbon GTS Display) or from the menu “*GTS Display>Member Local Axes*” or by typing `GTSDisplayLocalAxes` at the command prompt and immediately after you click at the point where you want the legend to be displayed.

In the legend, the X axis is displayed in cyan, Y axis in red and Z axis in yellow. The size of the arrow and its arrowhead is controlled by the value given at Display Options > Object Sizes > Load Arrowhead and the size of the legend fonts is controlled by the value given at Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in [2.6.66](#)).

2.6.70. Display Member Releases

You can view the member end releases of all members from the icon  **Releases** (Ribbon GTS Display) or from the menu “*GTS Display>Member Releases*” or by typing `GTSDisplayReleases` at the command prompt.

A text identifying the released degrees of freedom appears next to members having releases. No text appears for members that do not have releases. The size of the legend fonts is controlled by the value given at Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in [2.6.66](#)).

2.6.71. Display Shell Planar Axes

You can view the planar axes of all shell elements from the icon  **Shell Planar Axes** (Ribbon GTS Display) or from the menu “*GTS Display>Shell Planar Axes*” or by typing `GTSDisplayPlanarAxes` at the command prompt and immediately after you click at the point where you want the legend to be displayed.

In the legend, the X axis is displayed in cyan, Y axis in red and Z axis in yellow. The size of the arrow and its arrowhead is controlled by the value given at Display Options > Object Sizes > Load Arrowhead and the size of the legend fonts is controlled by the value given at Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in [2.6.66](#)).

2.6.72. Display Joint Supports

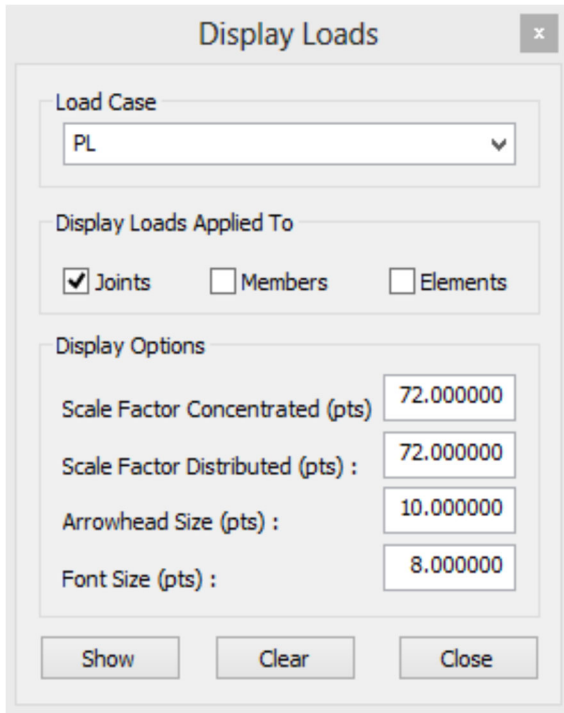
You can view the support status of each joint from the icon  **Joint Supports** (Ribbon GTS Display) or from the menu “*GTS Display>Joint Supports*” or by typing `GTSDisplaySupports` at the command prompt.

A red arrow is displayed for the translational restrained degrees of freedom and a yellow arrow is displayed for the rotational restrained degrees of freedom. The size of the arrow and its

arrowhead is controlled by the value given at Display Options > Object Sizes > Load Arrowhead and the size of the legend fonts is controlled by the value given at Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in [2.6.66](#)).

2.6.73. Display Joint Loads

You can view the joint loads applied in the structure from the icon  Joint (Ribbon GTS Display) or from the menu “GTS Display>Joint Loads” or by typing `GTSDisplayJointLoads` at the command prompt.



The “Display Loads” form appears where you can select the desired *Load Case*, the *Scale Factor* for Joint Loads, *Arrowhead Size* and the *Font Size*. The “Show” button displays the load arrows, and the “Clear” button erases them.

2.6.74. Display Member Loads

You can view the member loads applied in the structure from the icon  Member (Ribbon GTS Display) or from the menu “GTS Display>Member Loads” or by typing `GTSDisplayMemberLoads` at the command prompt.

The “Display Loads” form appears where you can select the desired Load Case, the Scale factor for Concentrated or Distributed Member Loads, Arrowhead Size and the *Font Size*. The “Show” button displays the load arrow, and the “Clear” button erases them.

2.6.75. Display Shell Loads

You can view the finite element loads applied in the structure from the icon  (Ribbon GTS Display) or from the menu “GTS Display>Shell Loads” or by typing `GTSDisplayElementLoads` at the command prompt.

The “Display Loads” form appears where you can select the desired Load Case, the Scale factor for Concentrated or Distributed Member Loads, Arrowhead Size and the *Font Size*. The “Show” button displays the load arrow, and the “Clear” button erases them.

2.6.76. Display Area Loads

You can view the area loads applied in the structure from the icon  **Area** (Ribbon GTS Display) or from the menu “*GTS Display>Area Loads*” or by typing `GTSDisplayAreaLoads` at the command prompt. Loaded areas appear in yellow solid hatch. If you want to display only one area load, you can use the area load command (2.6.48) to bring up the area load dialog, select the specific area load and click “Display >>”.

2.6.77. Display Deformed Structure

You can view the deformed shape of the structure from the icon  **Deformed** (Ribbon GTS Display) or from the menu “*GTS Display>Deformed Structure*” or by typing `GTSDisplayJointDisplacements` at the command prompt. You must then immediately select the load case, press ENTER, and then give the desired scale factor.

You can switch back to original view from the icon  **Undeformed** (Ribbon GTS Display) or from the menu “*GTS Display>Undeformed Structure*” or by typing `GTSResetJointDisplacements` at the command prompt.

Note that Deformed Structure can be displayed in both 3D and Wireframe views of your model and that you can also switch between levels using the Levels Form or the Upper Level, Lower Level icons.

2.6.78. Annotate Joint Displacements

At the deformed state, you can annotate joint displacements of the structure from the icon



Annotate Displacements

(Ribbon GTS Display) or by typing

`GTSAnnotateJointDisplacements` at the command prompt. This command is valid when the deformed shape of the structure is visible, where you select a joint and then a point in screen for annotation position.

2.6.79. Display Displacements

You can view the displacements of the model (including a member's deformation between joints) from the icon  **Displacements** (Ribbon GTS Display) or from the menu "*GTS Display>Displacements*" or by typing `GTSDisplaySectionDisplacements` at the command prompt.

The “Displacement” form appears where you can select:

- The desired Load Case or Combination
- The Scale factor
- The *Font Size* (in pts) for Annotations and the Annotation Format.
- Choose to display the model or Hide it, so that the deformed shape is clearer.
- The “*Display >>*” button displays the deformed shape for the visible members and elements. If there are any hidden members or elements their deformed shapes are not displayed.
- The “*Annotate >*” button allows you to annotate any value on the deformed shape by first clicking on the deformed shape curve and then at the position that annotation will be placed.
- The “*Legend >*” button allows you to place a legend on screen, having information about the load case.
- The “*Generate Animation Frames*” button creates the animation frames that can then be played in a loop by using the “*Animate>>*” button.

2.6.80. Display Member Diagrams

You can view the force and moment diagrams from the icon  **Diagrams** (Ribbon GTS Display) or from the menu “*GTS Display>Member Diagrams*” or by typing `GTSDisplayMemberForces` at the command prompt.

The “Member Diagrams” form appears where you can select:

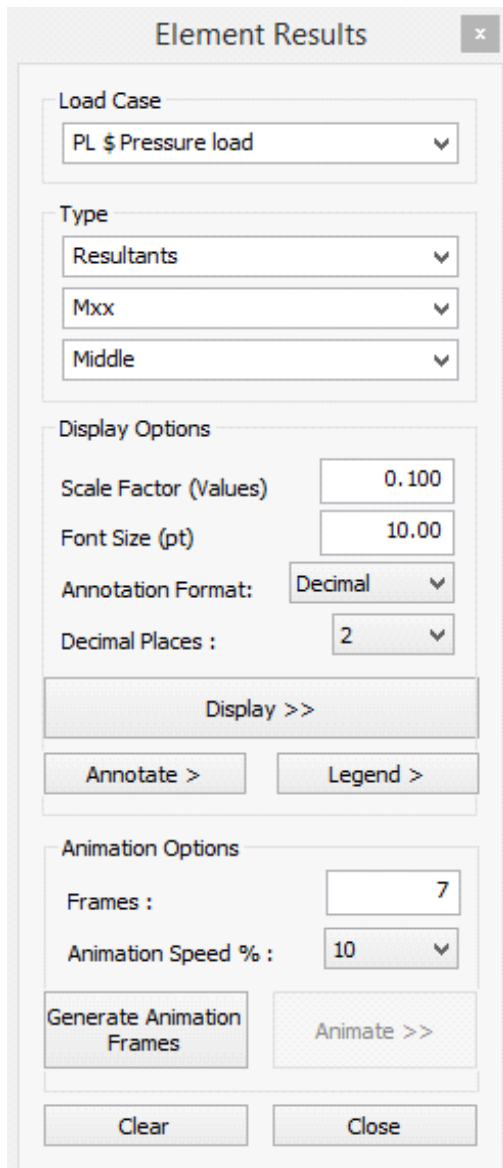
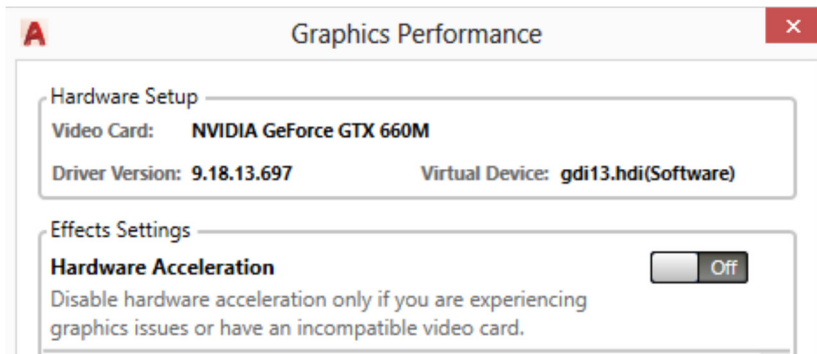
- The desired Load Case or Combination
- The Envelope option and the load cases that form the envelope.
- The Forces or Moments to be displayed (FX, FY, FZ, MX, MY, MZ)
- The Scale factor
- The *Font Size* (in pts) for Annotations and the Annotation Format.
- Automatically *Label Maximum and Minimum* values for each diagram
- Choose the direction of the diagrams by switching the *Positive Sign*.
- The “*Display >>*” button creates the diagram for the visible members. If there are any hidden members their diagrams are not displayed.
- The “*Annotate >*” button allows you to annotate any value of the diagram by first clicking on the member diagram curve and then at the position that annotation will be placed.

- The “*Legend >*” button allows you to place a legend on the screen, having information about the load case and member diagram.

2.6.81. Display Finite Element Results

You can view the finite element results from the icon  **Elements** (Ribbon GTS Display) or from the menu “*GTS Display>Element Results*” or by typing `GTSDisplayElementResults` at the command prompt.

NOTE: Hardware graphics acceleration may cause AutoCAD to incorrectly display the colors of the contour. In such a case it is recommended that you turn OFF Hardware Acceleration, during displaying the stress contours, by typing the command `GRAPHICSCONFIG`. You can turn it back ON afterwards.



The “Element Results” form appears where you can select:

- The desired Load Case
- The desired item to be displayed: Stress, Strain, Resultants, Principal Stress, Principal Strain, Principal Membrane Resultant, Principal Bending Resultant, Von Misses
- The component of the desired item to be displayed, i.e. Sxx, Syy, Szz
- The Location: Top, Middle or Bottom surface of the element which is defined by the local or planar z axes of the shell element. The top surface is in the positive z direction.
- The “Annotate >” button allows you to annotate any value of the diagram by first clicking on a joint and then at the position that annotation will be placed.
- The “Display >>” button creates the contour and a popup legend showing the limits of each color.
- The “Annotate >” button allows you to annotate any value of the diagram by first clicking on the member diagram curve and then at the position that annotation will be placed
- The “Generate Animation Frames” button creates the animation frames (contours and displacements) that can then be played in a loop by using the “Animate>>” button.

2.6.82. Display Finite Element Selection Results

You can view the finite element results of selected elements from the icon  Selection (Ribbon GTS Display) or from the menu “*GTS Display>Element Results Selection*” or by typing `GTSDisplayElementResultsSel` at the command prompt.

This command is similar to the “Display Finite Element Results” of the previous paragraph. The only difference is that you have to give a selection of members for the contours. This is useful in cases where you want to examine only one surface of the structure, so display the limits of the specific area. Or, if you do not want to take into account stresses from elements not belonging to a specific plane.

NOTE: An “Execute GTI” command, including stiffness analysis, must be given prior to this command.

2.6.83. Display Member Code Check Results

You can view the pass/fail result of a Steel Code check or design from the icon  Code Check (Ribbon GTS Display) or from the menu “*GTS Display>Member Code Check Results*” or by typing `GTSColorCodeCheck` at the command prompt. You must then select the members to be displayed (or “ALL” for all of them).

The “Code Check Results” form appears where you can select:

- The text to be displayed for each member, giving additional information such as stress ratios, controlling provisions and KL/r ratios.
- The *Font Size* (in pts)
- Limits for the Values to be displayed (in example All of them, or Greater Than a given value, or Less Than a given value).
- The “*Display >>*” button regenerates the view and members which passed the code check will appear in blue, those that fail the code check will appear in red and those that were not included in the code check will appear in white
- The “*Legend >*” button allows you to place a legend on screen, having information about the colors used.

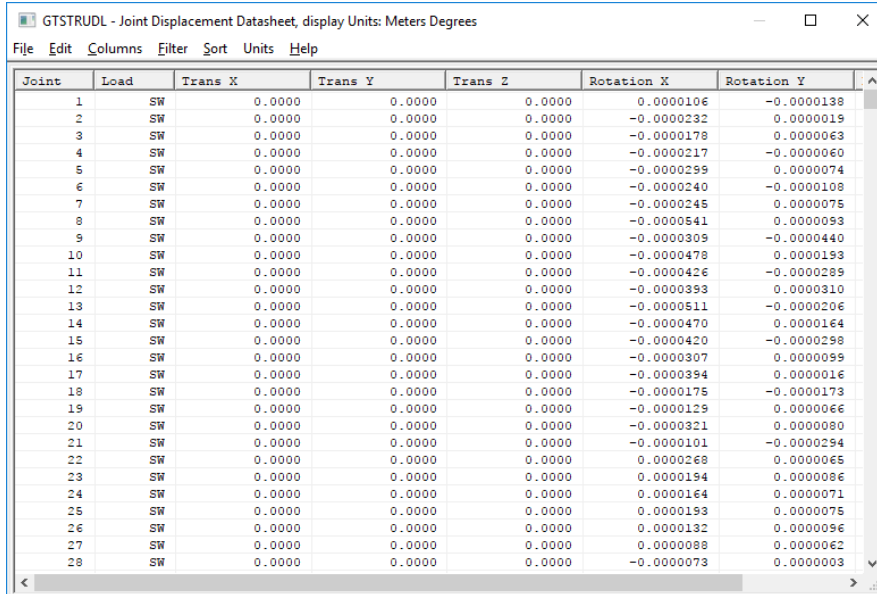
When you press the clear button, member colors remain blue and red for your convenience. If you want to change them go to GTS Display > Colors.

2.6.84. Results Datasheets

In addition to the graphical display of results, you can also view the results in datasheets from item “Results Datasheets” of the menu or ribbon tab “GTS Display”. In the table below, you can find all the available results that can be viewed in datasheets and the corresponding command.

Description	Menu Item	Command
Displacements	<i>GTS Display>Results Datasheets> Displacements</i>	GTSDataSheetJointDisp
Member Forces	<i>GTS Display>Results Datasheets> Member Forces</i>	GTSDataSheetMemberForces
Section Forces	<i>GTS Display>Results Datasheets> Section Forces</i>	GTSDataSheetMemberForces
Reactions	<i>GTS Display>Results Datasheets> Reactions</i>	GTSDataSheetReactions
Stresses	<i>GTS Display>Results Datasheets> Stresses</i>	GTSDataSheetStresses
Code Check	<i>GTS Display>Results Datasheets> Code Check</i>	GTSDataSheetCodeCheck

For example, click on the icon  **Displacements** (Ribbon GTS Display) or from the menu “GTS Display>Results Datasheets>Displacements” or by typing `GTSDatasheetsJointDisp` and the “GTSTRUDL – Joint Displacement Datasheet” dialog appears where you can filter, sort, write results to text file or change results units as shown in figure below:



Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y
1	SW	0.0000	0.0000	0.0000	0.0000106	-0.0000138
2	SW	0.0000	0.0000	0.0000	-0.0000232	0.0000019
3	SW	0.0000	0.0000	0.0000	-0.0000178	0.0000063
4	SW	0.0000	0.0000	0.0000	-0.0000217	-0.0000060
5	SW	0.0000	0.0000	0.0000	-0.0000299	0.0000074
6	SW	0.0000	0.0000	0.0000	-0.0000240	-0.0000108
7	SW	0.0000	0.0000	0.0000	-0.0000245	0.0000075
8	SW	0.0000	0.0000	0.0000	-0.0000541	0.0000093
9	SW	0.0000	0.0000	0.0000	-0.0000305	-0.0000440
10	SW	0.0000	0.0000	0.0000	-0.0000478	0.0000193
11	SW	0.0000	0.0000	0.0000	-0.0000426	-0.0000289
12	SW	0.0000	0.0000	0.0000	-0.0000393	0.0000310
13	SW	0.0000	0.0000	0.0000	-0.0000511	-0.0000206
14	SW	0.0000	0.0000	0.0000	-0.0000470	0.0000164
15	SW	0.0000	0.0000	0.0000	-0.0000420	-0.0000298
16	SW	0.0000	0.0000	0.0000	-0.0000307	0.0000099
17	SW	0.0000	0.0000	0.0000	-0.0000394	0.0000016
18	SW	0.0000	0.0000	0.0000	-0.0000175	-0.0000173
19	SW	0.0000	0.0000	0.0000	-0.0000129	0.0000066
20	SW	0.0000	0.0000	0.0000	-0.0000321	0.0000080
21	SW	0.0000	0.0000	0.0000	-0.0000101	-0.0000294
22	SW	0.0000	0.0000	0.0000	0.0000268	0.0000065
23	SW	0.0000	0.0000	0.0000	0.0000194	0.0000086
24	SW	0.0000	0.0000	0.0000	0.0000164	0.0000071
25	SW	0.0000	0.0000	0.0000	0.0000193	0.0000075
26	SW	0.0000	0.0000	0.0000	0.0000132	0.0000096
27	SW	0.0000	0.0000	0.0000	0.0000089	0.0000062
28	SW	0.0000	0.0000	0.0000	-0.0000073	0.0000003

2.6.85. Report Builder



Report
Builder

You can generate your reports by calling Report Builder from the icon  (Ribbon GTS Display) or by typing `GTSReportBuilder` at the command prompt. GT STRUDL Report Builder is self-contained software, which allows you to generate reports graphically from DBX GT STRUDL database files on the top of GT STRUDL software. CAD Modeler automatically generates the necessary DBX files and launches Report Builder. You can find more information on how creating your report in the GT STRUDL® Report Builder Getting Started Guide.

2.6.86. Clear Results Layer

You can clear the displayed output (Display Model, Display Loads, Display Results), hide the Legend form of the contours and return to model from the icon  **Clear** (Ribbon GTS Display) or from the menu “GTS Display>Clear Results Layer” or by typing `GTSDisplayResultsClear` at the command prompt. This command should be given after any of the previous “display” commands.

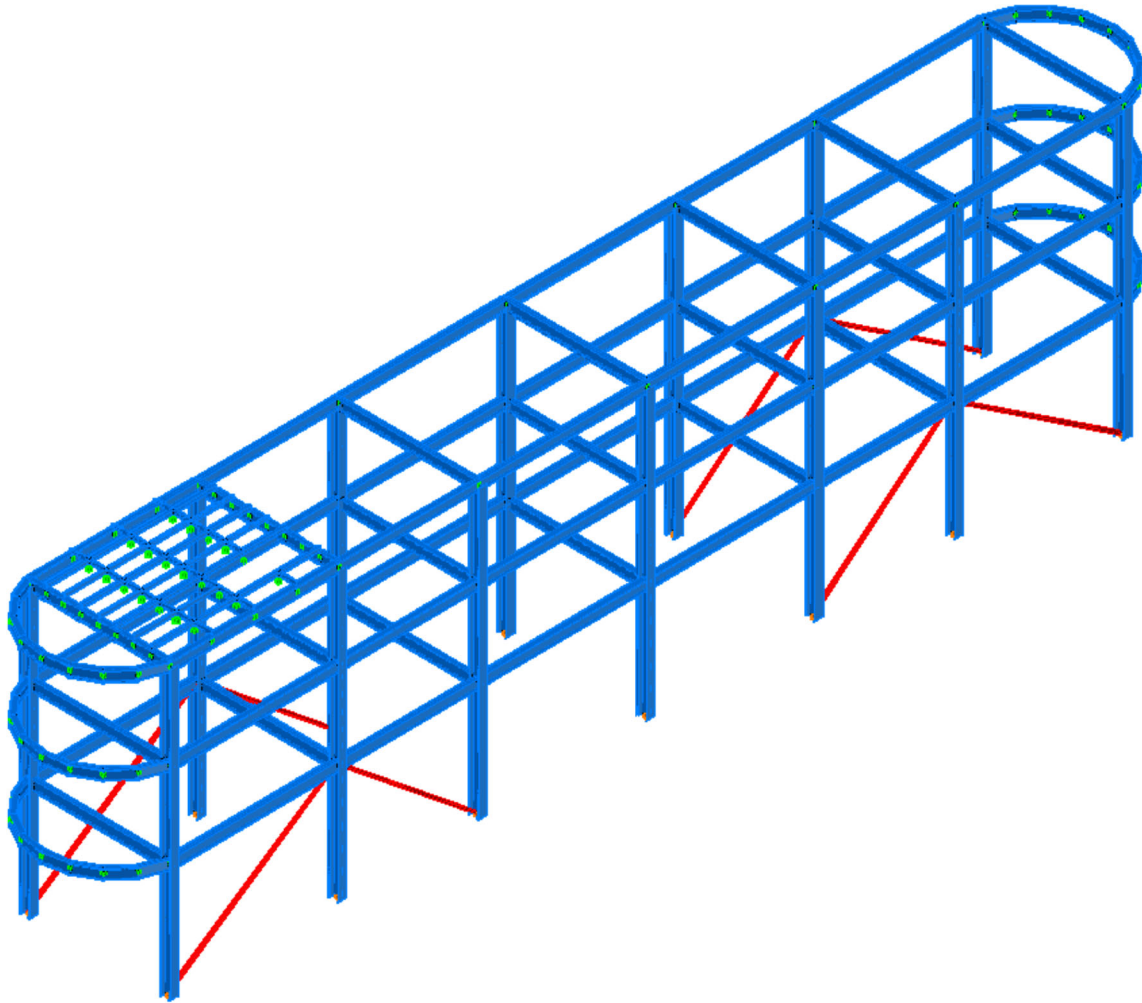
2.6.87. Version

The current version of CAD Modeler can be displayed by selecting from the menu “GTS Display>Version” or by typing `GTSVersion` at the command prompt. The current version will be displayed at the command line: The current version of CAD Modeler is
xxxxx

3. Tutorial Example #1

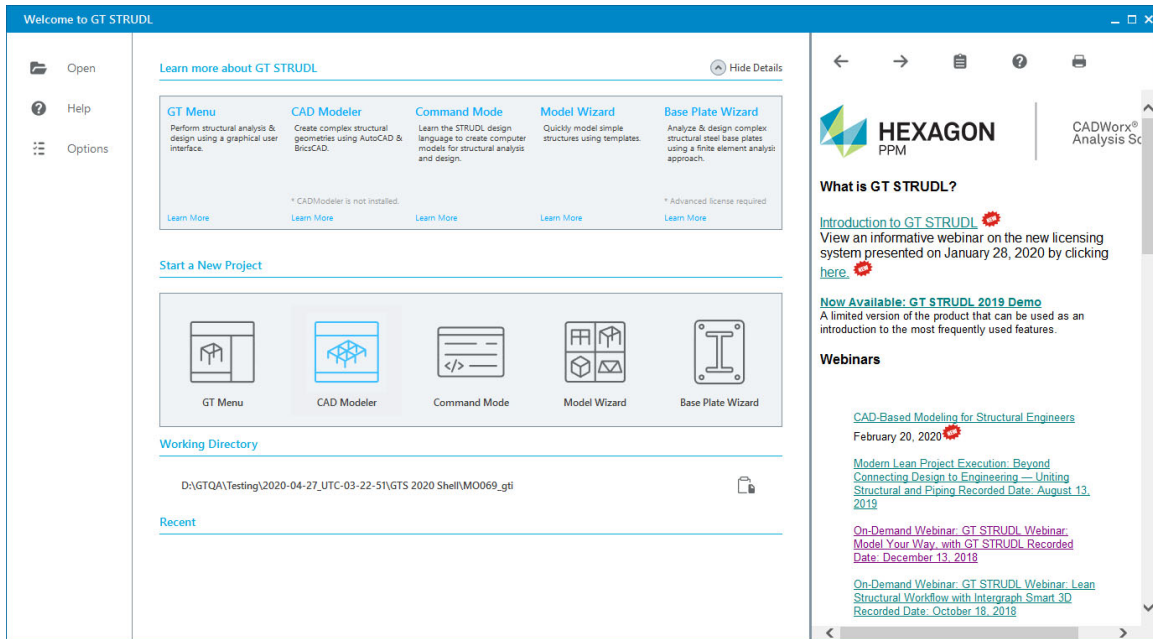
3.1. Introduction

The modeling of a three story building using CAD Modeler shown below is demonstrated in a step-by-step process.

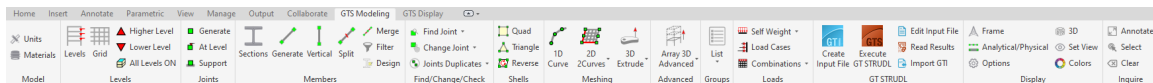


3.2. Open CAD Modeler and start working

Step #1. Launch GT STRUDL by selecting the icon “CAD Modeler” in the Welcome to GT STRUDL dialog shown below. The version of AutoCAD/BricsCAD selected during the installation will be automatically launched, together with CAD Modeler’s menus and ribbons.



Step #2. Make sure that CAD Modeler's ribbons and menus are visible.



If AutoCAD's/BricsCAD's menu is not visible, type `MENUBAR` at AutoCAD's/BricsCAD's command prompt, then 1 and press `<ENTER>`.

If AutoCAD's/BricsCAD's ribbon area is not visible, type `RIBBON` and press `<ENTER>`.

Ribbon commands will be used in this tutorial example. However, since all ribbon commands appear in the menu area, you can use the menu bar as well.

3.3. Define the basic geometry of the model

Step #3. Define the correct Units by pressing the icon  **Units** and select *Meters (m)* and *KiloNewtons* in the *Units Form*.

Units

Length

☐ Inches (in)

☐ Feet (ft)

☒ Meters (m)

☐ Centimeters (cm)

☐ Millimeters (mm)

Force

☐ Pounds (lbs)

☐ Kips

☐ Tons

☐ Kilograms

☐ Metric Tons

☐ Newtons

☒ KiloNewtons

☐ MegaNewtons

Angles

☒ Degrees

☐ Radians

☐ Cycles

Temperature

☐ Fahrenheit

☒ Centigrade

Time

☒ Seconds

☐ Minutes

☐ Hours

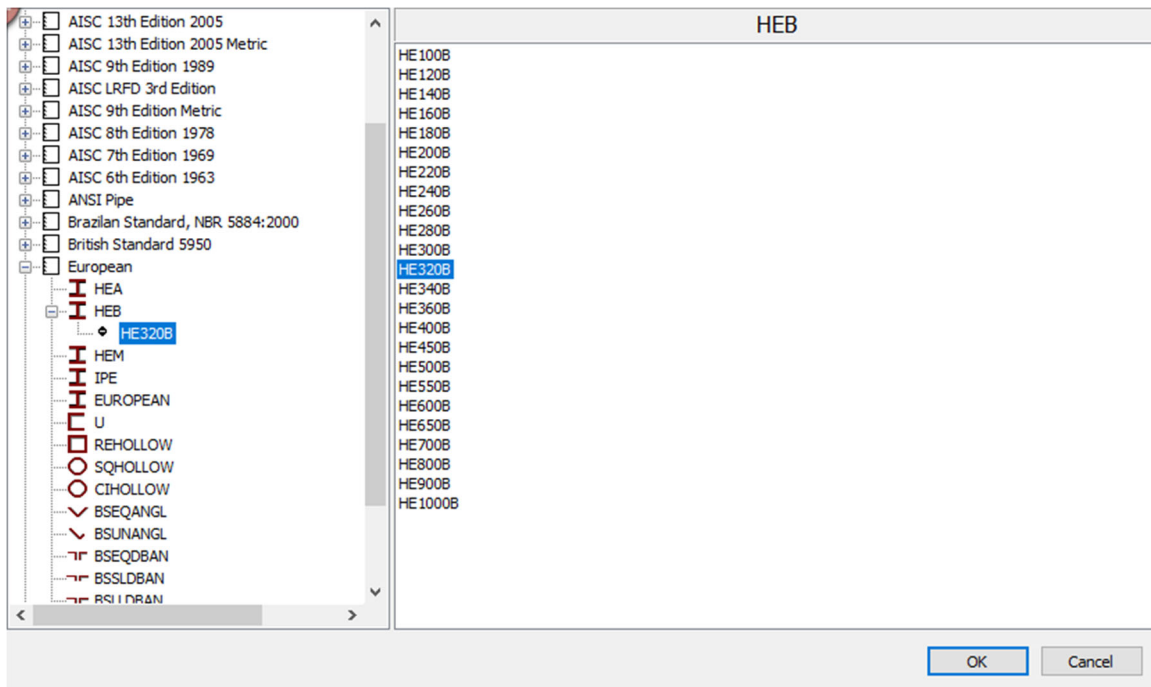
☒ Scale non-structural CAD entities
(grids, structural lines, curves etc)

OK Cancel

Step #4. Enter the cross-section profiles that will be used at the model by pressing the icon



Sections. Click on the *European* list and then on the *HEB* table of profiles. Select the profile *HE320B* that will be used for the columns, by double clicking on it.



→ European
 HEA
 HEB
 ● HE320B
 HEM
 IPE
 ● IPE330
 ● IPE120
 EUROPEAN
 U
 REHOLLOW
 SQHOLLOW
 CIHOLLOW
 BSEQANGL
 ● 60x60x5
 BSUNANGL
 BSEQDBAN
 BSSLDBAN
 BSLLDBAN

The profile is added to the project and it appears in the left list-box having a black dot in front of it.

Using the same procedure, add 3 additional profiles: *IPE330*, for beams, *IPE120* for the girders from table *IPE* and *60x60x5* for the bracing from the table *BSEQANGL*. Press **OK** to close the form.

Note: You can add additional profiles at any time by following this procedure and also view the full list of profiles used in your model and add more profiles if needed.



Step #5. Define the 3 levels of the model by pressing the icon **Levels** . Press the *Add Level* button 3 times to add 3 levels to your model. Modify the height of the 1st level by selecting the *Height* cell of the 1st Level and entering 4.

Note: Some Edit Boxes appear in yellow background and green fonts, like the one at the Level Heights. You can use mixed units in the yellow edit boxes. For more information about Mixed Units and the valid syntax please read GT STRUDL GT Menu Guide

Make sure that *Z Vertical Axis* option is checked and press **OK** to close the form.

Levels	Height	Elevation	Visible
1	4.000000	4.000000	☒
2	3.000000	7.000000	☒
3	3.000000	10.000000	☒

Options

Add Level

Delete Level

Detect Levels Automatically

Merge Levels

Base Elevation: 0.0000

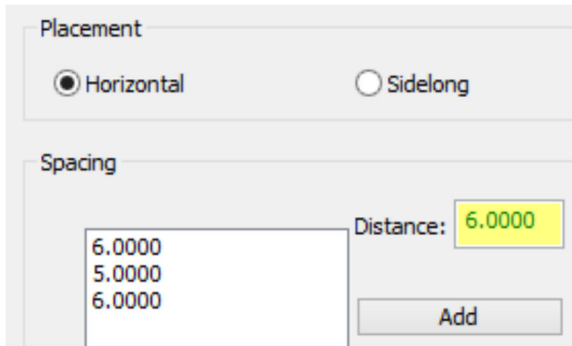
☒ Z Vertical Axis (else Y)

☐ Update Levels for All Entities

OK **Cancel**



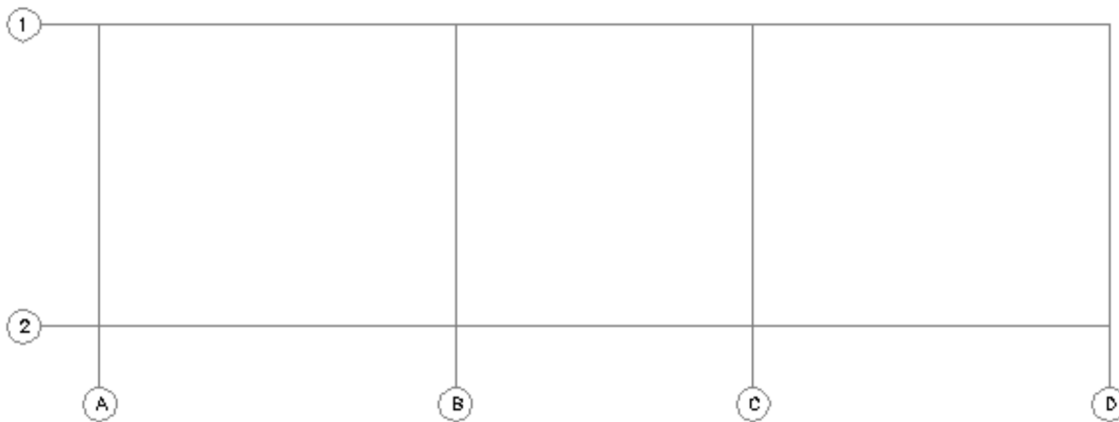
Step #6. Enter a Grid that will help you enter the columns quickly by clicking on the icon **Grid** . The grid is going to have 3 spaces (6m, 5m, 6m) in the horizontal direction (X) and 1 space (5m) in the sidelong direction (Y). Enter 6 in the *Distance* text box and press the *Add* button. Repeat by entering 5 and *Add*, and 6 and *Add*.



Then, click on *Sidelong* in the *Placement* area to enter the spacing in the Y direction, and enter 5 and *Add*.

By pressing OK you are prompted (command prompt) to enter the Insert Point for the grid. Type 0, 0, 0 and press <ENTER>.

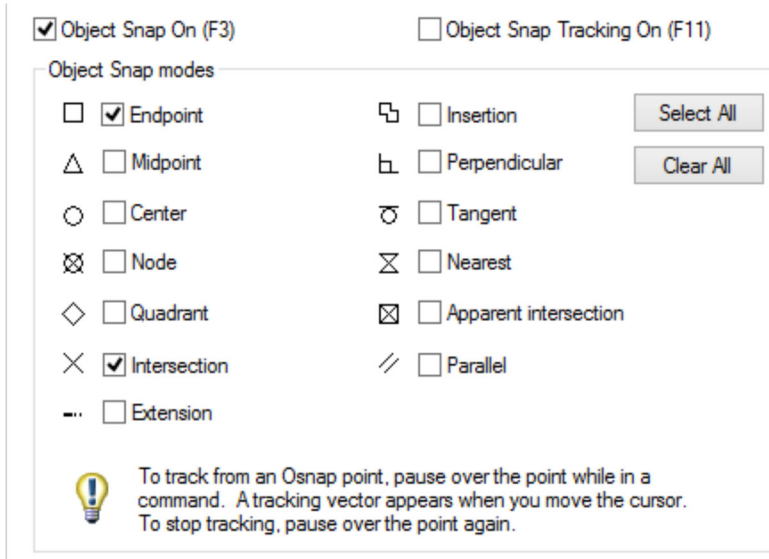
The grid is created, having its upper left corner A-1 at the point 0,0,0.



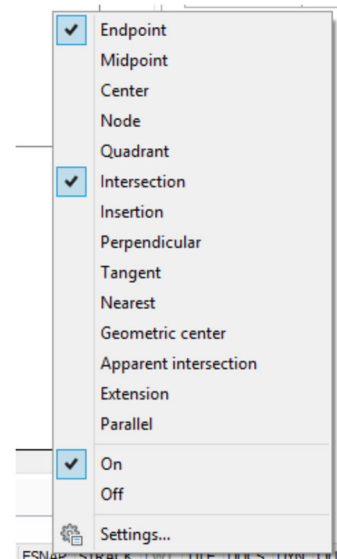
You will be able to view the grid by pressing the Top Icon in AutoCAD's View Cube or BricsCAD's Chair Icon, or preferably by typing Z (for Zoom), E (for Extents) and press <ENTER>.



Note: In order to be able to snap at the intersection of the grid lines, while placing columns, make sure that the AutoCAD's/BricsCAD's Object Snap is ON, and the Intersection mode is enabled. Type OSNAP in both AutoCAD and BricsCAD to set the various snap settings. Shown below is the ObjectSnap tab in AutoCAD's Drafting settings dialog.



AutoCAD's Snap Setting

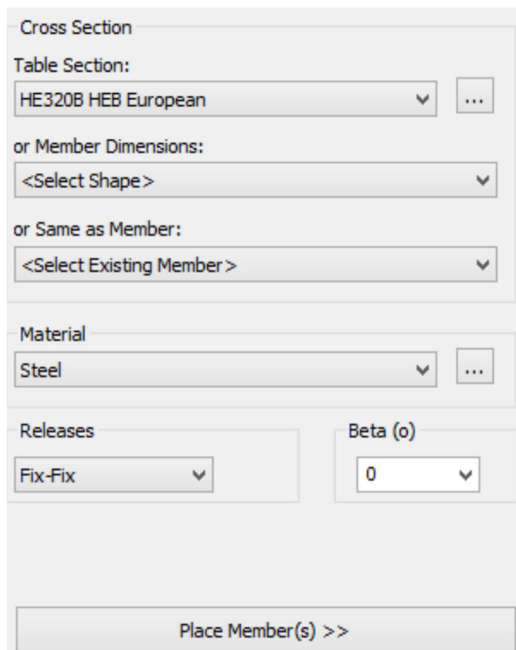


BricsCAD's Snap Settings

3.4. Create the 1st floor



Step #7. Start entering the columns by clicking on the icon **Vertical**. The dialog *Place Member* appears that helps you to quickly select properties for the members that are going to be entered.



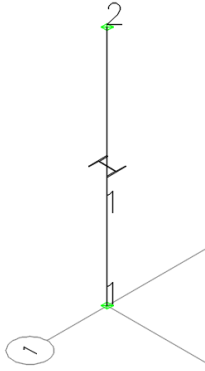
Select *HEB320B* as the cross section for the columns. Make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *0*. There is no need to close this dialog manually.

Press the “Place Member(s) >>” button.

Click at the intersection point between line A and line 1 (point A-1) and the column will be placed at this position.

Repeat the same procedure by clicking at the points B-1, C-1, D-1, A-2, B-2, C-2 and D-2.

When you are done, press ESC to exit the Vertical column command. The Place Member form is automatically hidden.



Note: Each time you create a member the orientation of the cross section will appear in the middle of the element, unless you clear it with command “Clear” (see 2.6.86).

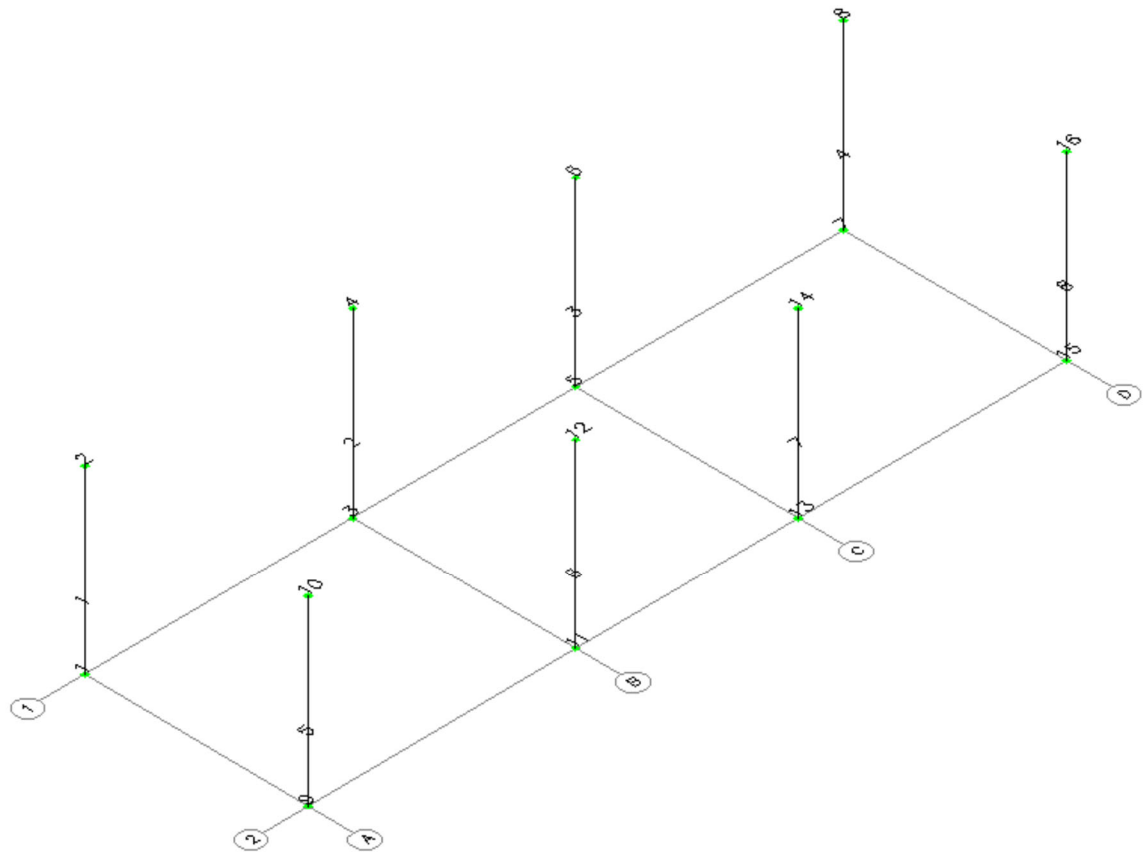
Step #8. You can easily change to an isometric view of the structure by pressing the small house icon in AutoCAD’s View Cube or the small arrow “Top Front Left” of the following image in BricsCAD’s View Chair. As you can see in the isometric view below, column members 1 to 8 were created together with joints 1 to 16 at their ends. Each column is 4.00m long, as defined in Level Properties (height of the first floor).



AutoCAD’s View Cube for Isometric View



BricsCAD’s View Cube for Isometric View





Step #9. Start entering the beams, along X axis, by clicking on the icon **Generate** . The *Place Member* form appears.

Cross Section

Table Section:
IPE330 IPE European

or Member Dimensions:
<Select Shape>

or Same as Member:
<Select Existing Member>

Material
Steel

Releases
Fix-Fix

Beta (o)
90

☒ Split Intersecting Members ☐ Physical Member
☐ Split Ending Members

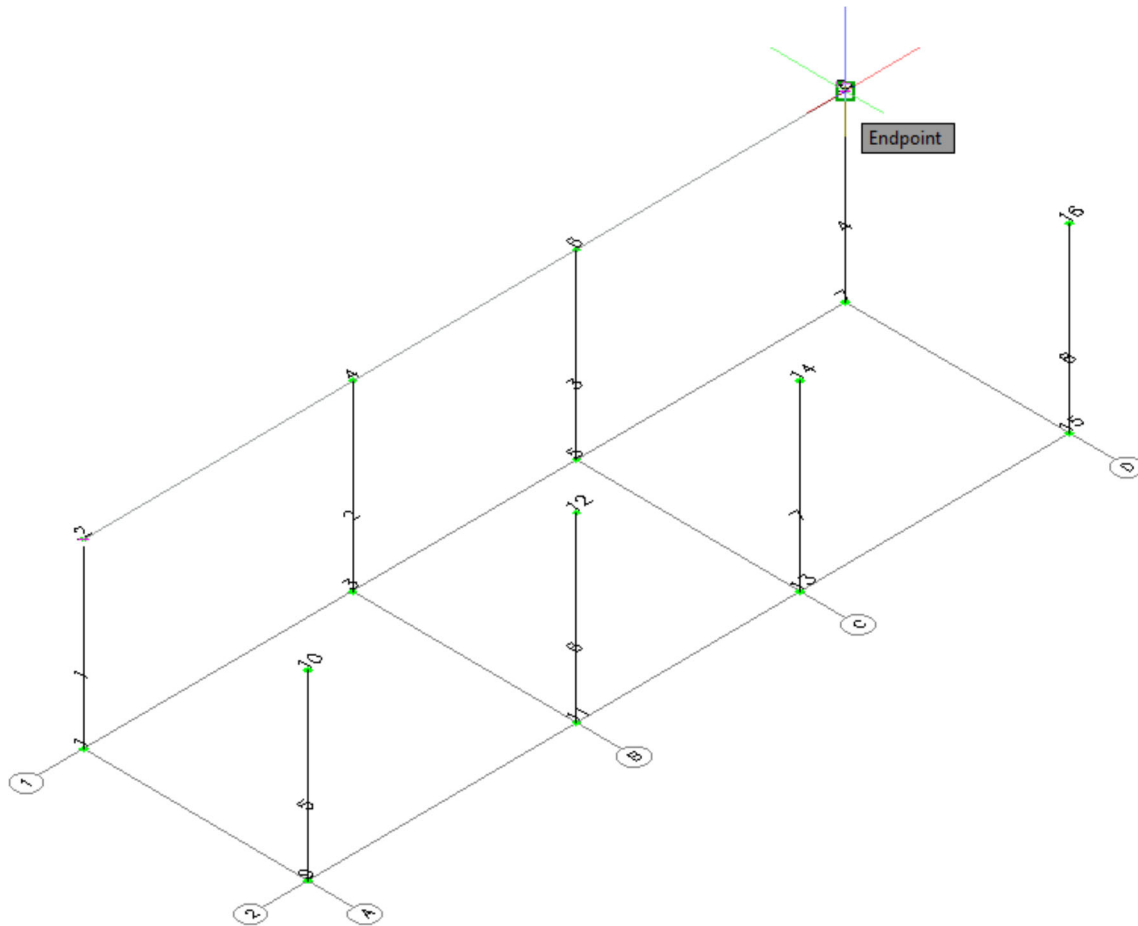
Place Member(s) >>

Select *IPE330* as the cross section and make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *90*. You have to set the Beta angle equal to 90 degrees in order to orientate the local Y axis of the IPE cross section along the Z global axis. Moreover, make sure that the option *Split Intersecting Members* is checked and then uncheck Split Ending Members.

Press “Place Member(s) >>” button.

Starting Point (x,y,z) message appears, asking you to enter the coordinates or click on a specific point on screen. Click on Joint 2 at the top joint of column 1 at position A-1.

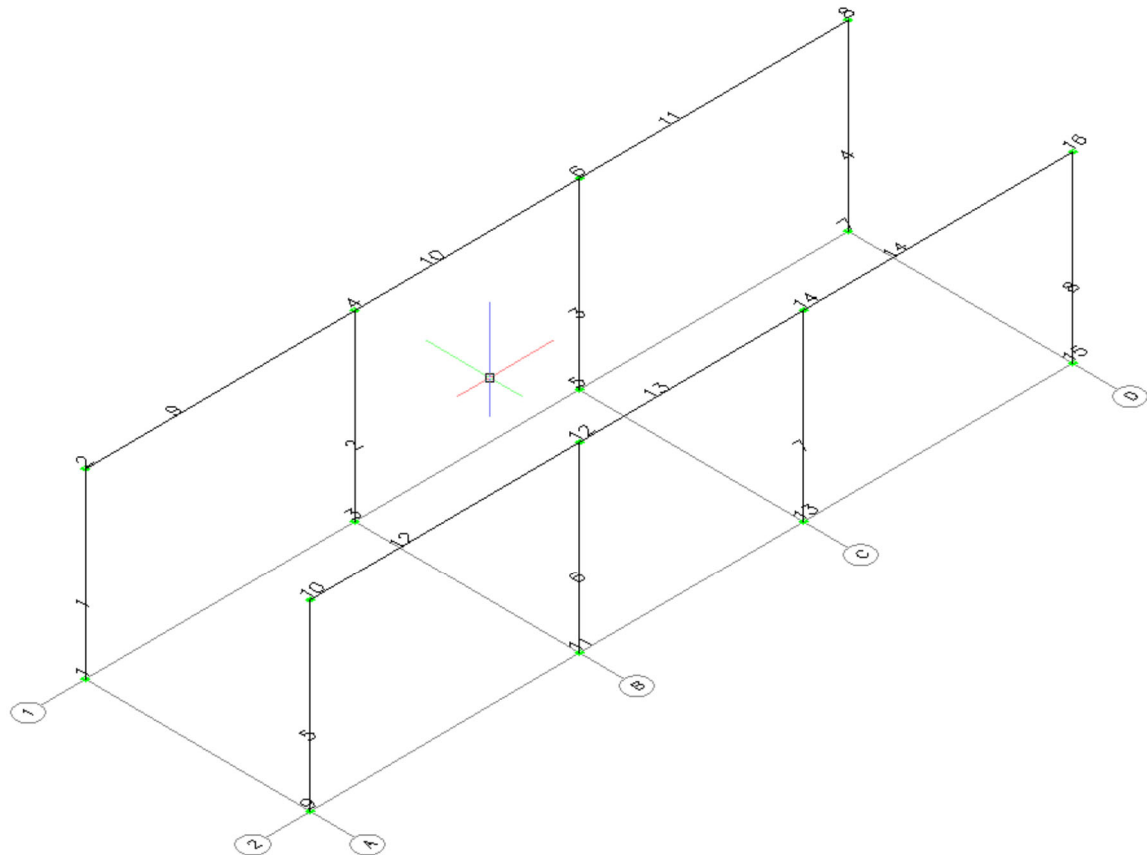
Then you have to define the *Ending Point (x,y,z)* so click on joint 8, as shown in the picture below.



Members 9, 10 and 11 will be created.

All three beams along X axis were generated with only two clicks of the mouse: at joints 2 and 8. The beam from joint 2 to joint 8, was split into three parts, between joints 2, 4, 6 and 8, since joints 4 and 6 (columns at positions B-1 and C-1) intersect this member.

Since the command is still active, you are prompted to enter the Starting Point (x,y,z), repeat the same procedure by clicking on joint 10 (top of column at position A-2), and then click at joint 16 (top of column at position D-2). Members 12, 13 and 14 will be created.



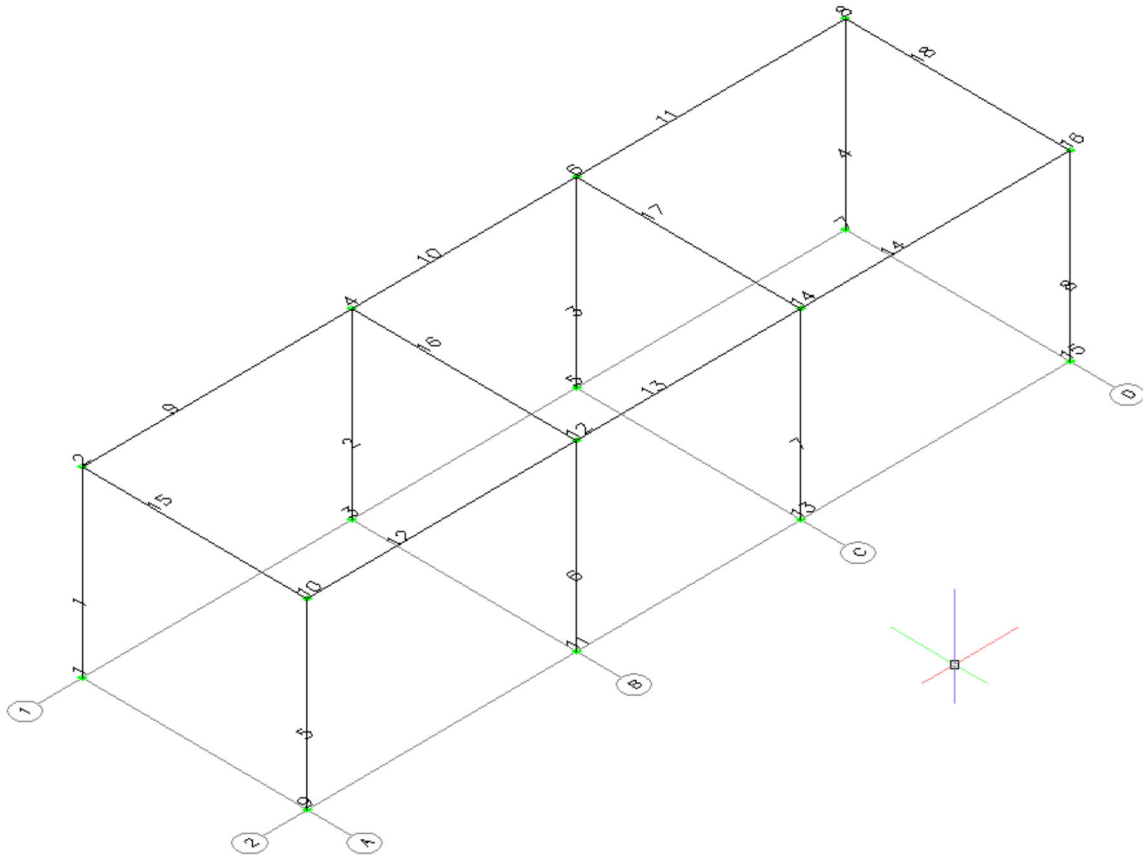
Step #10. Enter the beams, along Y axis. The command Generate Beams should be still active,



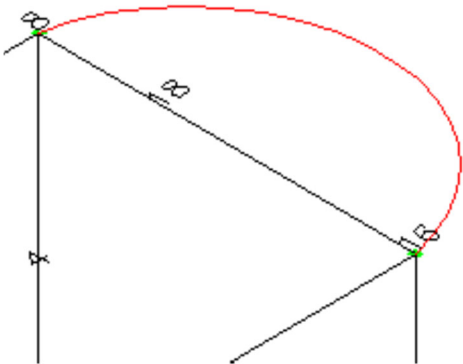
else you can call it again by clicking again on the icon **Generate**. Keep the same settings at the Place Member Form, as in the previous step, regarding the cross section and Beta angle, but do NOT click on *Split Intersecting Members*. Press the "Place Member(s) >>" button.

The prompt message *Starting Point (x,y,z)* appears, asking you to enter the coordinates or click on a specific point on screen. Click on the Joint 2, that is the top of column at position A-1. In order to define the *Ending Point (x,y,z)* click at joint 10 (top of column at position A-2). Member 15 is generated.

Repeat the same procedure by clicking on the joints 4 and 12 to generate member 16, joints 6 and 14 to generate member 17 and joints 8 and 16 to generate member 18. Then, press ESC to terminate the command.



Step #11. Create an arc on the right side of the structure:

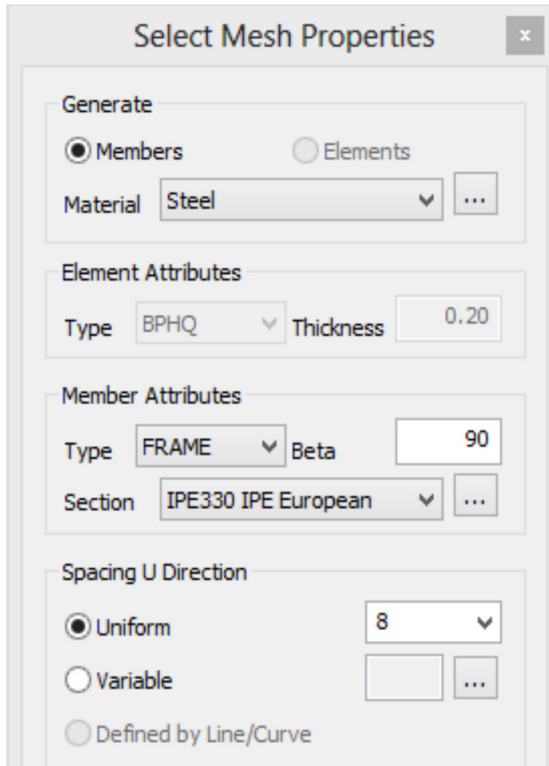


Type ARC and

- in order to *Specify start point of arc or [Center]:* click on joint 8,
- to *Specify second point of arc or [Center/End]:* type @2,-2 and press <ENTER> and
- to define the end point of arc: click on joint 16.



Step #12. Generate Members along the Arc: Click on the icon **1D Curve** and when the prompt message *Select Curve (Line or Arc)* appears, click on the Arc that you have created in the previous step.



The 'Select Mesh Properties' dialog box is shown with the following settings:

- Generate:** ☒ Members, ☐ Elements
- Material:** Steel
- Element Attributes:** Type: BPHQ, Thickness: 0.20
- Member Attributes:** Type: FRAME, Beta: 90, Section: IPE330 IPE European
- Spacing U Direction:** ☒ Uniform (8), ☐ Variable, ☐ Defined by Line/Curve

The Select Mesh Properties form appears where you enter:

- Material : Steel
- Beta Angle: 90
- Section: IPE330
- Spacing U Direction: Uniform 8

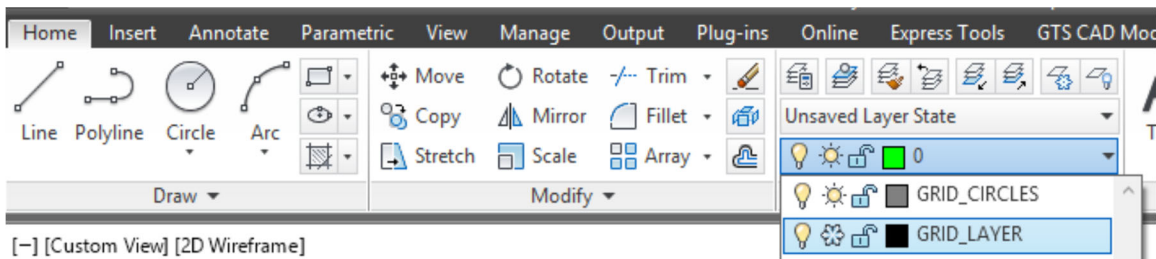
This command is going to generate 8 linear members equally spaced along the arc.

You can press the Preview button to see the members as they will be generated.

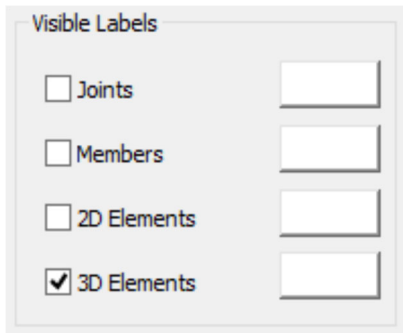
Press the Create button and 8 members, named 19 to 26, were created and 9 joints, named 17 to 25 were created along the arc.

Note that joints 17 and 25 are created on the top of joints 16 and 8 correspondingly. Later on, they are going to be merged together by deleting duplicate joints.

Step #13. Hide Grid: Since the Grid is no longer needed it can be hidden using AutoCAD/BricsCAD commands to freeze the layer GRID_LAYER. This can be done by clicking on the Home Tab and then selecting the GRID_LAYER from the drop down list of layers and clicking on its freeze icon.



Step #14. Turn OFF labeling:



Click on the icon  **Options** in the ribbon bar and then uncheck the Visible Labels option for Joints, Members and 2D Elements.

Now labeling is turned off and it is easier and faster to view and control the model.

Note: You can also delete or hide the Arc line as it is no longer needed. It can be deleted by selecting it with the mouse, and then pressing the keyboard button <Delete>. Be careful not to select any members or joints, but only the Arc line.

Step #15. Mirror the structure: Switch to a floor plan view, by pressing the TOP of AutoCAD's View Cube or BricsCAD's View Chair.

Then, type MIRROR and when you get the notification *Maintain incidence order (Yes/No)*. Press Yes to maintain the local coordinate system when mirroring.

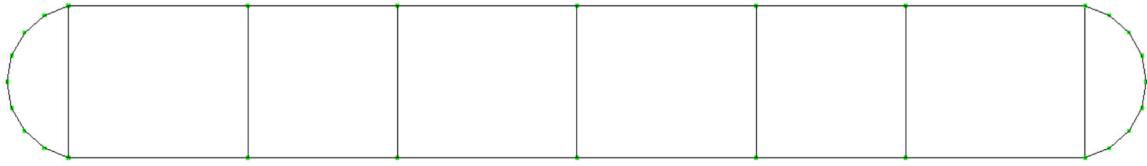
NOTE: This option is very useful when mirroring structures having shell finite elements and the mirror line lies along the element's XY plane. If you choose to maintain the incidence order, then after the mirroring the local Z axis of the source and copied finite elements have the same orientation.

You are then immediately prompted to *Select objects: (AutoCAD)* or *Select entities to mirror: (BricsCAD)* select the right part of the structure, but not the members and joints that are on the mirror line. In order to make this selection, make the 1st and the 2nd click of the mouse at the points 1 and 2 as shown in the picture below and press <ENTER>. You will get a confirmation that 44 objects were found (or 45 if you still have the arc).



When you get the message *Specify first point of mirror line (AutoCAD)* or *Start of mirror line (BricsCAD)*: click on the joint at points 3 and then *Specify second point of mirror line (AutoCAD)* or *End of mirror line (BricsCAD)*: click on the joint at point 4 as shown in the picture above.

Then, press <ENTER> and reply to the question *Erase source objects?* [Yes/No] <N> (AutoCAD) or *Delete the original entities?* [Yes-Delete entities(Yes)/No-Keep entities(No)] <No-Keep entities> (BricsCAD) so as not to delete the right part of the structure. The structure after the mirror command will look like the following picture:

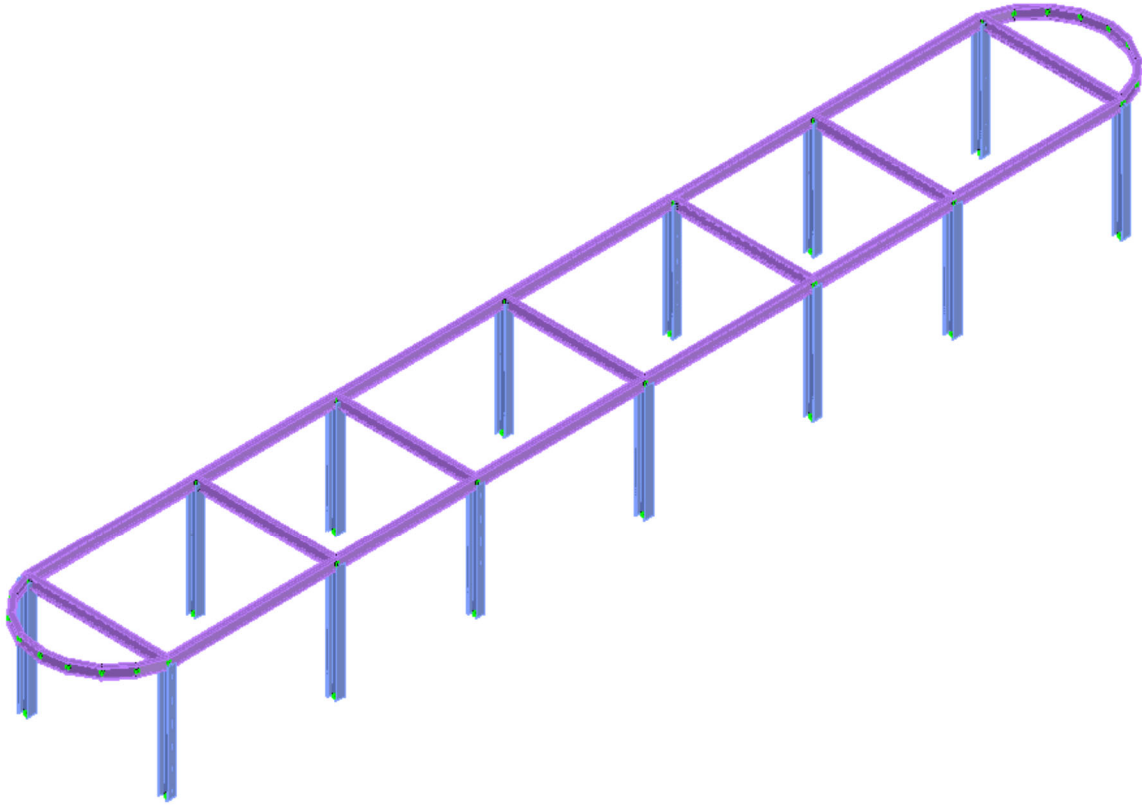


Step #16. Switch to 3D View: Press the house (AutoCAD) or chair (BricsCAD) icon to change the view to Isometric, and type Z and E (Zoom, Extents). Click on the icon  Colors to set different colors for each profile.

Sections Groups		
Categories		
Sections	Color	Visible
HE320B	161	☒
IPE330	191	☒
IPE120	1	☒
60x60x5	50	☒

Press OK to close the Color Options Dialog. Each cross section will now have a different color.

Press the icon  to display the 3D solid view of the model, replacing the wireframe view:

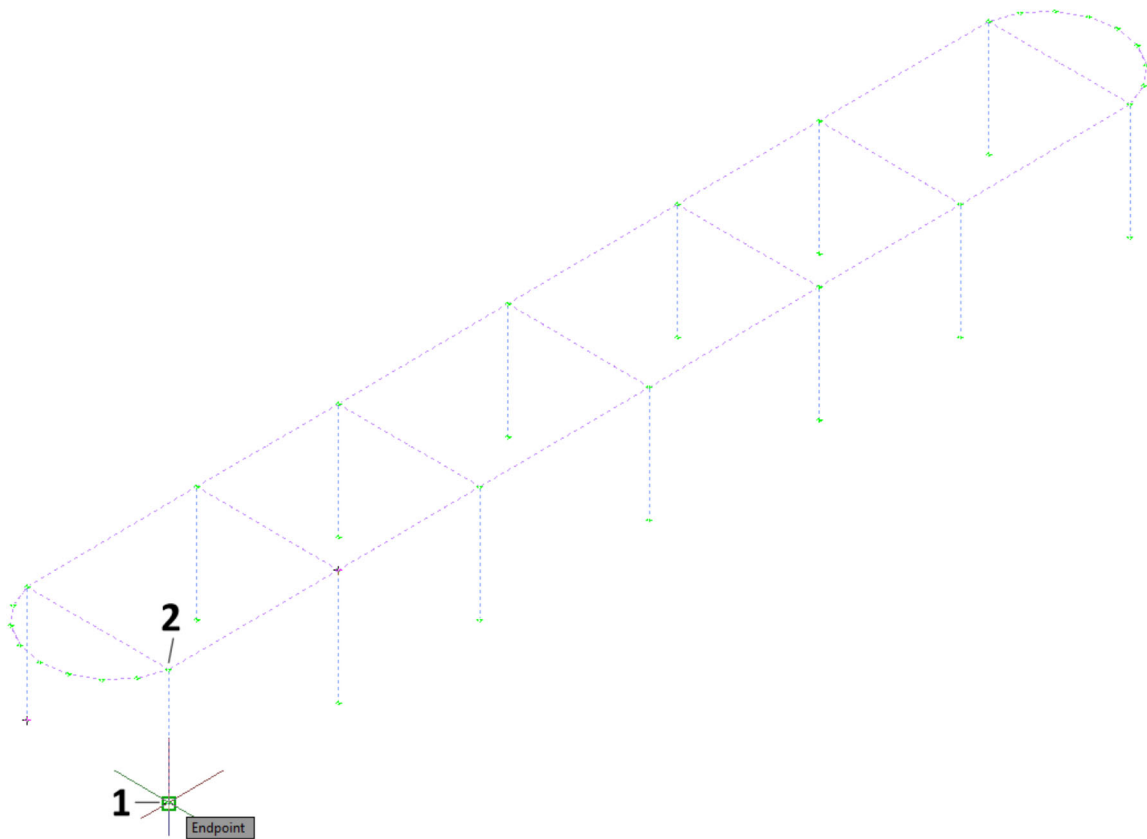


Press the icon  **Frame** to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

Step #17. Save your Model: In order to save your model just use AutoCAD's or BricsCAD's save command and store the DWG using any filename that you want.

3.5. Create the 2nd floor

Step #18. Copy the members and joints of the 1st floor to the 2nd: Type the command `COPY` and when asked to *Select objects*: type `ALL`, so that everything is selected. You will get a verification with the number of objects/entities selected and then press `<ENTER>`.



In order to *Enter base point or [Displacement/mOde] <Displacement>*, click at the base of any column such as point 1 in the picture above.

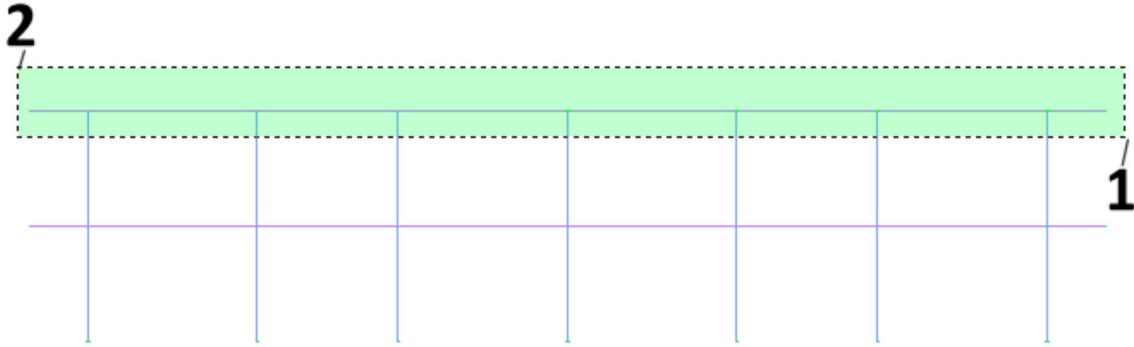
In order to *Enter second point* : click at the top of the same column such as point 2 of the picture and then press ESC in order to terminate the copy function.

All entities of floor 1 are now copied to floor 2.

Step #19. Correct the Z coordinates of the 2nd floor: The copied columns are 4.00m long, since they were copied from the 1st floor. In addition, the Z coordinate of the beams is equal to 8.00m instead of 7.00m. Therefore, all copied joints that have Z coordinate equal to 8.00m should be moved 1.00m lower.

Switch to FRONT View, by clicking on AutoCAD's view cube or BricsCAD's view chair and make sure that you are in the World UCS by typing UCS and W.

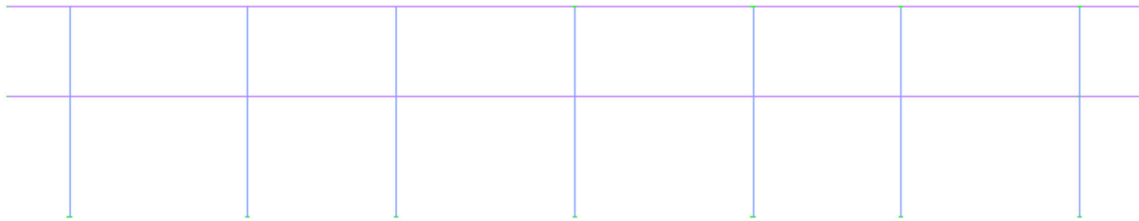
Type MOVE in order to initiate AutoCAD's/BricsCAD's move command and when asked to Select objects (or entities), click on points 1 and 2, as shown in the picture below, selecting all the entities that belong to the top of the 2nd floor. You will get a notification about the selected entities and press <ENTER>.



In order to *Enter base point or [Displacement] <Displacement>*: click ANYWHERE on the screen. It makes no difference where you click since relative coordinates will be used to define the displacement.

In order to *Enter second point*: type @0,0,-1 and press <ENTER>.

Now the height of the 2nd floor is correct and equal to 3.00 meters.



Note that you can reach the same result by switching to the 2nd Level using the icon  **Higher Level** and start entering the columns one-by-one, as you did in the 1st Level. Columns will then have the correct height (3.00m), since the height of the second Level is defined equal to 3.00m when the levels were defined in a previous step. Then, copy only the beams from the 1st Level to the correct position. However, it is somewhat faster to copy everything and then fix the Z coordinate with a simple MOVE command as you did in the previous steps.

3.6. Create the 3rd floor

Step #20. Copy the members and joints of the 2nd floor to the 3rd : Type the command COPY and when asked to *Select objects/entities*: click on the similar two points that were used in the previous MOVE command as shown in the following figure.

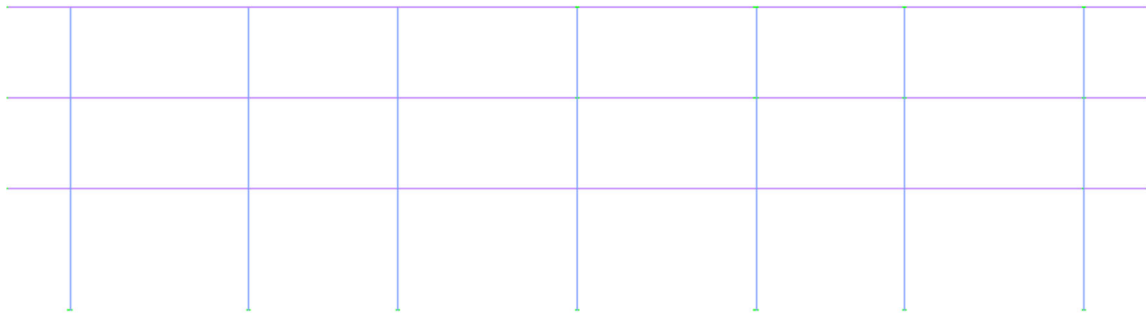


Since this is a crossing window, the columns are automatically selected. You will get a notification about the selected entities and press <ENTER>.

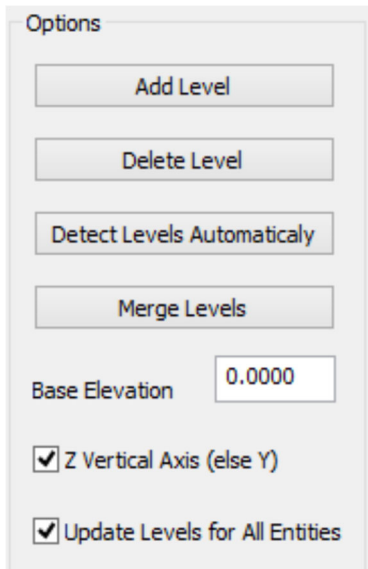
In order to *Enter base point*: click ANYWHERE on the screen. It makes no difference where you click since relative coordinates will be used to define the displacement.

In order to *Enter second point*: type @0,0,3 and press ESC.

Now the 2nd floor is copied to the 3rd one. There is no need to correct the Z coordinates as was done when the 2nd floor was moved since the second and third levels have the same height. Press the ESC button to exit the Copy command.



Step #21. Assign Level Properties:



Since all members were created from Level 1 using COPY commands, all of them belong to level 1 and their correct Level property should be assigned.



Click at icon **Levels** , check the option Update Levels for All Entities and press OK.

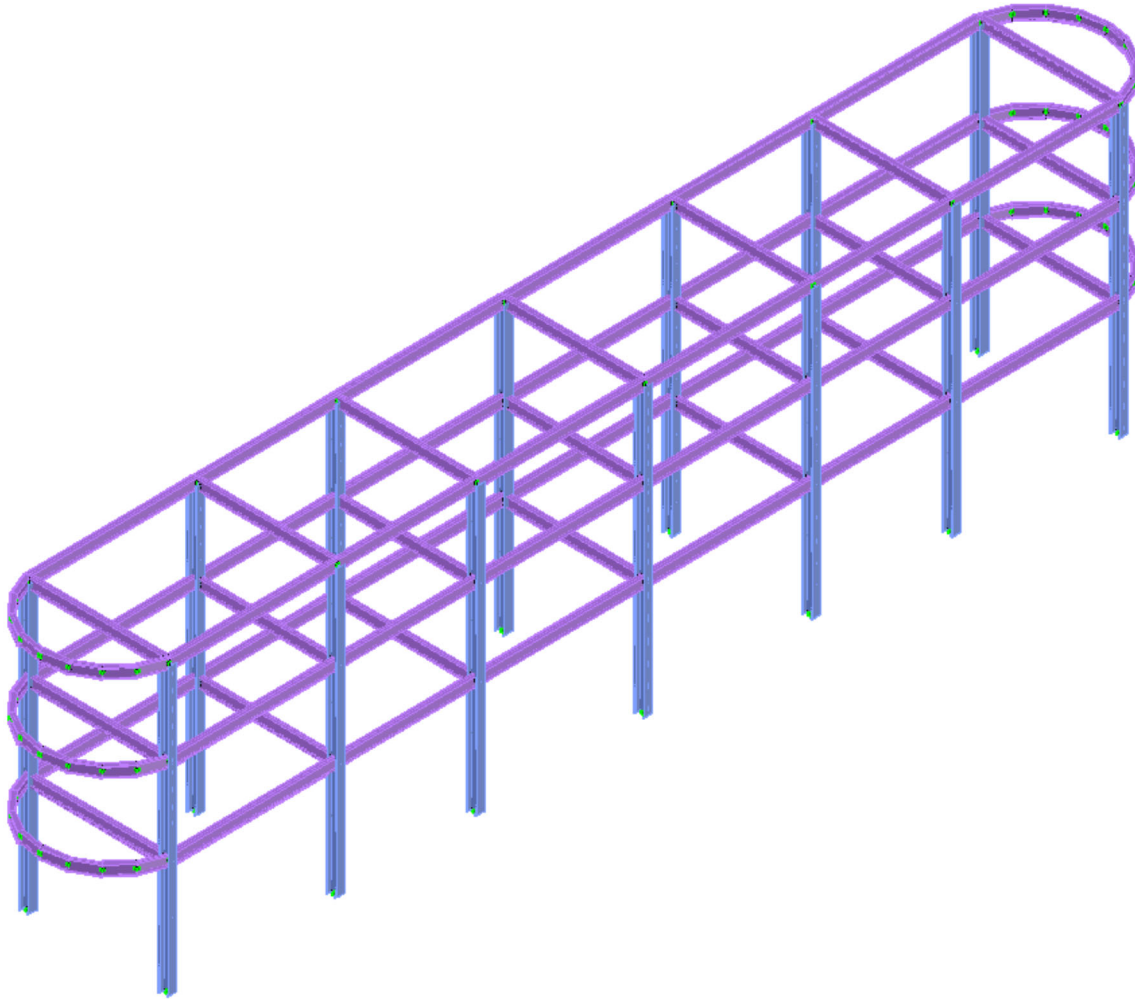
Now every Member or Joint has the correct Level property depending on its Z coordinate.

You can switch between the levels of the model using the **▲ Higher Level** and **▼ Lower Level** icons. The current level appears in the top caption of AutoCAD's screen.

You can make whole structure visible by clicking at the icon **All Levels ON** .

Step #22. View and Save your model: Press the house icon to change the view to Isometric, and type Z and E (Zoom, Extents).

Press the icon  to display the 3D solid view of the model, replacing the wireframe view as shown in the following figure:



Save your model, using a different file name (Save As...). By saving your model with a different name each time, it is easier to back up to a previous state of the model.

3.7. Create bracing

Step #23. Place bracing members at the front:

Press the icon  **Frame** to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

Levels	Height	Elevation	Visible
1	4.000000	4.000000	☒
2	3.000000	7.000000	☐
3	3.000000	10.000000	☐



Click at icon **Levels**, check the visible property for level 1 and uncheck it for all other levels and press OK.

Make sure that “Level :1” appears at the top of CAD window, else use the  Higher Level and  Lower Level icons to move to Level 1.

Now only Level 1 is visible and it is easier to add the bracing members. Click on the icon  Generate and the Place Member form appears.

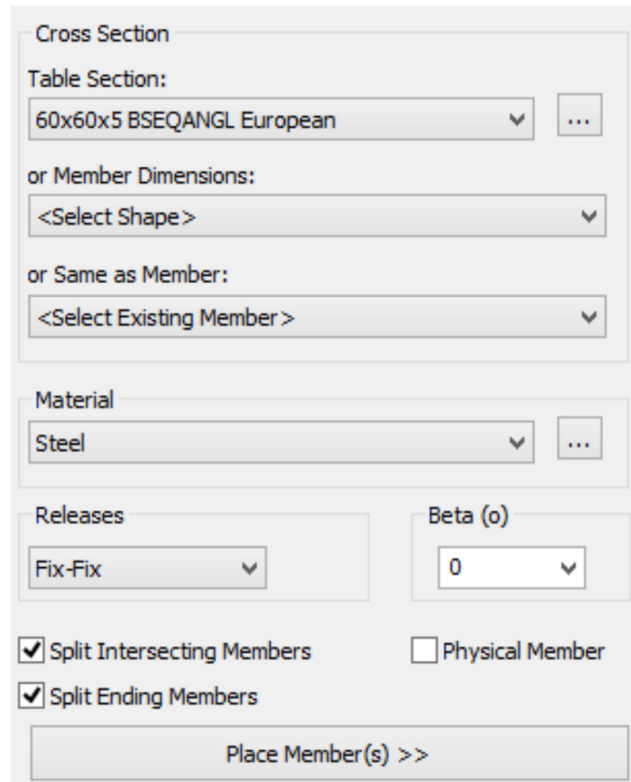
Select 60x60x5 as the cross section and make sure that Material is set to *Steel*, Releases to *Fix-Fix* and Beta angle is 0.

Press the “Place Member(s)>>” button.

Click on the joint located at Point 1 of the following image.

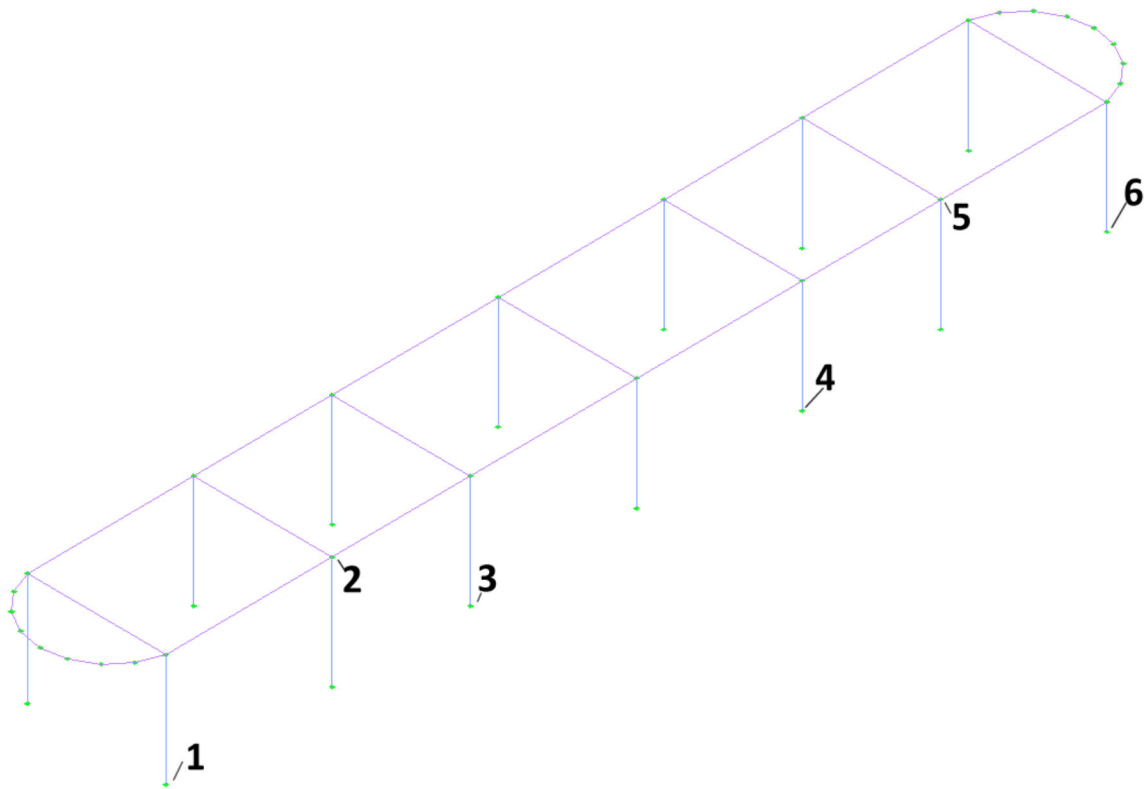
Click on joint located at Point 2 and the first bracing member is created.

Click again on joint located at Point 2 and then on the joint located at Point 3 and the second bracing member is created



Click on the joint located at Point 4, click on the joint located at Point 5 and the third bracing member is created.

Click again on the joint located at Point 5, click on the joint located at Point 6 and the fourth bracing member is created.



Press ESC to terminate the Generate Beam command.

Step #24. Change the properties of the Bracing Members: Click on the icon  **Change Member** in the Find/Change/Check panel and when asked to Select objects: click on the 4 bracing members created in a previous Step and press <ENTER>.

The Member Properties [Multiple Selection] form is displayed. Now, change the type to SPACE TRUSS and press OK.

Member Properties [Multiple Selection]

Model Section Properties Member Loads Member Temperature Loads Member Distortions

General

Name :

Level :

Type - Incidences

Type : SPACE TRUSS
 <Select Type>
 SPACE FRAME
 SPACE TRUSS

Start :

End :

Beta Angle :

Physical Member

Groups

☐ Inactive

Groups that Member belongs

Section Properties

Section : <Select Section>

Ax : Ix :

Ay : Iy :

Az : Iz :

Sy : Ey :

Sz : Ez :

Yd : Yc :

Zd : Zc :

Shape Code :

Material Properties

Material : <Select Material>

E : Density :

G : CTE :

Poisson :

Releases & Elastic Connection spring values

Quick Selection : Fix-Fix

Start	Spring	End	Spring
☐ Fx		☐ Fx	
☐ Fy		☐ Fy	
☐ Fz		☐ Fz	
☐ Mx		☐ Mx	
☐ My		☐ My	
☐ Mz		☐ Mz	

End Sizes OR Member Eccentricities (Offsets)

Start	End
Sizes :	
X :	X :
Y :	Y :
Z :	Z :

OK Cancel Apply Help

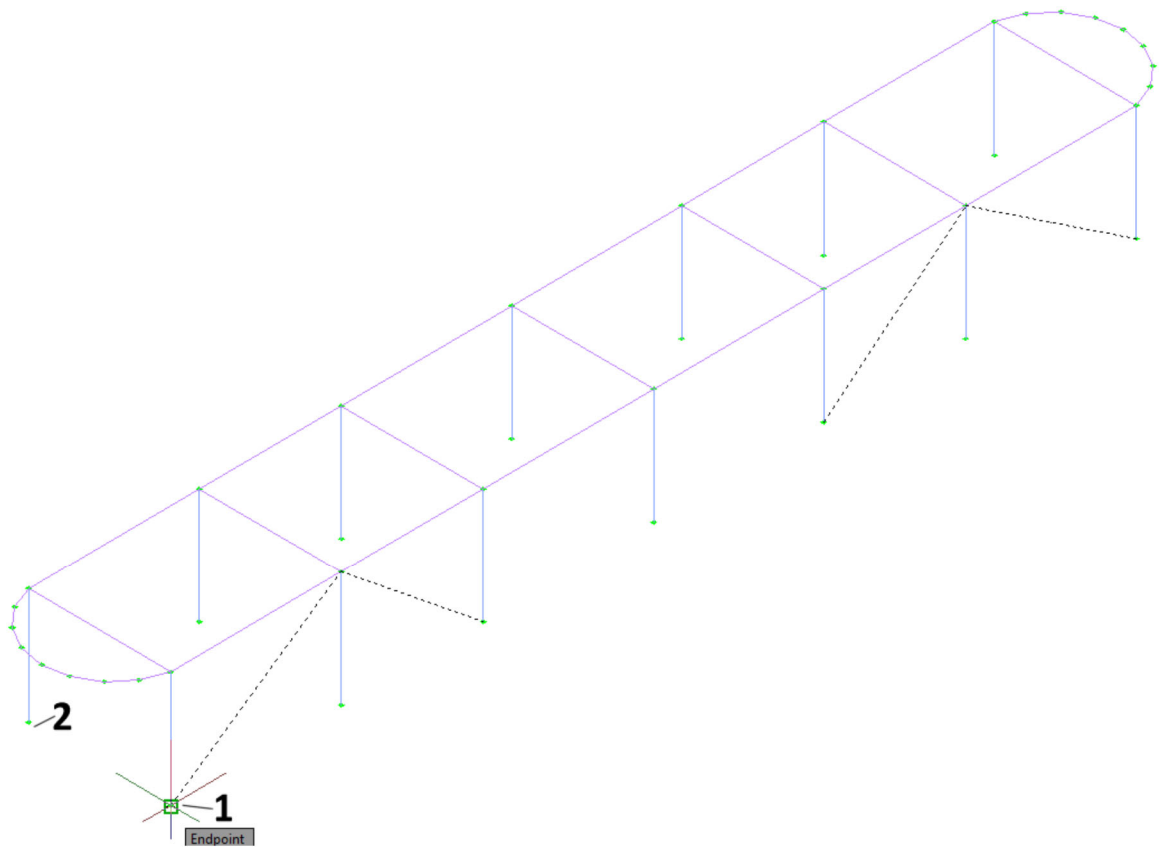
This modification applies to all selected members.

Step #25. Copy bracing member to the back: Type COPY and when asked to Select objects/entities: click on the 4 bracing members created in a previous Step and press <ENTER>.

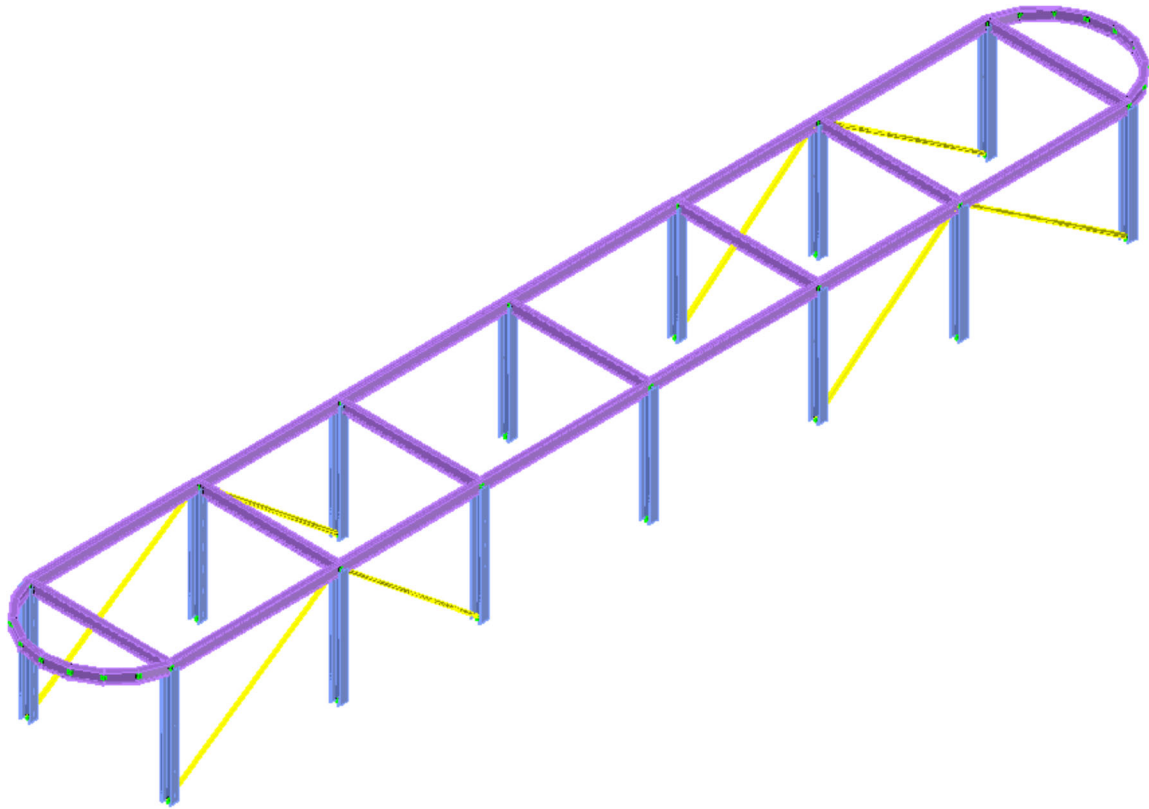
In order to *Enter base point*: click on the Joint at Point 1 of the following image.

In order to *Enter second point*: click on the Joint at Point 2 inf the image on the next page.

Press ESC to terminate the COPY command.



Step #26. View and Save your model: Press the icon  to display the 3D solid view and then the icon  Colors and OK. The model looks like as shown in the following image:



Save your model, using a different file name (Save As...).

Make the entire structure visible by clicking at the icon  All Levels ON .

Click on the icon  Frame to switch back to the wireframe view.

3.8. Create girders

Step #27. Split the beam members at the top level: Clicking on the  Higher Level icon move to level 3. The current level is displayed at the top caption of AutoCAD/BricsCAD:

GTS CAD Modeler | M KN DEG CEN SEC | Level: 3



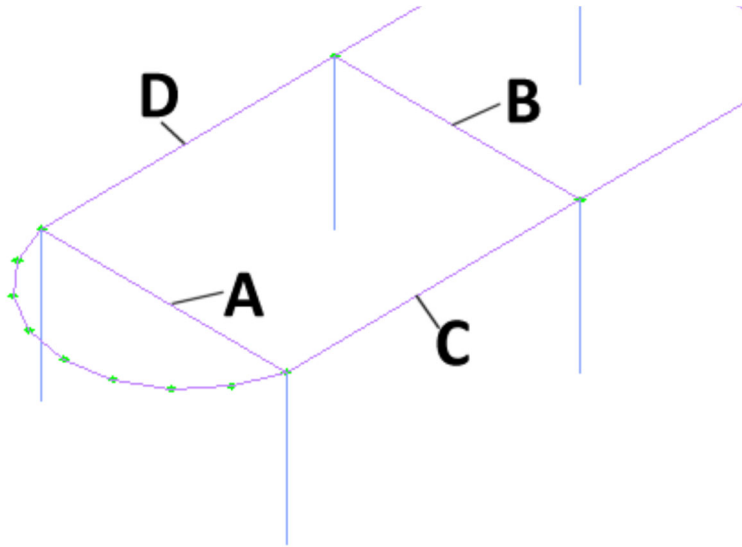
Click on the icon  Split and click on the members A and B as shown at the following image, and then press <ENTER>.

In order to define the Distance for splitting the member or the number of parts (negative number), enter -8 , so that the beams A and B will be split into 8 equal parts.



Click again on the icon  Split and click on the members C and D as shown at the following image, and then press <ENTER>.

In order to define the Distance for splitting the member or the number of parts (negative number), enter -4 , so that the beams C and D will be split into 4 equal parts.



Step #28. Place girder members at the top level: Click on the icon **Generate** and Place Member form appears.

Cross Section

Table Section:
 IPE120 IPE European ...

or Member Dimensions:
 <Select Shape>

or Same as Member:
 <Select Existing Member>

Material
 Steel ...

Releases
 Fix-Fix

Beta (°)
 90

☒ Split Intersecting Members ☐ Physical Member

☐ Split Ending Members

Place Member(s) >>

Select *IPE120* as the cross section and make sure that Material is set to *Steel*, Releases to *Fix-Fix* and Beta angle is *90*.

Press the “Place Member >>” button.

Click on the joint located at the point 1 of the following image. Click on the joint at point 2 and the girder member is generated.

Having the command still active, click on the joints at points 3 and 4 and another girder member is generated.

Continue by clicking on joints at points 5 and 6 and another girder member is generated.

Continue by clicking on joints at points 7 and 8 and another girder member is generated.

Continue by clicking on joints at points 9 and 10 and another girder member is generated.

Continue by clicking on joints at points 11 and 12 and another girder member is generated.

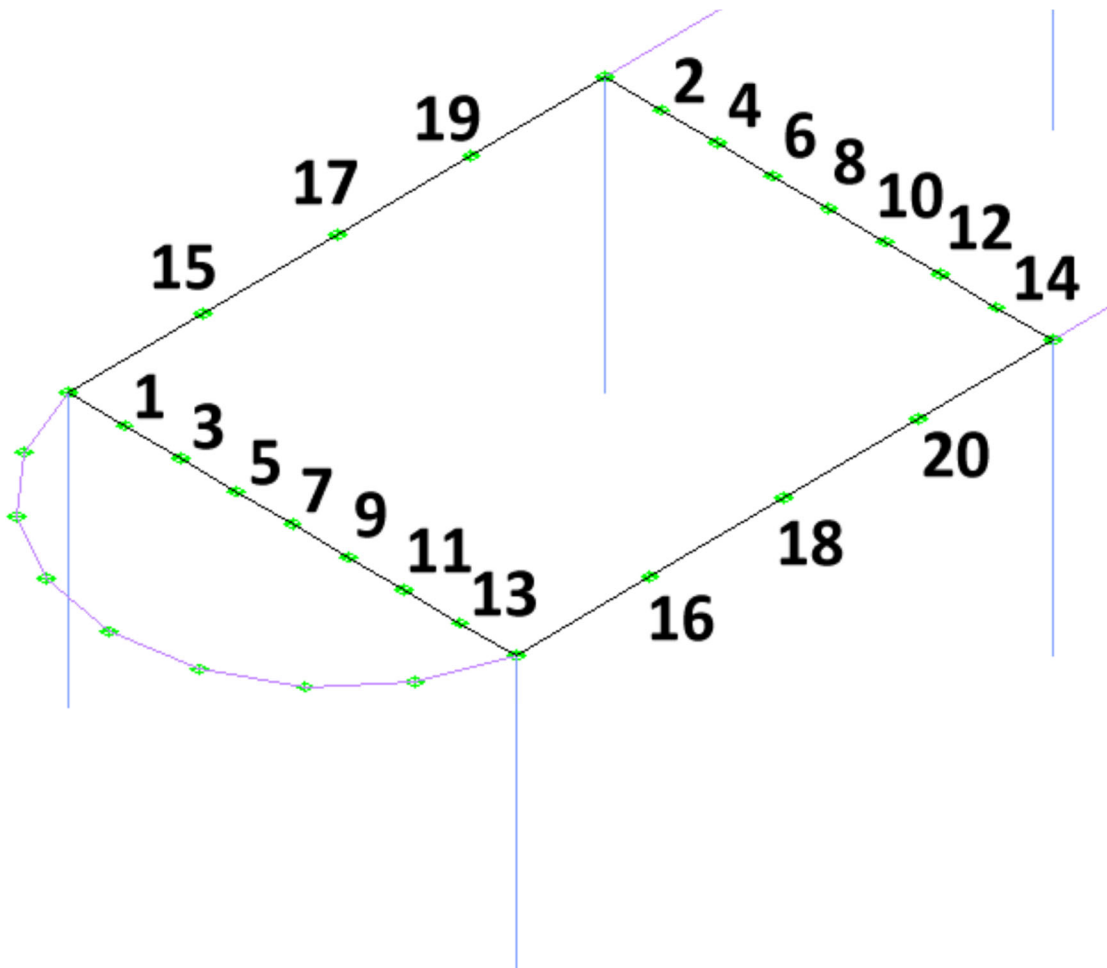
Continue by clicking on joints at points 13 and 14 and another girder member is generated.

Make sure that the option "Split Intersecting Members" is ON, so that common joints will be created along the previously created X-direction girders. Click on the joint located at point 15 and then click on joint at point 16 and the girder member is generated. Existing girders are split.

Repeat the procedure:

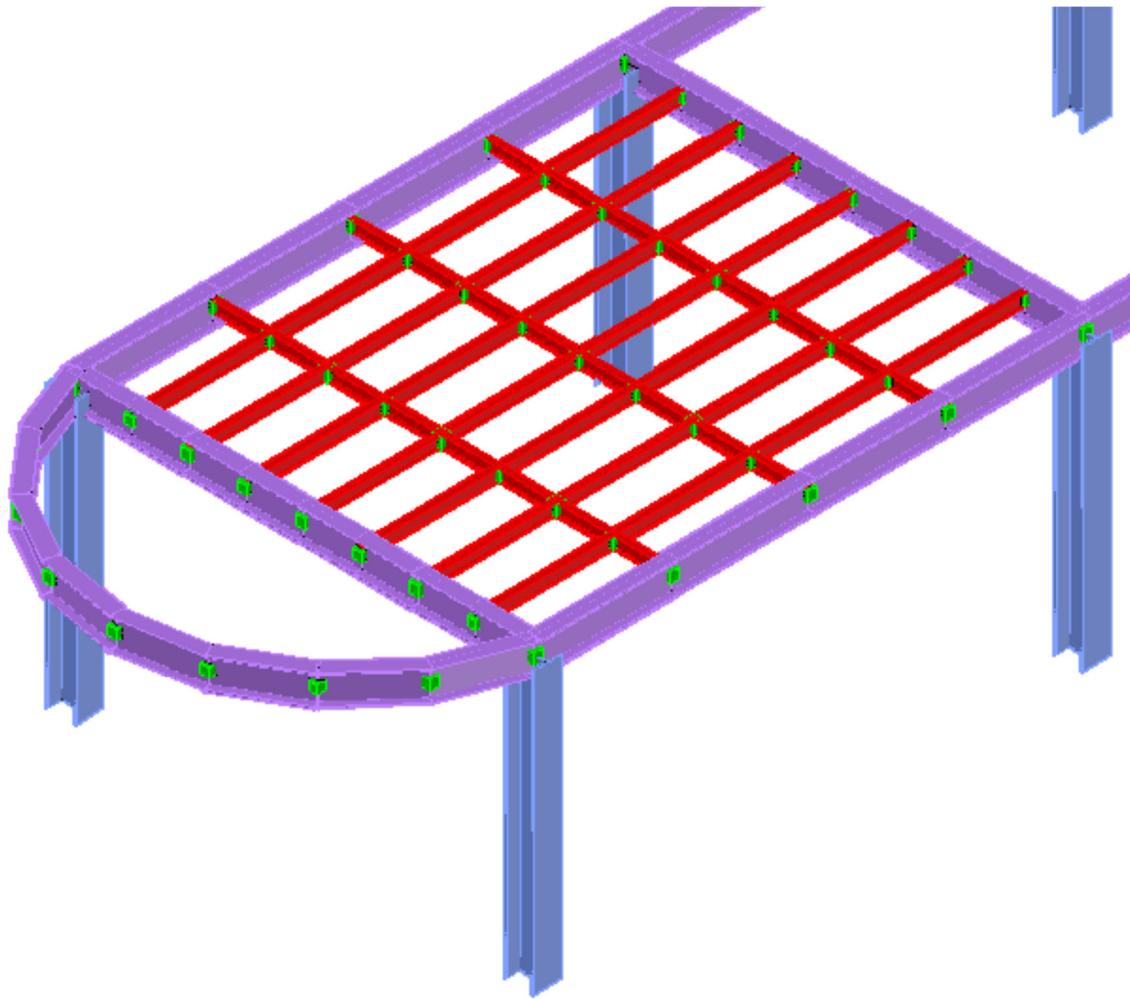
Click on the joint located at point 17 and then click on the joint at point 18 and the girder member is generated. Existing girders are split.

Click on the joint located at point 19 and then click on the joint at point 20 and the girder member is generated. Existing girders are split.



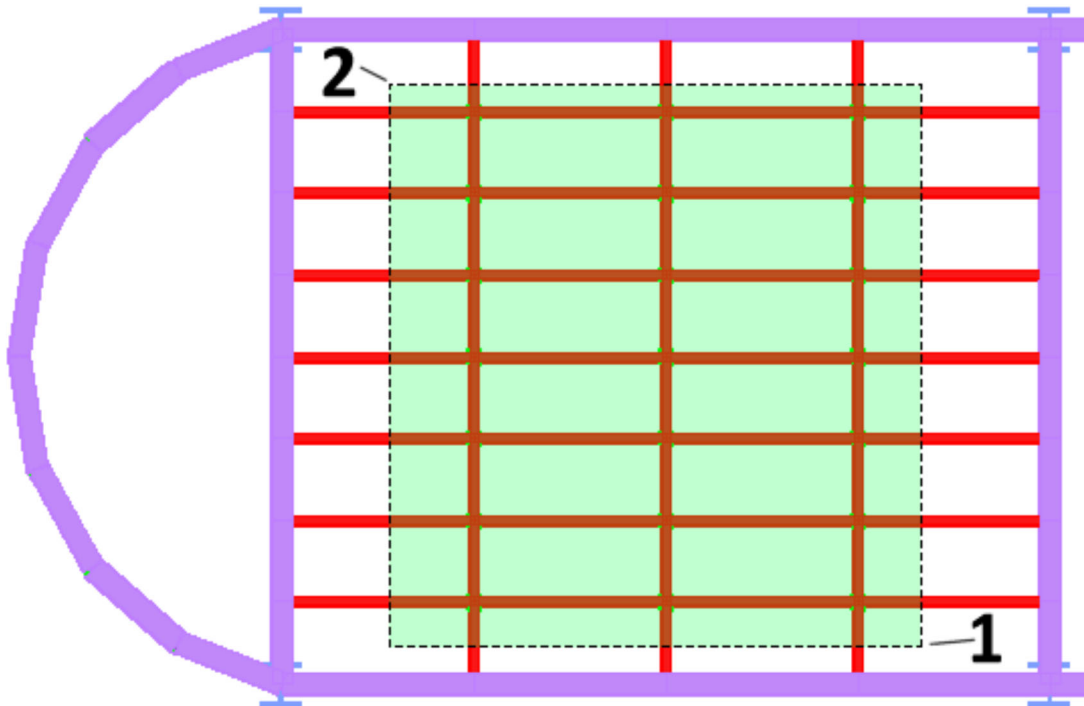
Press ESC to terminate the Generate Beam Command.

Step #29. Add eccentricities to the Girders: Press the icon  to display the 3D solid view and then the icon  Colors and OK. The model looks like:



Click on the Top Icon on AutoCAD's View Cube or BricsCAD's View Chair to switch to the top view in order to be able to select girder members easily.

Click on the icon  Change Member and when asked to Select objects: click on the 2 Points of the following image, to select all girder members, and press <ENTER>.



The *Member Properties [Multiple Selection]* form is displayed so you may now specify member eccentricities.

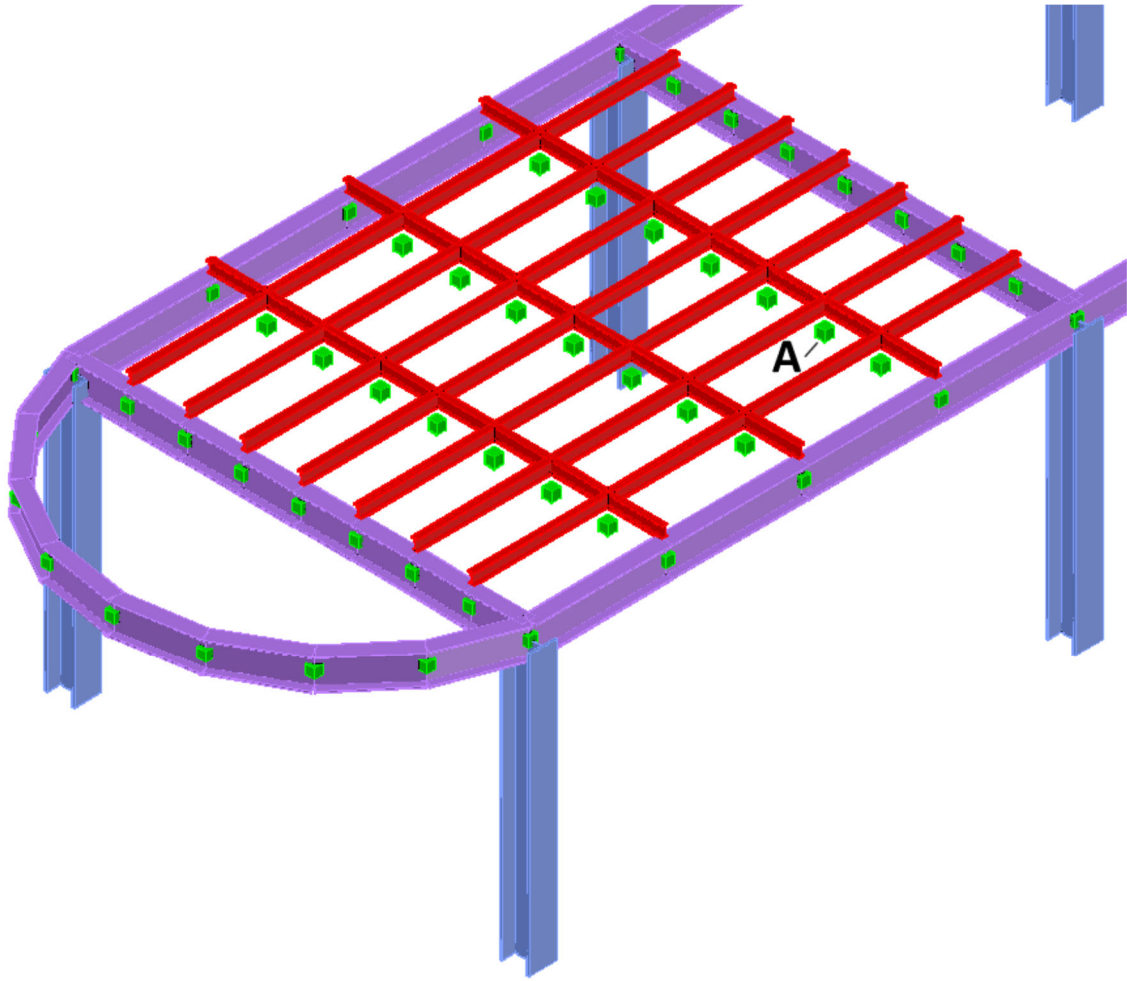
End Sizes OR Member Eccentricities (Offsets)			
	Start		End
Sizes :			
X :		X :	
Y :		Y :	
Z :	0.25	Z :	25cm

Note: Some Edit Boxes appear in yellow background and green fonts, like Eccentricities. You can use mixed units in the yellow edit boxes. To understand this feature enter 0.25 for the Z Starting Eccentricity and 25cm as the Z Ending Eccentricity.

Press OK.

Now the deformable axis of the girder members has been moved 0.25m up in the Z direction.

Switch back to the isometric view by clicking on the House icon on AutoCAD's Cube or BricsCAD's Top Front Left at view Chair to see the result. The girder members now sit on the upper flange of the beam members.

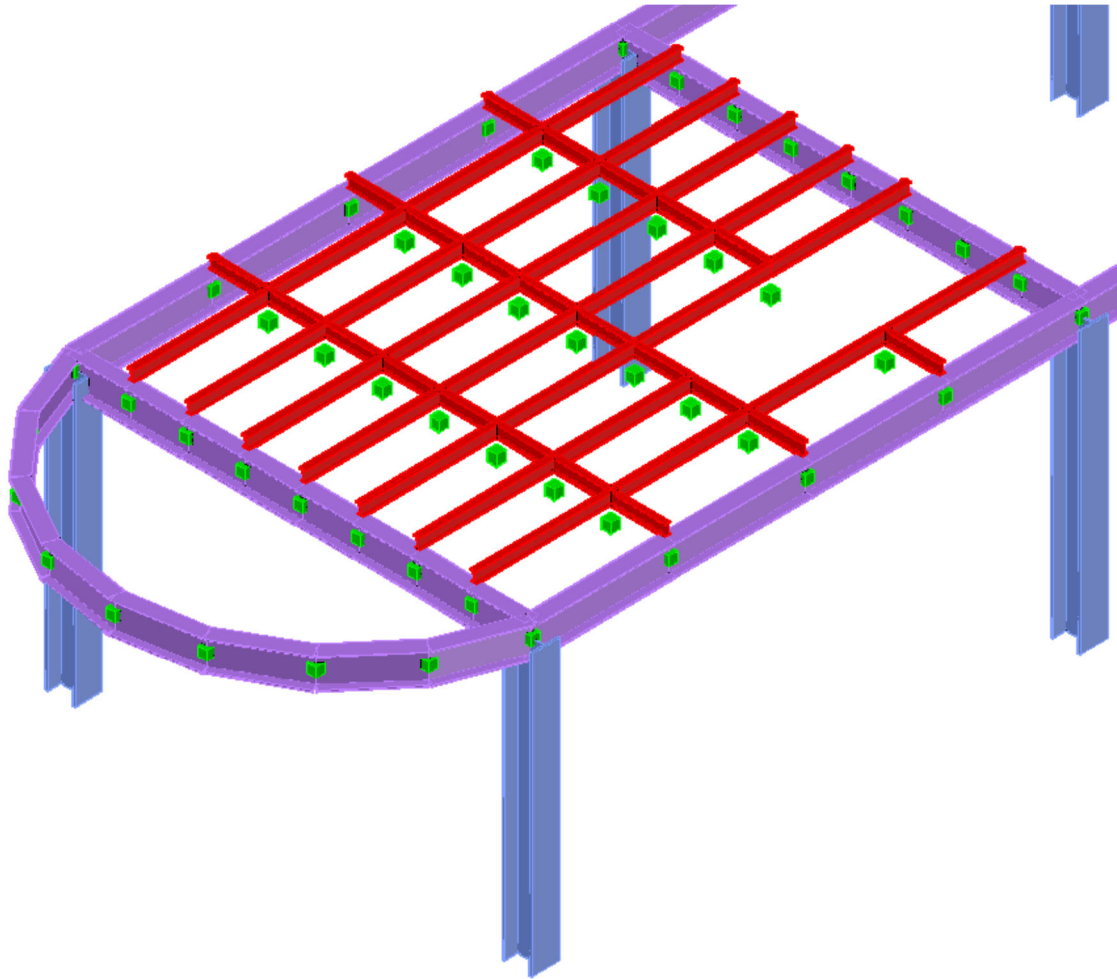


3.9. Create an opening

Step #30. Delete a joint to create an opening: Select the Joint Located in Point A of the image above and press the DEL key.

Warning: *All structural entities (members, elements, etc) connected to this Joint will also be deleted? (Yes/No)* appears and continue by pressing Y and <ENTER>.

The Joint is deleted together with all members connected to the joint.



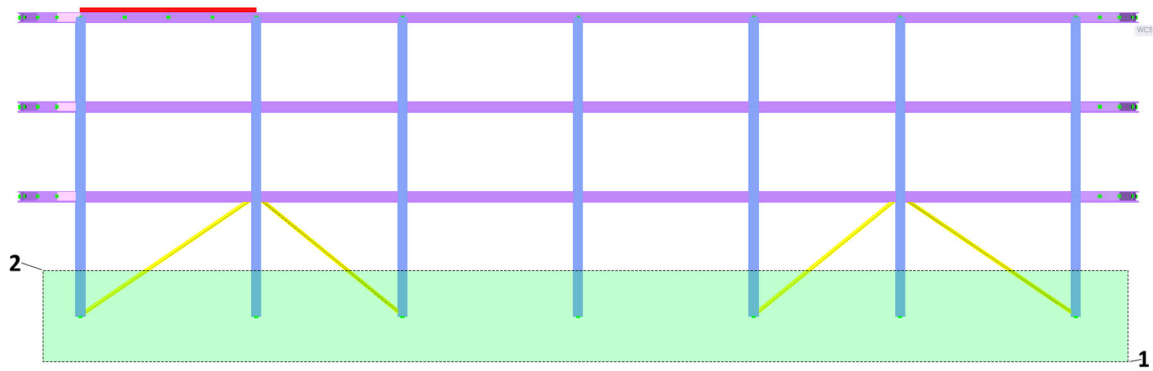
3.10. Create Supports

Step #31. Support the joints at the base of the model:

Make the entire structure visible by clicking at the icon  All Levels ON and press Z and E (Zoom Extents).

Switch to the FRONT View, by clicking on Front on AutoCAD's cube or BricsCAD's chair.

Click on the icon  Support and select the window by clicking at points 1 and 2 in the following image. All the bottom joints are selected and press OK to finish the selection.



Restraints & Spring values

Quick Selection : Pin

Restraint	Spring	Restraint	Spring
☒ Fx		☐ Mx	
☒ Fy		☐ My	
☒ Fz		☐ Mz	

The Joint Properties [Multiple Selection] form appears.

Select Pin using Quick Selection, and Fx, Fy and Fz are automatically checked.

Press OK.

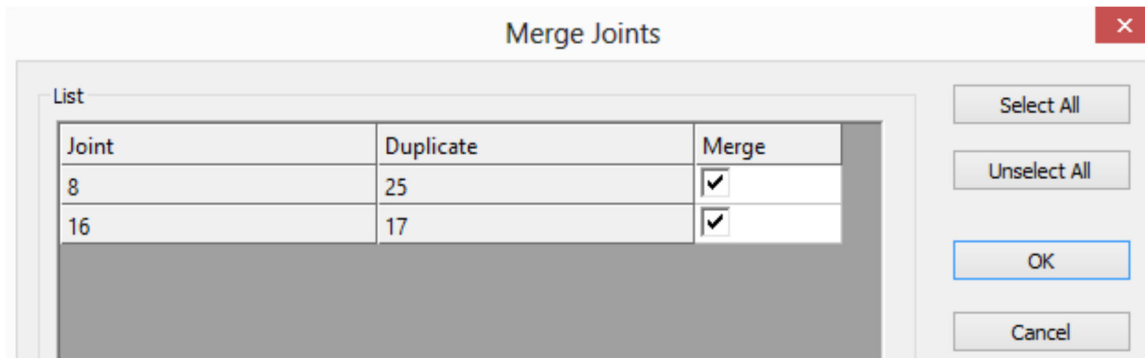
All the bottom joints are now pinned and have an orange color instead of green to indicate that they are supported.

3.11. Check the model

Step #32. Check for duplicate joints: In order to check for joints having the same coordinates, click on the icon  Joints Duplicates, under the “Check” Drop Button, located in Find/Change/Check at Ribbon Area.

For the Merge Tolerance <0.001000>, just press <ENTER> to accept the default value.

The Merge Joints form appears where you can see the list of joints having the same coordinates. Make sure that Merge option is checked for all pairs and press OK.



By entering the same command again, for the 2nd time, you should get the notification that *0 duplicate joints found*.

Step #33. Check for floating joints: In order to check for joints not connected to the model, click on the icon  **Joints Floatings**, under the “Check” Drop Button, located in Find/Change/Check at Ribbon Area. If your model was created as described so far, you should get a notification *0 floating joints found*.

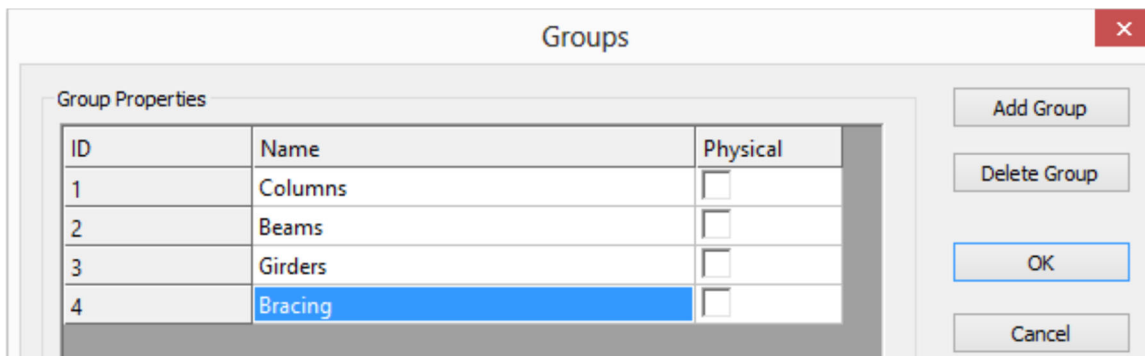
Note: You can also run all other checks of the same drop list, to check for Interference Joints, Duplicate Members, Zero Length Members, Duplicate Names and Database Integrity. You should not get any errors or warnings.

3.12. Define Groups

Step #34. Create Group Names: It is optional to define Groups in your model but it is strongly recommended to do so since it will be easier to control the display and selection for parts of your structure.



Click on the icon **List** in the Groups panel and the Group dialog appears.



Press the *Add Group* button and enter `Columns` as *Name* of the group.

Press the *Add Group* button and enter `Beams` as *Name* of the group.

Press the *Add Group* button and enter `Girders` as *Name* of the group.

Press the *Add Group* button and enter *Bracing* as *Name* of the group.

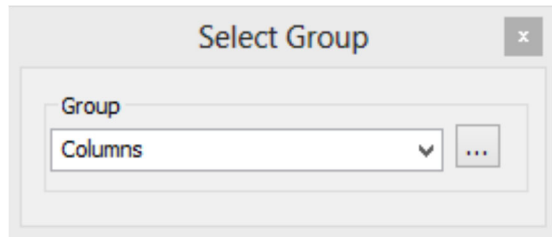
Press OK to close the form.

Step #35. Add Columns to their Group.

Click on the icon  Colors and make only profiles HE320B visible by unchecking all others.

Press OK.

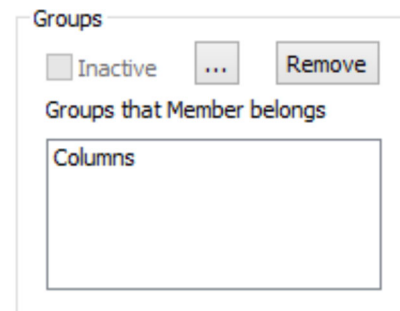
Categories		
Sections	Color	Visible
HE320B	161	☒
IPE330	191	☐
IPE120	1	☐
60x60x5	50	☐



Click on the icon  +Members in the Groups panel and make sure that the Group Columns is selected as the active group.

Select everything on the screen in a full window and press the Enter key.

To verify that the group is applied correctly, you can double click on a column and in the Member Properties form, you will be able to see its Group definition.

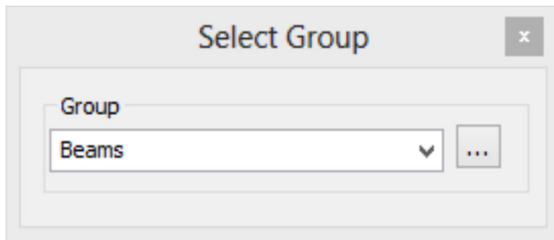


Step #36. Add Beams to their Group:

Click on the icon  Colors and make only profiles IPE330 visible by unchecking all others.

Press OK.

Categories		
Sections	Color	Visible
HE320B	161	☐
IPE330	191	☒
IPE120	1	☐
60x60x5	50	☐



Click on the icon  +Members and make sure that the Group Beams is selected as the active group.

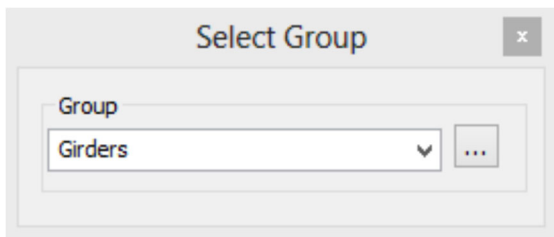
Select everything on the screen in a full window and press the Enter key.

Step #37. Add Girders to their Group:

Click on the icon  Colors and make only profiles IPE120 visible by unchecking all others.

Press OK.

Categories		
Sections	Color	Visible
HE320B	161	☐
IPE330	191	☐
IPE120	1	☒
60x60x5	50	☐



Click on the icon  +Members and make sure that the Group Girders is selected as the active group.

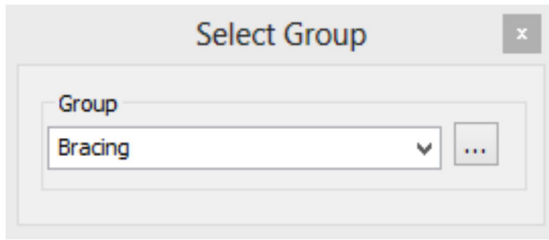
Select everything on the screen in a full window and press the Enter key.

Step #38. Add Bracing to their Group:

Click on the icon  Colors and make only profiles 60x60x5 to be visible by unchecking all others.

Press OK.

Categories		
Sections	Color	Visible
HE320B	161	☐
IPE330	191	☐
IPE120	1	☐
60x60x5	50	☒



Click on the icon  +Members and make sure that the Group Bracing is selected as the active group.

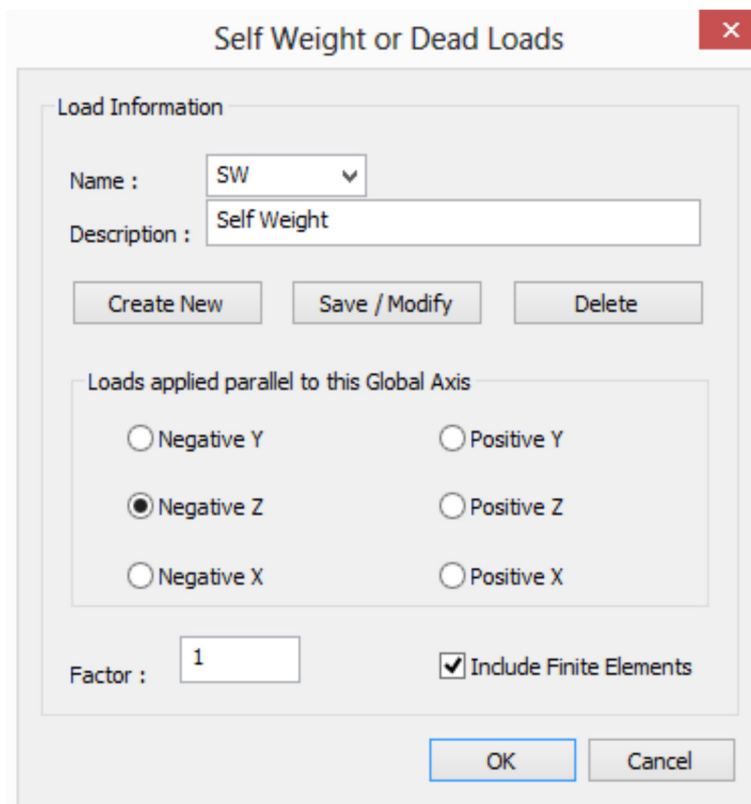
Select everything on the screen in a full window and press the Enter key.

3.13. Define Loads

Step #39. Define Self Weight: Click on the icon  Self Weight and the Self Weight dialog appears. Enter:

- SW as Name
- *Self Weight* as the Load Description
- *Negative Z* as the Global Axis for the direction of the load
- 1.0 as Factor
- Check Include finite elements

and press Create New to create the new loading and then OK to close the dialog.

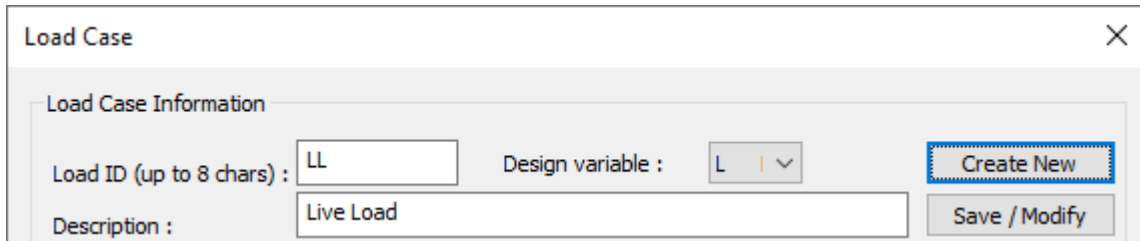


Step #40. Define Load Cases: Click on the icon  Load Cases and the Load dialog appears.

Enter:

- *LL* as Name
- *Live Load* as the Load Description
- *L* as Design variable

and press Create New.

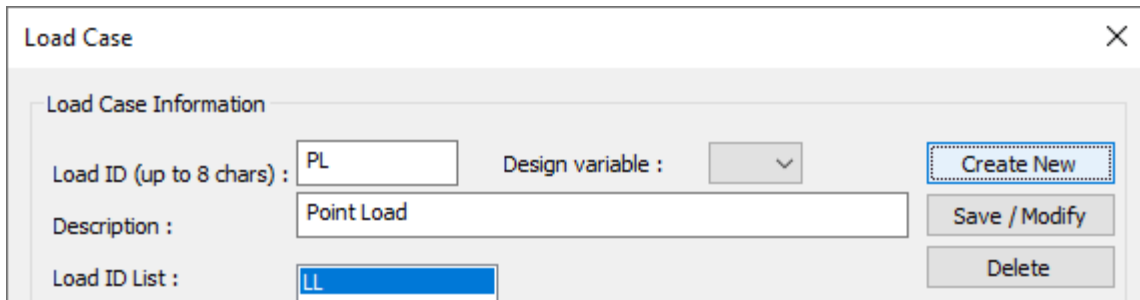


The 'Load Case' dialog box is shown. It has a title bar with a close button (X). Inside, there's a section titled 'Load Case Information'. It contains three input fields: 'Load ID (up to 8 chars):' with the value 'LL', 'Design variable:' with a dropdown menu showing 'L', and 'Description:' with the value 'Live Load'. To the right of these fields are two buttons: 'Create New' (highlighted with a blue dashed border) and 'Save / Modify'.

Enter:

- *PL* as Name
- *Point Load* as the Load Description

and press Create New.



The 'Load Case' dialog box is shown again. It has the same title bar and 'Load Case Information' section. The 'Load ID (up to 8 chars):' field now contains 'PL' and the 'Description:' field contains 'Point Load'. The 'Design variable:' dropdown menu is empty. To the right of these fields are two buttons: 'Create New' (highlighted with a blue dashed border) and 'Save / Modify'. Below the 'Description:' field, there is a new field 'Load ID List:' containing 'LL'. To the right of this field is a 'Delete' button.

Press Exit to close the dialog.

Step #41. Apply Live Loads: Live Loads will be applied only to Beam Members, therefore click on the icon  Colors. Select the 2nd Tab in order to colorize members by their group and make only the Group Beams visible and press OK.

Sections Groups		
Categories		
Groups	Color	Visible
Columns	161	☐
Beams	191	☒
Girders	1	☐
Bracing	50	☐
UnGrouped data	7	☒

Click on the icon  **Member** . Using a full window, select all entities that appear on screen and press <ENTER> to finish with the selection.

The Member Properties [Multiple Selection] form appears having the tab “Member Loads” active.

Click on LL at the “Empty Load Cases” list box and then enter

- *Uniform* as the Load Distribution
- *Force* as Load Type
- *Z Global* as Direction
- -5 as V1
- Fractional as Location
- 0.0 as L1
- 1.0 as L2

Press **Create New** and the number 125 appears next to the \$ symbol the Empty Load Cases list box. This is a notification that 125 members are loaded in Load Case LL.

Member Properties [Multiple Selection]

Model Section Properties Member Loads Member Temperature Loads Member Distortions

Applied Load Cases

Empty Load Cases

LL \$ Live Load \$[0]
PL \$ Point Load \$[0]

Loading

Load Case : LL

Description : Live Load

Loads in this Load Case :

Create New
Save / Modify
Delete

Load Distribution

Concentrated
Uniform
Linear
Triangular

Member Start

W

L1 L2

Load Type and Direction

Force
Moment

X
Y
Z

Local
Global

Loading Magnitude

V1 : -5
V2 :

Loading Location from Start

Fractional
Absolute

L1 : 0.0
L2 : 1.0

OK Cancel Apply Help

Press OK to close the dialog. Member Loads appear in red arrows. To clear the arrows select  Clear from the GTS Display Ribbon Tab.

By double clicking on one beam and then selecting the Member Loads tab and clicking on LL at the Applied Load Cases List Box, you will be able to view and edit the existing loading values of the specific beam.

Click on the icon  Colors, select the 1st Tab in order to colorize members by their section and then select everything to be visible and press OK.

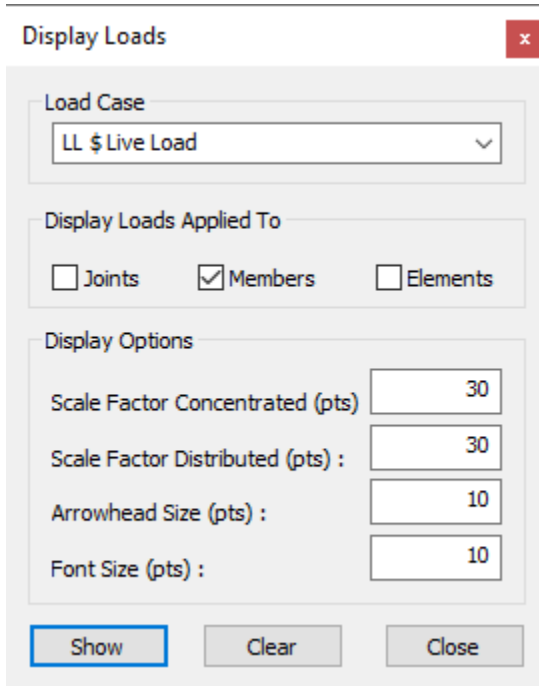
Sections Groups

Categories

Sections	Color	Visible
HE320B	161	☒
IPE330	191	☒
IPE120	1	☒
60x60x5	50	☒

Step #42. View Live Loads: Click at the icon  Frame to switch to the wireframe view.

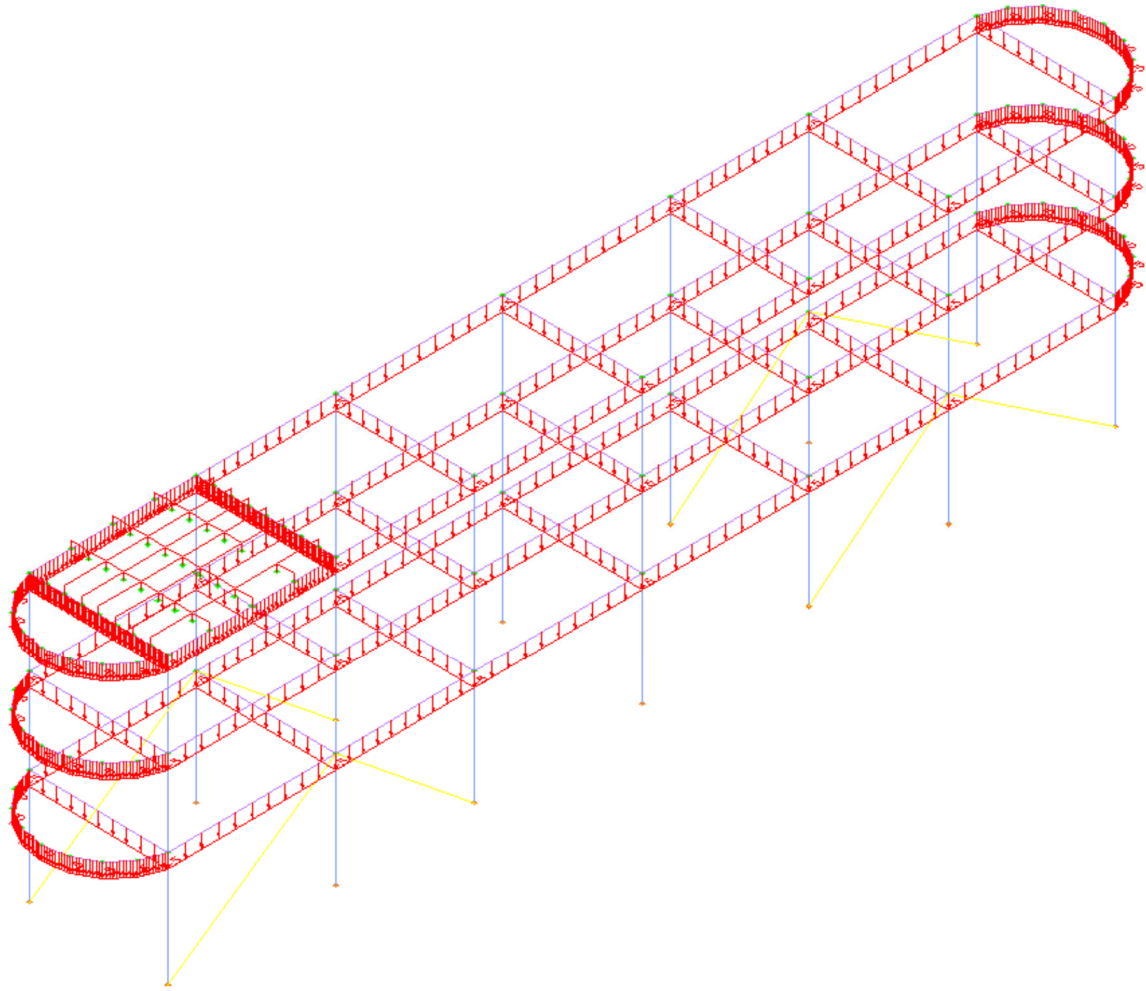
On the Menu Bar, click on GTS Display>Display>Member Loads and the Display Loads form appears:



Select LL as the Load case and leave the other display options at their default values.

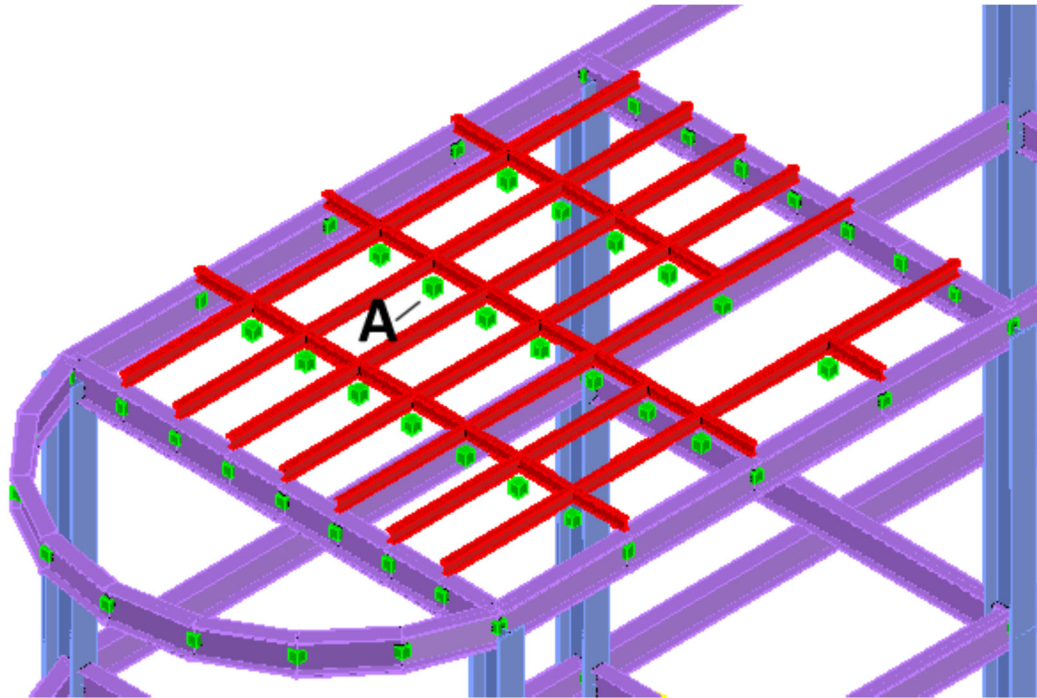
Press Show and the loading arrows are displayed.

Click on Clear when you are done and Close to exit from the dialog.



Click at the icon  to display the 3D solid view.

Step #43. Apply Joint Load: A Joint load will be applied to the Joint located at Point A of the following image.



Zoom closer to the specific point using AutoCAD's/BricsCAD's zooming functions.

Click on the icon  **Joint** , click on the joint at A and press <ENTER> to finish with the selection.

The Joint Properties form appears having the tab "Joint Generalized Loads" active.

Click on PL in the "Empty Load Cases" list box and then enter -3 for Force Z as shown in the Joint Properties form on the next page.

Press Create New, and the number 1 appears next to the \$ symbol in the Load Cases list box. This is a notification that 1 joint is loaded under the Load Case PL.

Joint Properties

Model Joint Generalized Loads

Applied Load Cases

Empty Load Cases

LL \$ Live Load \$[0]
PL \$ Point Load \$[0]

Loading

Load Case : PL Create New

Description : Point Load Save / Modify

Loads in this Load Case : ▼ Delete

Applied Load Values

Force X : Force Y : Force Z : -3

Moment X : Moment Y : Moment Z :

Applied Displacement Values

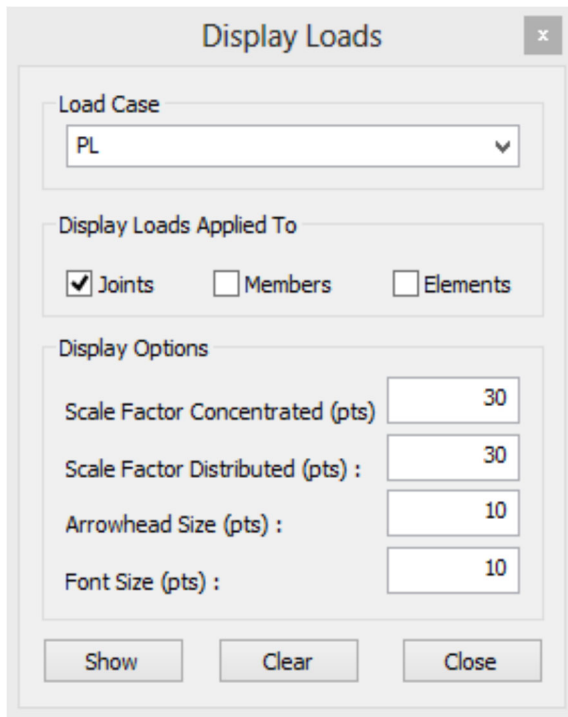
Tran X : Tran Y : Tran Z :

Rot X : Rot Y : Rot Z :

OK Cancel Apply Help

Step #44. View Joint Load: Click at the icon  Frame to switch to wireframe view.

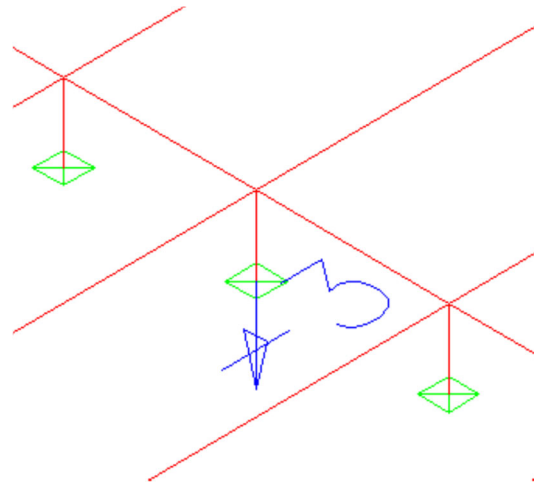
On the Menu Bar, click on GTS Display>Jont Loads and the Display Loads form appears:



Select PL as the Load case and leave the other display options at their default values.

Press Show and the loading arrows are displayed.

Click on Clear when you are done and Close to exit from the dialog.



Click at the icon  to display the 3D solid view.

Step #45. Define Area Load for Level 1: An area load equal to 1.0kN/m^2 along the vertical direction will be applied to whole level. Switch to Level 1 by clicking on the icon  Lower Level untill "Level 1" is displayed.

Click on the icon  Area Load in the Loads panel of the ribbon.

Area Load x

Generate

Name :

Description :

Load - Direction

Load Value :

Global Direction Perpendicular to the Loading Plane :

☐ X ☐ Y ☒ Z

Plane Tolerance :

Elevation

Plane Perpendicular at :

☒ Value (coordinate)

☐ Joint

Distribution

☒ Two way ☐ X

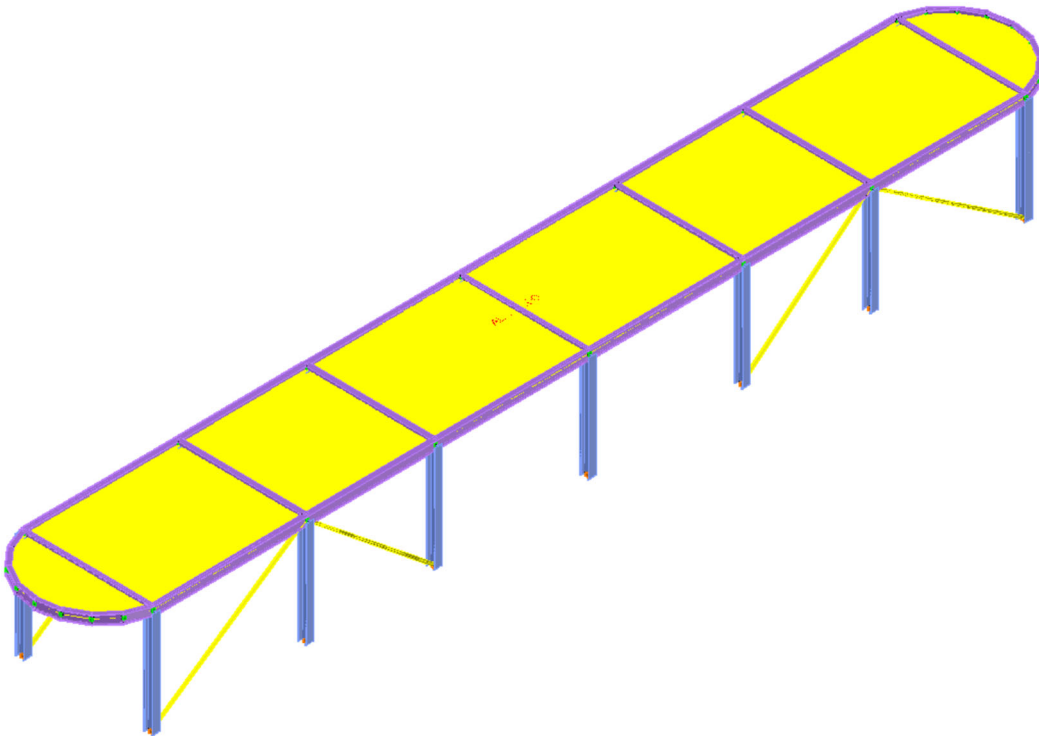
☐ One way ☐ Y

☐ Custom X Y

Type:

- *AL1* as Name
- *Area Load Level 1* as Description
- *1.0* as Load Value
- *Z* as Global Direction
- *4.0* as Elevation (the elevation of level 1)
- *Two way* as Distribution

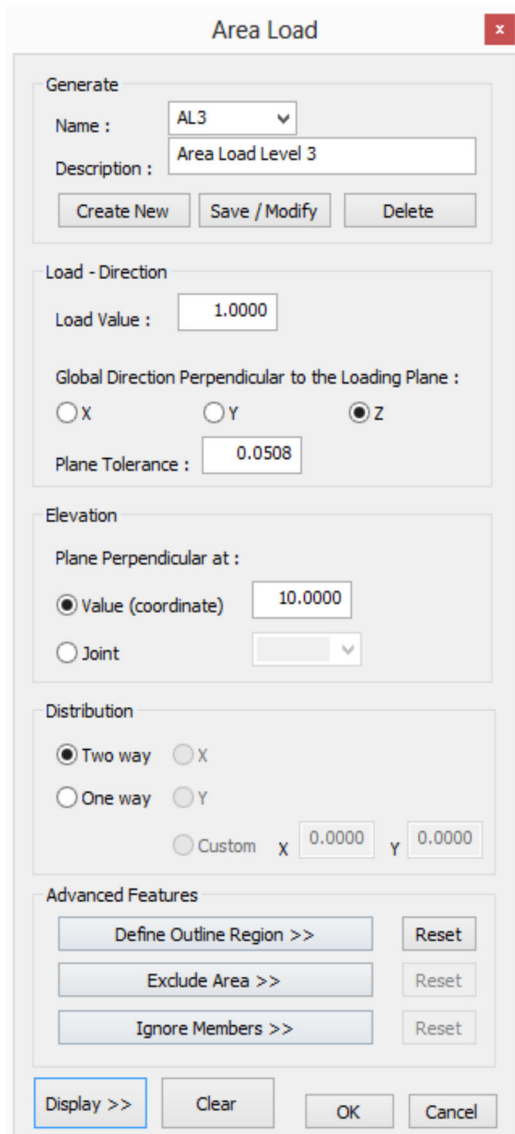
Press Display >> and the loaded areas that were automatically detected are displayed in yellow solid hatch, as shown below.



Press Clear to remove the solid hatch pattern and then OK to store the area load AL1.

Step #46. Define Area Load for Level 3: An area load equal to 1.0kN/m^2 along the vertical direction will be applied only to the two middle openings. Switch to Level 3 by clicking on the icon  **Higher Level** until “Level 3” is displayed.

Click on the icon  **Area Load** in the Loads panel of the ribbon.



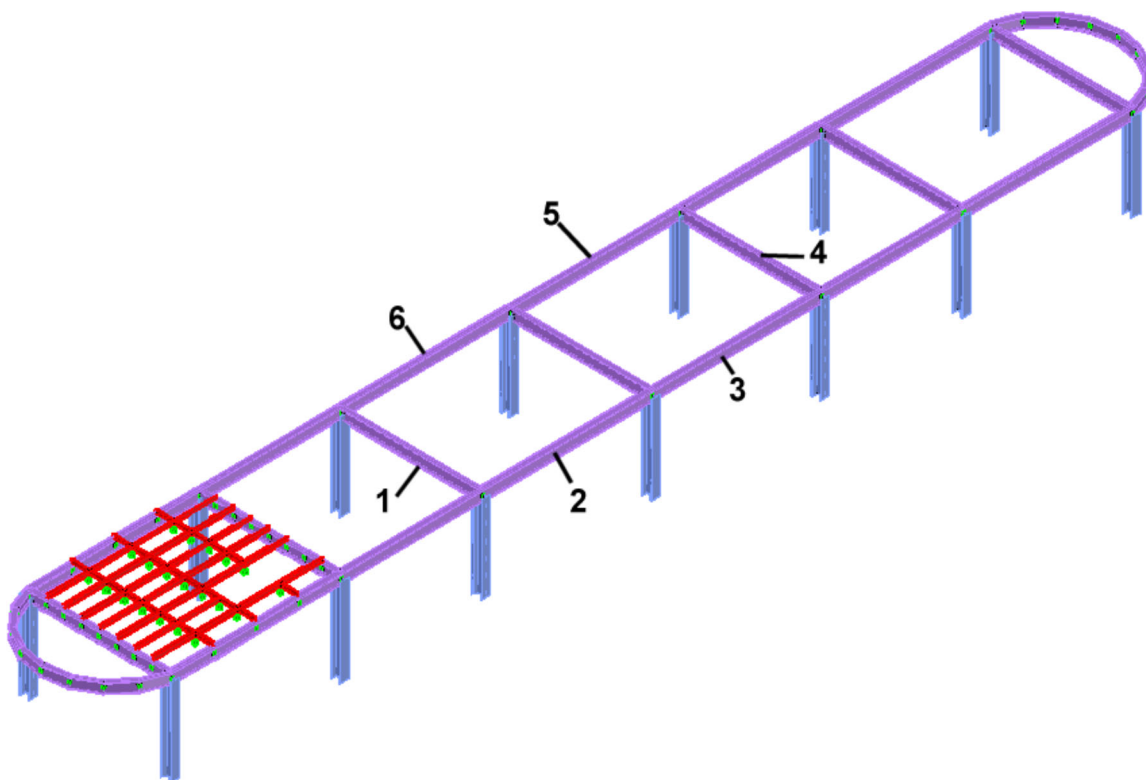
The dialog box is titled "Area Load" and contains the following sections:

- Generate:**
 - Name:
 - Description:
 - Buttons: Create New, Save / Modify, Delete
- Load - Direction:**
 - Load Value:
 - Global Direction Perpendicular to the Loading Plane:
 - ☐ X
 - ☐ Y
 - ☒ Z
 - Plane Tolerance:
- Elevation:**
 - Plane Perpendicular at:
 - ☒ Value (coordinate)
 - ☐ Joint
- Distribution:**
 - ☒ Two way ☐ X
 - ☐ One way ☐ Y
 - ☐ Custom X Y
- Advanced Features:**
 - Define Outline Region >>
 - Exclude Area >>
 - Ignore Members >>
- Buttons:** Display >>, Clear, OK, Cancel

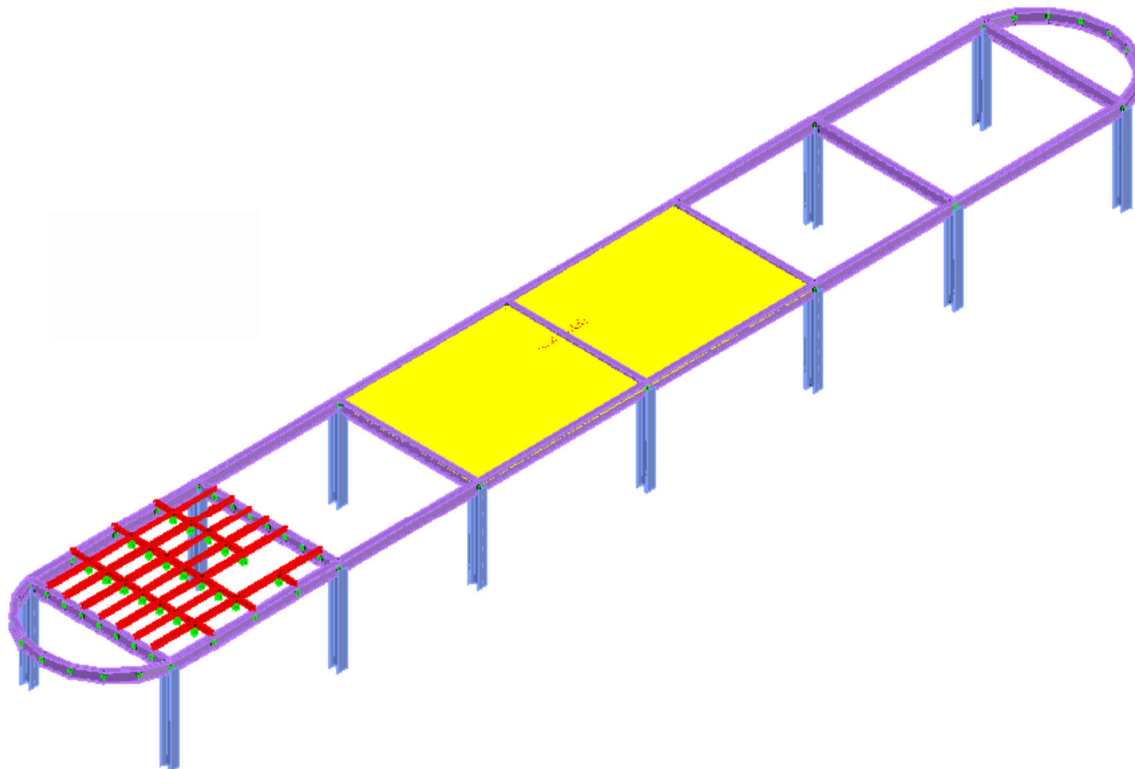
Type:

- AL3 as Name
- *Area Load Level 3* as Description
- 1.0 as Load Value
- Z as Global Direction
- 10.0 as Elevation (the elevation of level 3)
- Two way as Distribution

Press “Define Outline Region >>” and you are prompted to select the members that define the outline region of the area load. Click on the 6 members at the perimeter of the two middle openings, as shown at the image below.



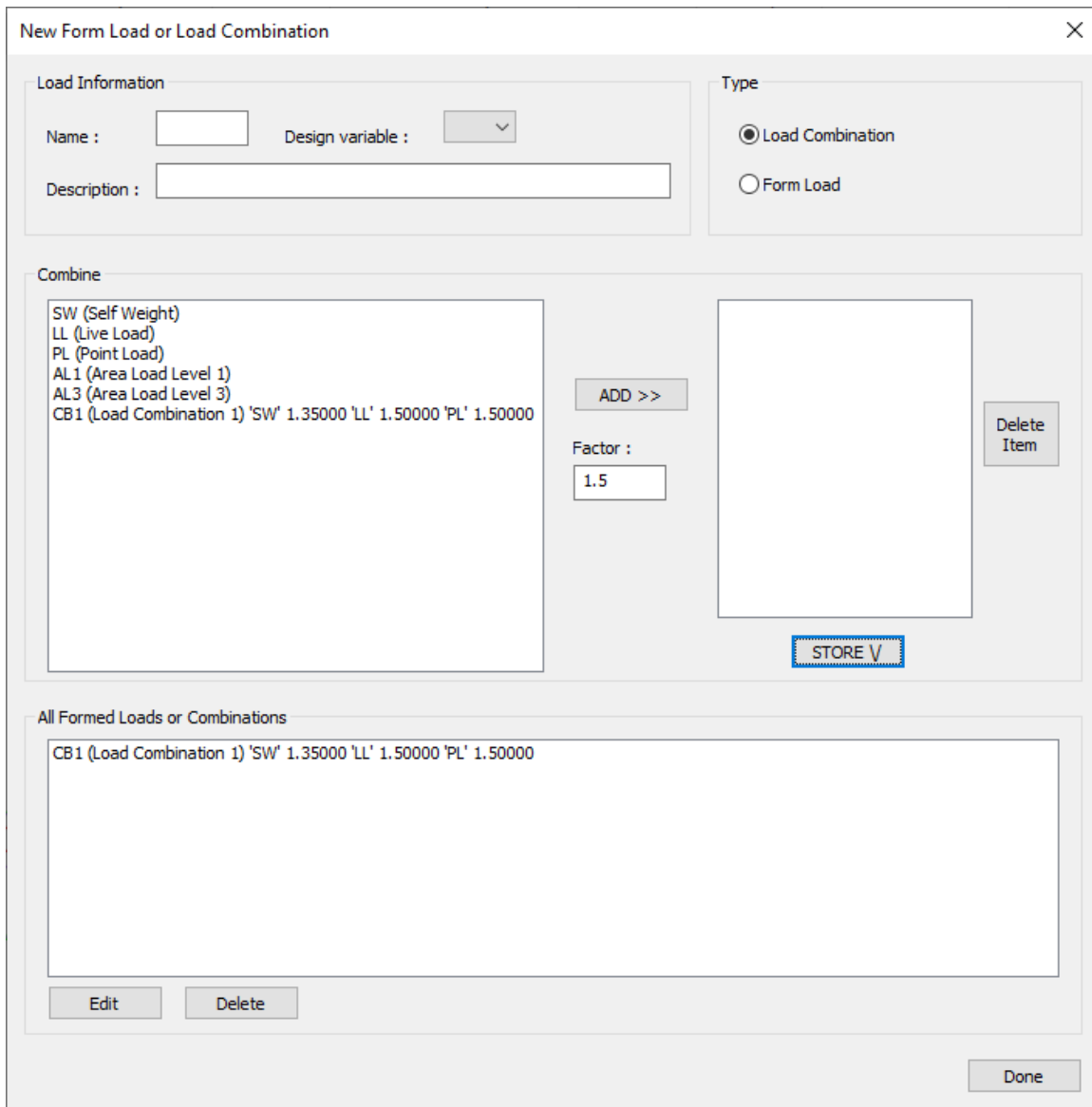
Press Display >> and the loaded area will be displayed in yellow solid hatch, as shown below.



Press Clear to remove the solid hatch pattern and then OK to store the area load AL3.

Step #47. Define Load Combinations: Click on the icon  **Combinations** and the Load Combination dialog appears. Enter:

- *CB1* as Name
- *Load Combination 1* for the Description of the Load Combination
- Click on SW, Enter 1.35 as the factor and press ADD>>
- Click on LL, Enter 1.5 as the factor and press ADD>>
- Click on PL, Enter 1.5 as the factor and press ADD>>
- Press Store
- Press Done to close the dialog.



New Form Load or Load Combination [X]

Load Information

Name : Design variable :

Description :

Type

☒ Load Combination
☐ Form Load

Combine

SW (Self Weight)
LL (Live Load)
PL (Point Load)
AL1 (Area Load Level 1)
AL3 (Area Load Level 3)
CB1 (Load Combination 1) 'SW' 1.35000 'LL' 1.50000 'PL' 1.50000

ADD >>

Factor :

Delete Item

STORE V

All Formed Loads or Combinations

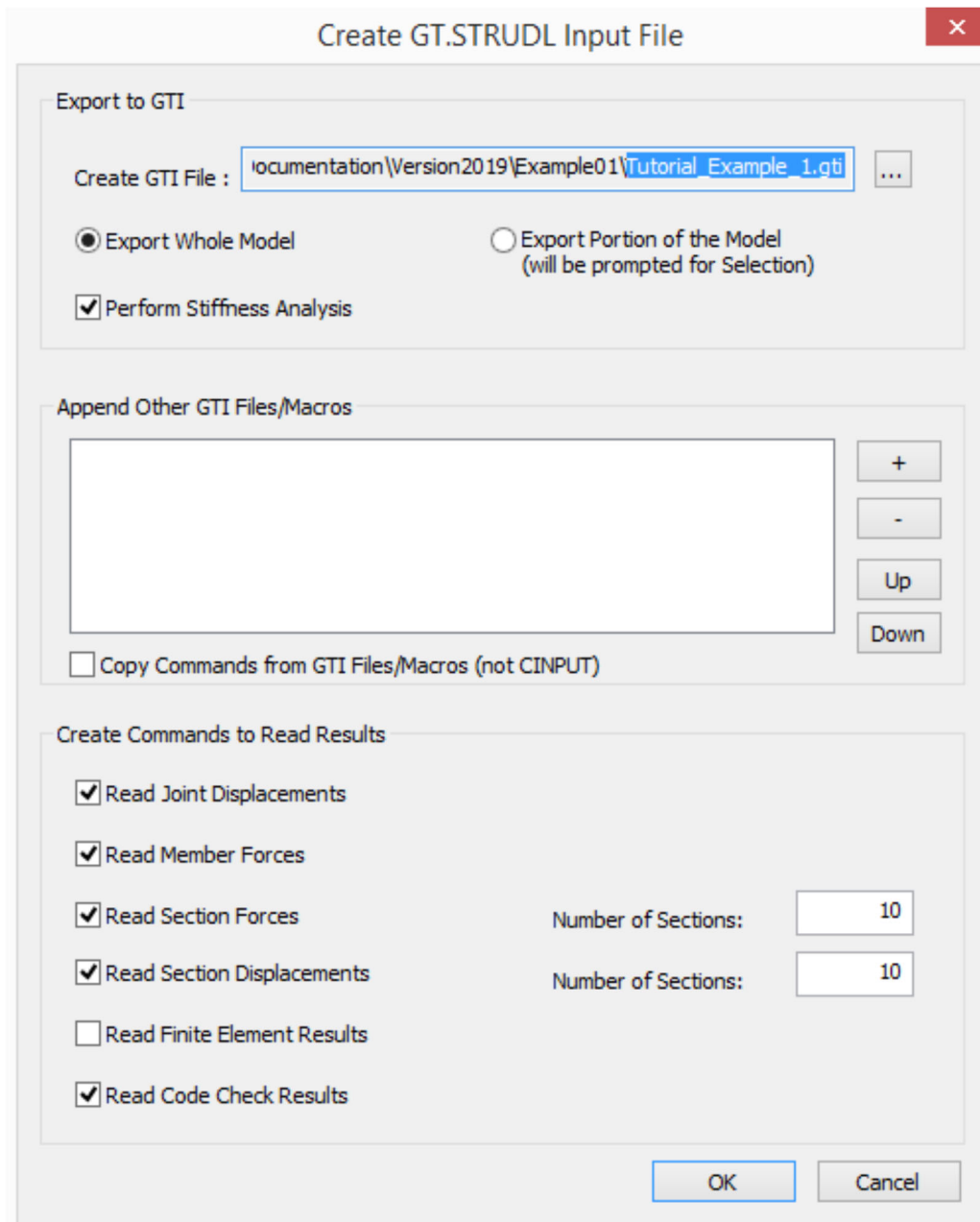
CB1 (Load Combination 1) 'SW' 1.35000 'LL' 1.50000 'PL' 1.50000

Edit Delete

Done

3.14. GT STRUDL Input File

Step #48. Create GTI: Save your model as “Tutorial_Example_1.dwg” and click on the icon  and the Create GT STRUDL Input file dialog appears. Keep the default GTI filename, check all options except “Read Finite Element Results” as shown in the following image and press OK.



Create GT.STRUDL Input File

Export to GTI

Create GTI File : documentation\Version2019\Example01\Tutorial_Example_1.gti ...

☒ Export Whole Model ☐ Export Portion of the Model
(will be prompted for Selection)

☒ Perform Stiffness Analysis

Append Other GTI Files/Macros

☐ Copy Commands from GTI Files/Macros (not CINPOT)

Create Commands to Read Results

☒ Read Joint Displacements

☒ Read Member Forces

☒ Read Section Forces Number of Sections: 10

☒ Read Section Displacements Number of Sections: 10

☐ Read Finite Element Results

☒ Read Code Check Results

OK Cancel

Step #49. View/Edit GTI: Click on the icon  Edit GTI and the GTI file created in the previous step will be opened by the system’s default text editor.

Step #50. Execute GT STRUDL: Click on the icon  and the GTI file created in the previous step will be sent to GT STRUDL main program that is waiting in the background.

Stiffness analysis is automatically performed and DBX result files are automatically created.

In order to demonstrate the use of the command that reads back design results, add the following commands into GT STRUDL main window (note that the steel design Parameters are incomplete)

```
PARAMETERS
```

```
CODE EC3 ALL MEMBERS
```

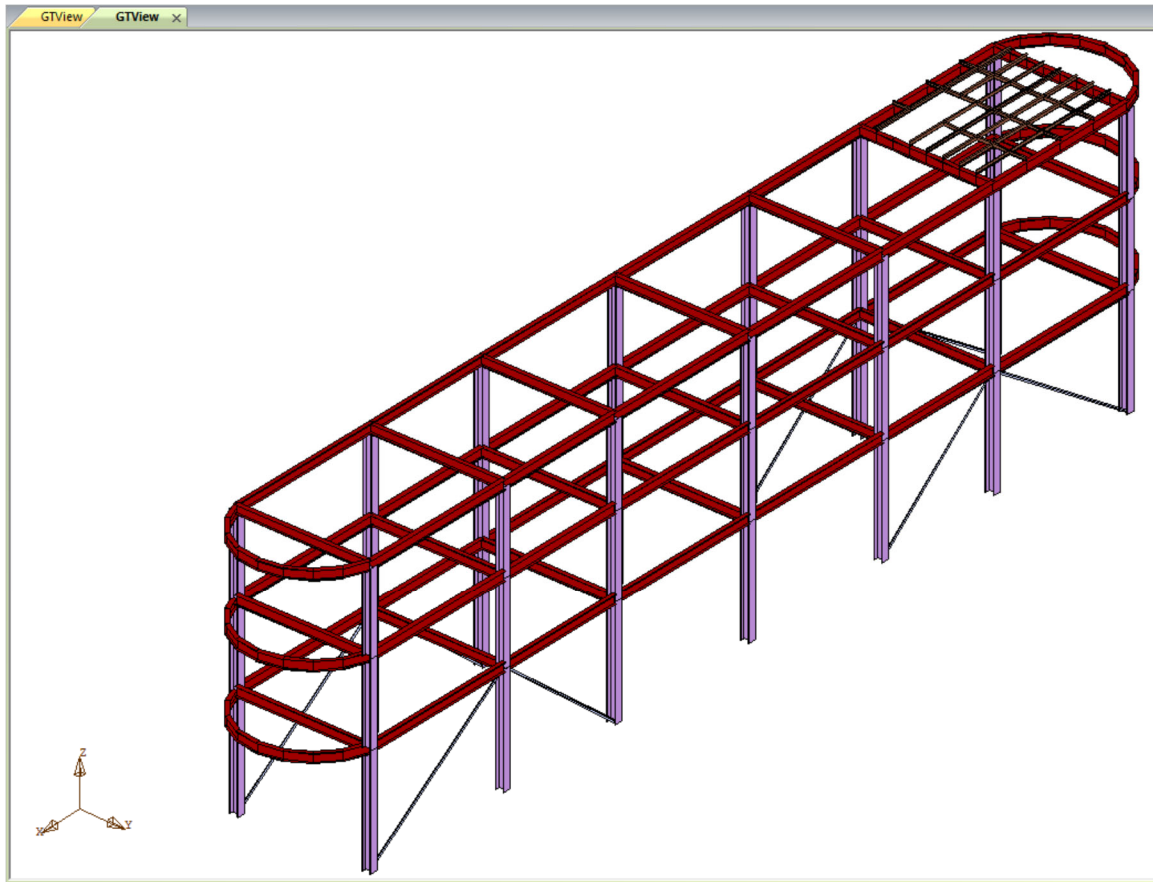
```
CHECK ALL MEMBERS AS BEAM
```

```
DBX BINARY 'Tutorial_Example_1.25' REPLACE
```

```
WRITE REPLACE CODE 'Tutorial_Example_1.25' MEMBERS EXISTING
```

The result of the CHECK ALL MEMBERS command shown above is that members 148 149
150 151 152 153 154 155 FAILED CODE CHECKS

In addition, you can enter GTMenu to view the model and the results as described in the GTMenu User Guide. After selecting the View button and checking Z-Up, and then selecting View 1, the structure is displayed in GTMenu as shown below.



Step #51. Read Results from GT STRUDL: In CAD Modeler, click on the icon  **Read Results** and the Read GT.STRUDL Results dialog appears. Check all options except “Read Finite Element Results” as shown below and press OK.

Read GT.STRUDL Results

Read DBX Results

GTI Directory : D:\Examples\GTSTRUDL\Example01

☒ Read Joint Displacements

☒ Read Member Forces

☒ Read Section Forces Number of Sections: 10

☒ Read Section Displacements Number of Sections: 10

☐ Read Finite Element Results

Type of element result : <ALL>

Surface : <ALL>

☒ Read Code Check Results

GTI Commands

Corresponding GT STRUDL commands:

```

DBX BINARY
WRITE REPLACE JOINT RESULTS JOINTS EXISTING
WRITE REPLACE MEMBER RESULTS MEMBERS EXISTING
WRITE REPLACE SECTION FORCES NS 10 AUTOMATIC MEMBERS EXISTING
WRITE REPLACE SECTION DISPLACEMENTS GLOBAL NS 10 MEMBERS EXISTING
WRITE REPLACE CODE MEMBERS EXISTING

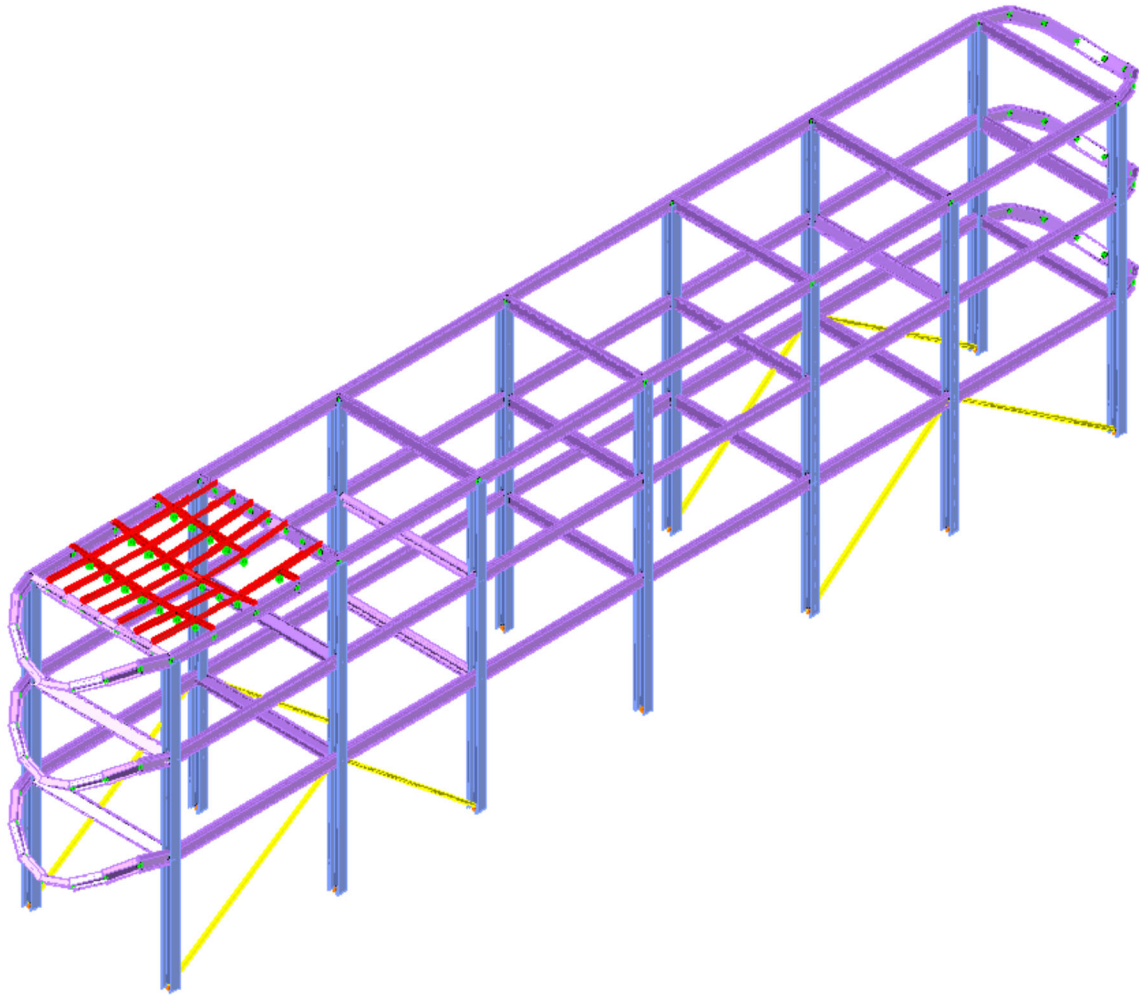
```

OK Cancel

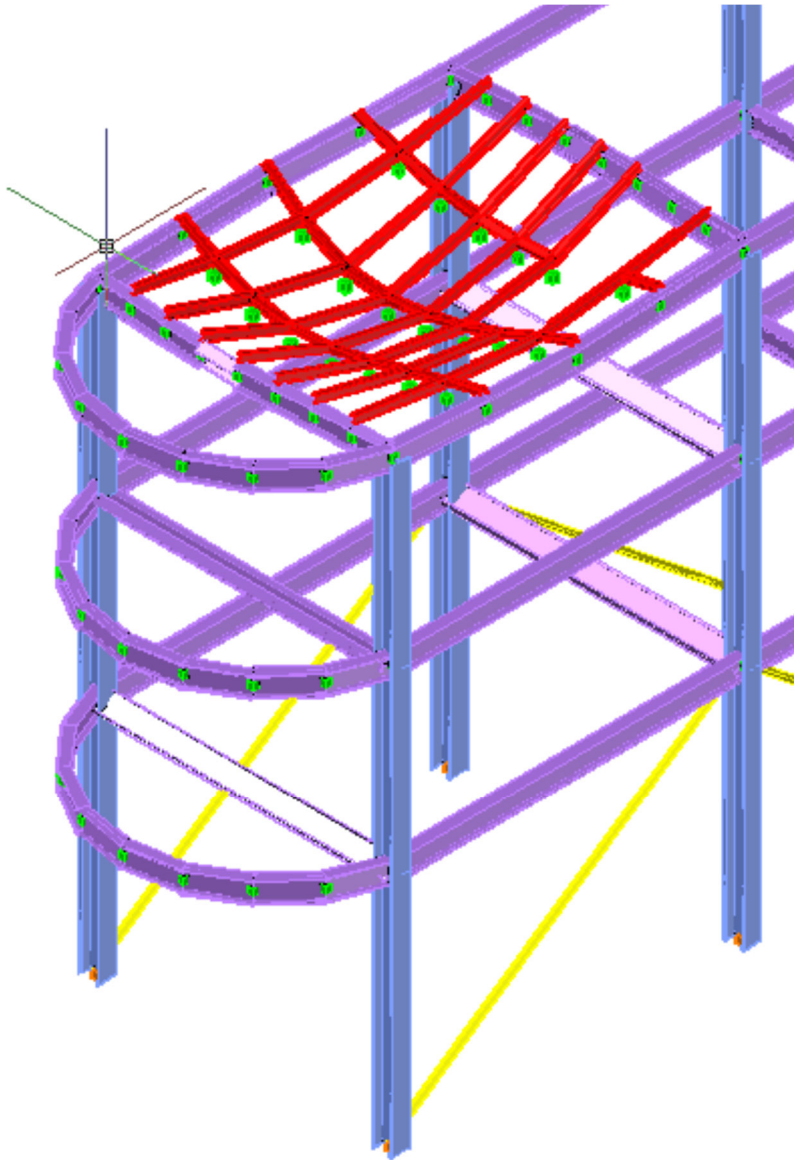
Note: You should get an information message “Results Loaded Successfully”

3.15. Display Results

Step #52. Display Deformed Model: on the Menu Bar, click on  Deformed (ribbon tab “GTS Display”) and then select SW as load Case and press ENTER twice. The deformed structure will be drawn as shown below.



Repeat and select PL as the Load Case and the deformed structure appears as shown in the following image:



Note: You can Annotate the Joint displacements by clicking on the  Annotate Displacements icon, that is located under the  Deformed in the ribbon and the selecting a joint and annotation position.

Click on  Undeformed (ribbon tab "GTS Display") to return to the original undeformed position of the model.

Step #53. Display Section Displacements: Click on the icon  Frame to switch back to the wireframe view. Click on  Displacements (ribbon tab "GTS Display").

Select:
- SW as Load Case

Displacements

Load Case / Load Combination

SW \$ Self Weight

Display Options

Scale Factor (Values) :
0.100

Font Size (pt) :
10.00

Annotation Format:
Exponential

Decimal Places :
2

☒ Hide Model

Display >>

Annotate >
Legend >

Animation Options

Frames :
7

Animation Speed % :
1

Generate Animation Frames
Animate >>

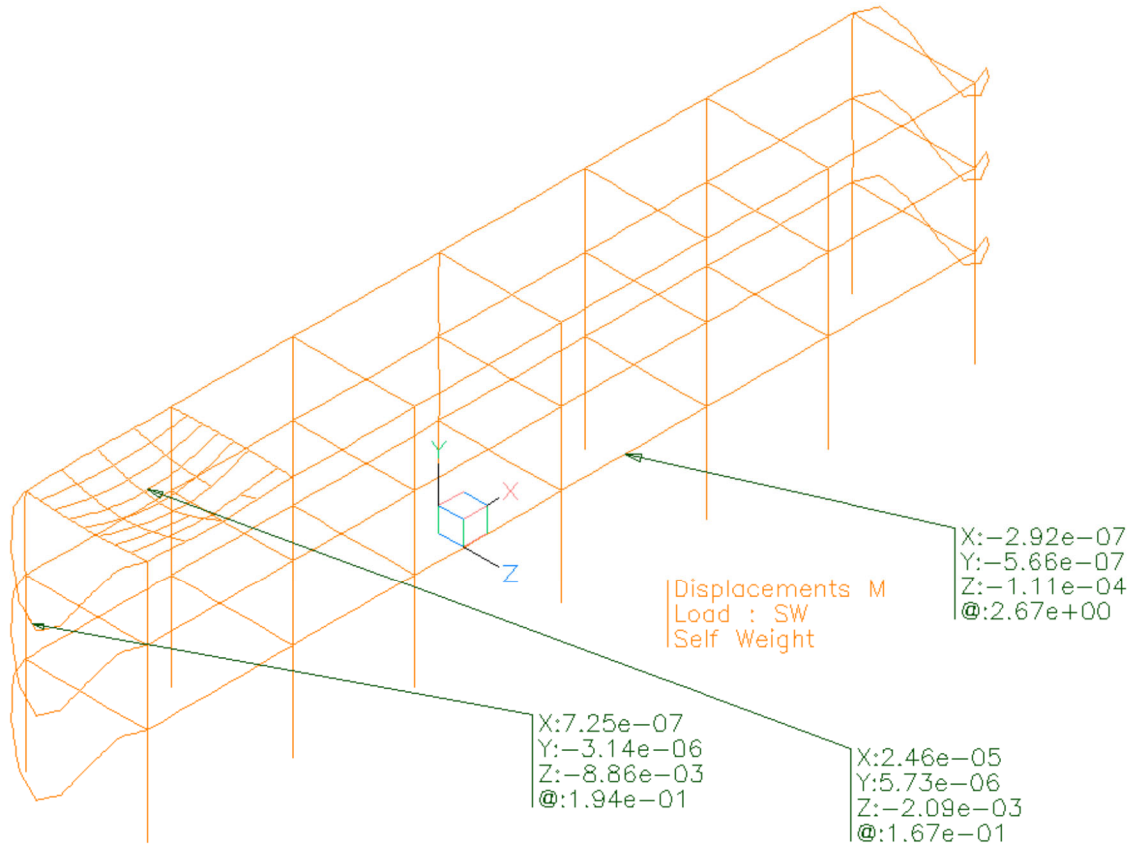
Clear
Close

- 0.1 as Scale Factor
- 10.00 as Font Size (default)
- Annotation Format: Exponential
- Check Hide Model

Press “Display >>” and zoom at the upper left end of the structure as shown in the image below.

Press “Annotate >” and click on any part of the deformed shape curve and then at the position that you want the annotation to be displayed.

Press on “Legend >” and click at any part of the screen to place the legend of the diagram.



In order to display the animation press “Generate Animation Frames” and then “Animate >>”.
To terminate the animation press “Stop” button.

In order to exit the command, uncheck “Hide Model”, press “Clear” Button and “Close”.

Step #54. Display Member Diagrams: Click on  **Diagrams** (ribbon tab “GTS Display”).

Member Diagrams [X]

Load Case / Load Combination
 SW \$ Self Weight ▼

☐ Envelope

Values
 MZ Moment ▼

Display Options

Scale Factor (Values) 0.100

Font Size (pt) 10.00

Annotation Format: Decimal ▼

Decimal Places : 2 ▼

☐ Label Max & Min ☐ Positive Sign

Display >>

Annotate > Legend >

Clear Close

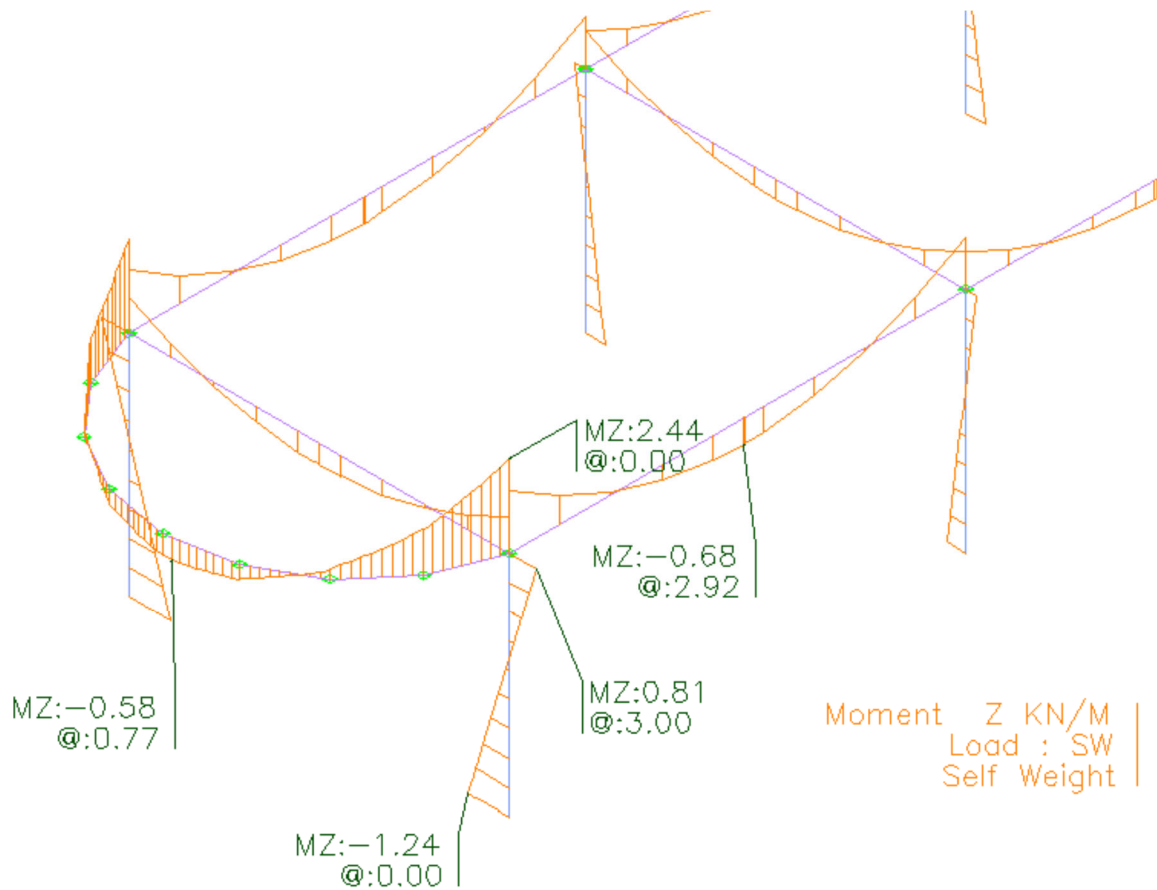
Select:

- SW as Load Case
- MZ Moment as Value to be displayed
- 0.1 as Scale Factor
- 10.00 as Font Size (default)

Press “Display >>” and zoom at the upper left end of the structure as shown in the image below.

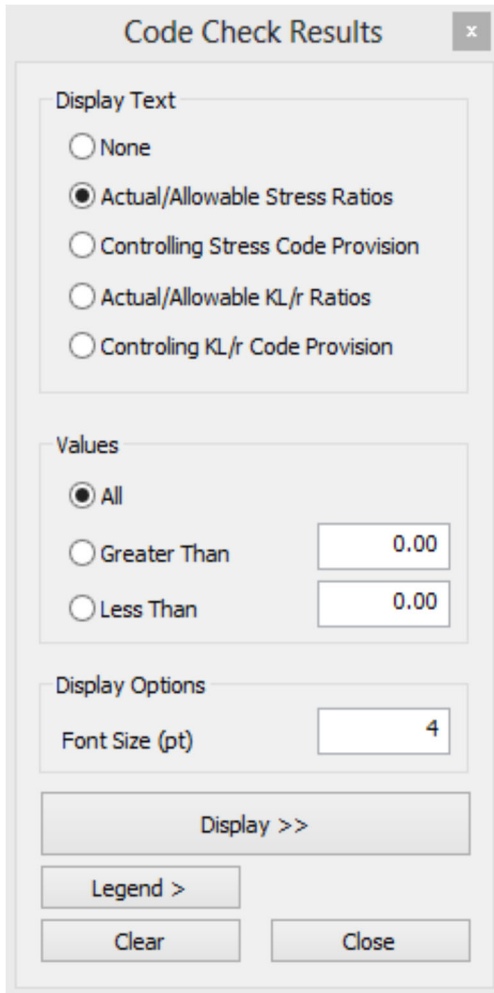
Press “Annotate >” and click on any part of the yellow MZ Moment curve and then at the position that you want the annotation to be displayed.

Press on “Legend >” and click at any part of the screen to place the legend of the diagram.



In order to exit the command, press the “Clear” Button and then the “Close” button.

Step #55. Display Code Check Results: Click on  **Code Check** (ribbon tab “GTS Display”) and select all mebers by typing ALL and pressing <Enter> twice.



The image shows a software dialog box titled "Code Check Results". It contains several sections for configuring the display of code check results. The "Display Text" section has five radio button options: "None", "Actual/Allowable Stress Ratios" (which is selected), "Controlling Stress Code Provision", "Actual/Allowable KL/r Ratios", and "Controlling KL/r Code Provision". The "Values" section has three radio button options: "All" (selected), "Greater Than" (with an adjacent input field containing "0.00"), and "Less Than" (with an adjacent input field containing "0.00"). The "Display Options" section has a label "Font Size (pt)" and an adjacent input field containing "4". At the bottom of the dialog are four buttons: "Display >>", "Legend >", "Clear", and "Close".

Select:

- *Display Text: Actual/Allowable Stress Ratios*
- *All Values*
- *4 as Font Size*

Press "Display >>" and the following image is drawn with the bracing members that failed the check (149 to 155) appearing in red and members that passed the check appearing in blue.

In order to exit the command, click the "Clear" Button and "Close".

GTSTRUDL - Joint Displacement Datasheet, display Units: Meters Degrees

File Edit Columns Filter Sort Units Help

Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y
1	SW	0.0000	0.0000	0.0000	-0.0002199	0.0000249
2	SW	0.0000	0.0000	-0.0000	0.0004223	0.0000347
3	SW	0.0000	0.0000	0.0000	-0.0002020	0.0001908
4	SW	0.0000	0.0000	-0.0000	0.0004311	-0.0003191
5	SW	0.0000	0.0000	0.0000	-0.0001981	-0.0001509
6	SW	0.0000	0.0000	-0.0000	0.0004329	0.0003413
7	SW	0.0000	0.0000	0.0000	-0.0003753	-0.0003670
8	SW	0.0000	-0.0000	-0.0000	0.0009405	0.0007478
9	SW	0.0000	0.0000	0.0000	0.0001757	-0.0000570
10	SW	-0.0000	0.0000	-0.0000	-0.0004446	-0.0000466
11	SW	0.0000	0.0000	0.0000	0.0001936	0.0001102
12	SW	-0.0000	-0.0000	-0.0000	-0.0004358	-0.0004006
13	SW	0.0000	0.0000	0.0000	0.0001975	-0.0002277
14	SW	-0.0000	-0.0000	-0.0000	-0.0004339	0.0002608
15	SW	0.0000	0.0000	0.0000	0.0004615	-0.0004304
16	SW	-0.0000	-0.0000	-0.0000	-0.0008937	0.0006344
18	SW	-0.0000	-0.0000	-0.0005	-0.5759841	-0.1616357
19	SW	-0.0000	-0.0000	-0.0046	-0.3254073	0.0838498
20	SW	-0.0000	-0.0000	-0.0095	-0.0569121	0.5516294
21	SW	-0.0000	-0.0000	-0.0117	-0.0000572	0.7820065
22	SW	-0.0000	-0.0000	-0.0095	0.0567759	0.5517620
23	SW	-0.0000	-0.0000	-0.0046	0.3252377	0.0840486
24	SW	-0.0000	-0.0000	-0.0005	0.5758479	-0.1614586
26	SW	0.0000	0.0000	0.0000	-0.0004927	-0.0005444
27	SW	0.0000	0.0000	-0.0000	0.0002704	0.0004355
28	SW	0.0000	0.0000	0.0000	-0.0007150	0.0001680
29	SW	0.0000	0.0000	-0.0000	0.0001630	-0.0002131
30	SW	0.0000	0.0000	0.0000	0.0003613	0.0003782

Click on the icon  **Reactions** and the “GTSTRUDL – Reactions Datasheet” dialog appears where you can filter, sort, write results to text file or change results units as shown in figure below.

GTSTRUDL - Joint Reactions Datasheet, display Units: Meters KiloNewtons

File Edit Columns Filter Sort Units Help

Joint	Load	Force X	Force Y	Force Z	Moment X	Moment Y
1	SW	0.000	-0.086	24.790	-0.000	-0.000
3	SW	0.206	-0.085	24.199	-0.000	-0.000
5	SW	0.020	-0.085	23.735	-0.000	-0.000
7	SW	-0.162	-0.177	25.863	0.000	0.000
9	SW	0.000	0.093	24.777	0.000	0.000
11	SW	0.267	0.095	24.261	0.000	0.000
13	SW	0.020	0.095	23.717	-0.000	-0.000
15	SW	-0.103	0.192	25.788	0.000	0.000
26	SW	-0.316	-0.103	24.309	0.000	0.000
28	SW	-0.015	-0.118	26.112	0.000	0.000
30	SW	0.148	-0.140	27.442	-0.000	-0.000
32	SW	-0.247	0.067	24.146	-0.000	-0.000
34	SW	-0.014	0.064	25.249	-0.000	-0.000
36	SW	0.195	0.227	27.381	-0.000	-0.000
1	LL	0.000	-0.881	128.774	-0.000	-0.000
3	LL	1.222	-0.881	120.588	-0.000	-0.000
5	LL	0.200	-0.881	116.237	-0.000	-0.000
7	LL	-0.476	-1.865	137.305	-0.000	-0.000
9	LL	0.000	0.881	128.774	0.000	0.000
11	LL	1.223	0.881	120.589	-0.000	-0.000
13	LL	0.200	0.881	116.237	0.000	0.000
15	LL	-0.475	1.865	137.303	0.000	0.000

Click on the icon Results Datasheets >  Code Check and the GT STRUDL Code Check Results" dialog appears

GT STRUDL Code Check Results										
File Edit Columns Filter Sort Not Checked Help										
Member	P/F	Load	Section Loc	Crit Ratio	Crit Prov	Stress Ratio	Stress Prov	KL/r Ratio	KL/r Prov	Properti
148	FAIL	CB1	7.211	3.395	KL/z	0.766	S.5.1	3.395	KL/z	60x60x5
149	FAIL	CB1	6.403	3.015	KL/z	0.981	S.5.1	3.015	KL/z	60x60x5
150	FAIL	CB1	6.403	3.015	KL/z	0.977	S.5.1	3.015	KL/z	60x60x5
151	FAIL	CB1	7.211	3.395	KL/z	0.751	S.5.1	3.395	KL/z	60x60x5
152	FAIL	CB1	7.211	3.395	KL/z	0.768	S.5.1	3.395	KL/z	60x60x5
153	FAIL	CB1	6.403	3.015	KL/z	0.992	S.5.1	3.015	KL/z	60x60x5
154	FAIL	CB1	6.403	3.015	KL/z	0.973	S.5.1	3.015	KL/z	60x60x5
155	FAIL	CB1	7.211	3.395	KL/z	0.755	S.5.1	3.395	KL/z	60x60x5
1	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
2	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
3	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
4	Pass	CB1	4.000	0.293	KL/z	0.134	S.5.4(2)	0.293	KL/z	HE320B
5	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
6	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
7	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
8	Pass	CB1	4.000	0.293	KL/z	0.134	S.5.4(2)	0.293	KL/z	HE320B
9	Pass	CB1	0.000	0.939	KL/z	0.391	S.5.4(2)	0.939	KL/z	IPE330
10	Pass	CB1	0.000	0.782	KL/z	0.243	S.5.4(2)	0.782	KL/z	IPE330
11	Pass	CB1	6.000	0.939	KL/z	0.439	S.5.4(2)	0.939	KL/z	IPE330
12	Pass	CB1	0.000	0.939	KL/z	0.391	S.5.4(2)	0.939	KL/z	IPE330
13	Pass	CB1	0.000	0.782	KL/z	0.243	S.5.4(2)	0.782	KL/z	IPE330
14	Pass	CB1	6.000	0.939	KL/z	0.439	S.5.4(2)	0.939	KL/z	IPE330
15	Pass	CB1	0.000	0.782	KL/z	0.205	S.5.2	0.782	KL/z	IPE330
16	Pass	CB1	0.000	0.782	KL/z	0.205	S.5.2	0.782	KL/z	IPE330
17	Pass	CB1	5.000	0.205	S.5.2	0.205	S.5.2	0.000	*****	IPE330
18	Pass	CB1	5.000	0.188	S.5.2	0.188	S.5.2	0.000	*****	IPE330
19	Pass	CB1	0.000	0.249	S.5.3	0.249	S.5.3	0.000	*****	IPE330
20	Pass	CB1	0.871	0.127	S.7.7	0.127	S.7.7	0.000	*****	IPE330
21	Pass	CB1	0.871	0.127	S.7.7	0.127	S.7.7	0.000	*****	IPE330
22	Pass	CB1	0.871	0.127	S.7.7	0.127	S.7.7	0.000	*****	IPE330
23	Pass	CB1	0.871	0.127	S.7.7	0.127	S.7.7	0.000	*****	IPE330
24	Pass	CB1	0.871	0.127	S.7.7	0.127	S.7.7	0.000	*****	IPE330
25	Pass	CB1	0.871	0.127	S.7.7	0.127	S.7.7	0.000	*****	IPE330
26	Pass	CB1	0.871	0.249	S.5.3	0.249	S.5.3	0.000	*****	IPE330
27	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
28	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
29	Pass	CB1	4.000	0.293	KL/z	0.135	S.5.4(2)	0.293	KL/z	HE320B
30	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B
31	Pass	CB1	4.000	0.293	KL/z	0.102	S.7.7	0.293	KL/z	HE320B

3.17. Report Builder



Report
Builder

Step #57. Click on the icon  to generate the report of this structure by calling Report Builder. Report Builder is launched in a separate window. You can find more information on how creating your report in the GT STRUDL® Report Builder Getting Started Guide (Help Icon on the right top corner on Report Builders' window).

Home

Open File Generate Overall Report Image Text RTF Insert Remove Header and Footer Print Preview Print

Joint Name: ALL Loading: ALL
Member Name: ALL Level: ALL
Element Name: ALL Surface: ALL
Filters

Apply Filters to Checked Items Format Options Help

Style

GTSTRUDL Reports

- ☐ Contents
- ☒ Model Data
 - ☒ Groups
 - ☒ Joint Coordinates
 - ☒ Joint Support Restraints
 - ☒ Member Incidences
 - ☒ Member Properties
 - ☒ Member/Element Constants
 - ☐ Element Incidences
 - ☐ Element Properties
- ☒ Load Data
 - ☒ **Summary of Loadings**
 - ☒ Loading Combinations
 - ☒ Joint Loads
 - ☒ Member Loads
 - ☐ Element Loads
- ☐ Analysis Results
 - ☐ Joint Displacements
 - ☐ Support Joint Reactions
 - ☐ Member Forces
 - ☐ Section Forces
 - ☐ Average Element Results
 - ☐ Member Results Graphs
- ☐ Design of Steel Members

Load Data

Summary of Loadings

Length: M, Force: KN, Angle: DEG, Temperature: DEGC, Time: SEC

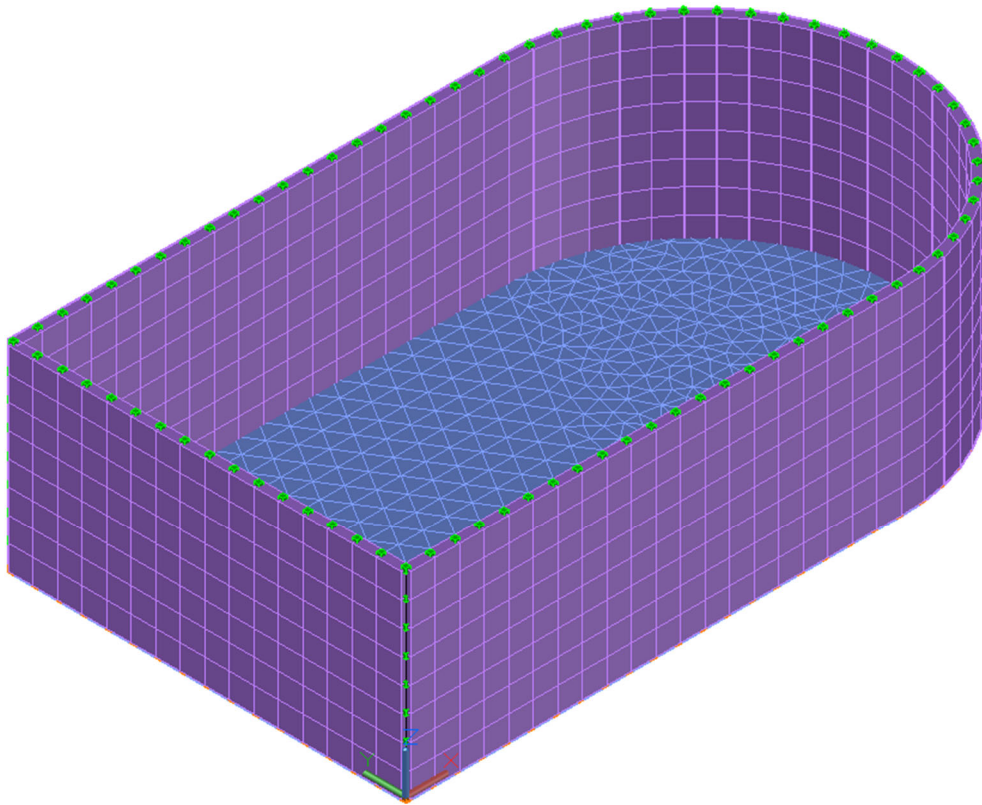
Name	Description
SW	Self Weight
LL	Live Load
PL	Point Load
AL1	Area Load Level 1
AL3	Area Load Level 3
CB1	Load Combination 1

Ready CAP NUM SCRI

4. Tutorial Example #2

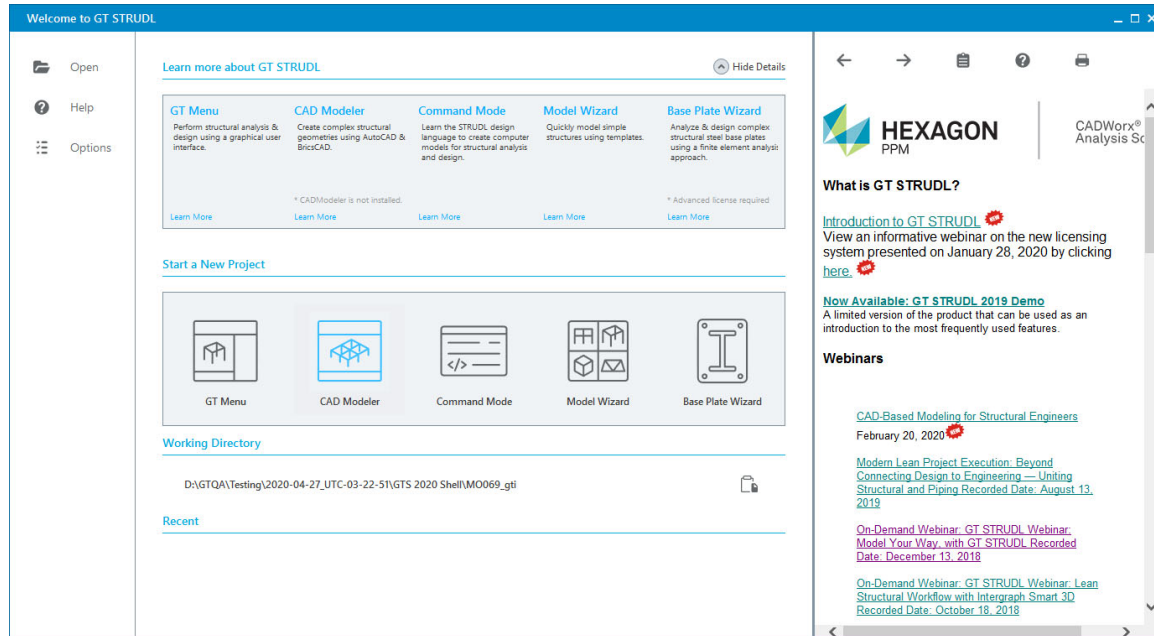
4.1. Introduction

The modeling and analysis of the tank shown below is demonstrated in a step-by-step process using CAD Modeler and GT STRUDL finite element analysis.



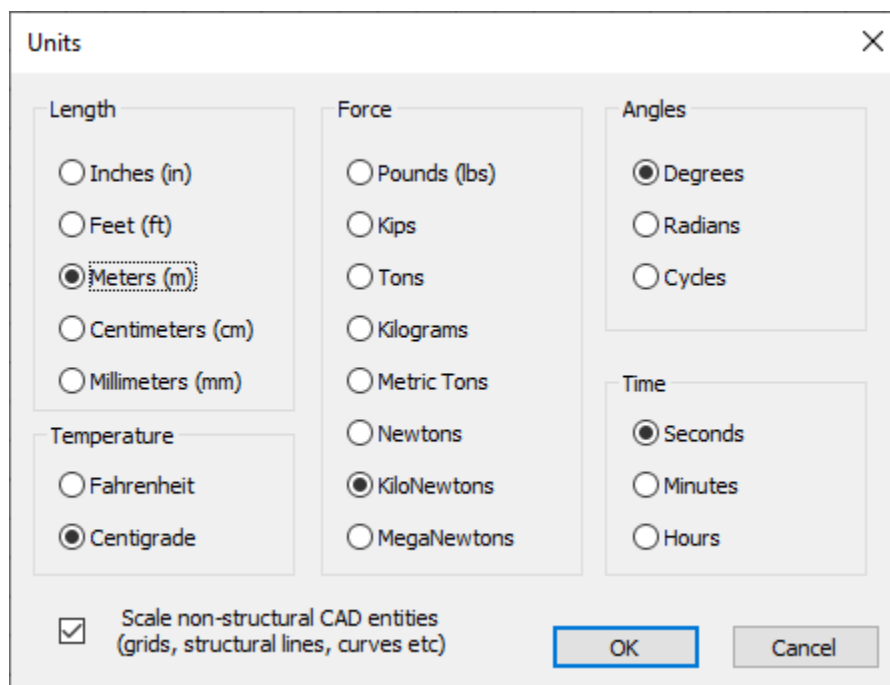
4.2. Open CAD Modeler and start working

Step #1. Launch GT STRUDL by selecting the icon “CAD Modeler” in the Welcome to GT STRUDL dialog shown below. The version of AutoCAD/BricsCAD selected during the installation will be automatically launched, together with CAD Modeler’s menus and ribbon bar.



4.3. Define the basic geometry of the model

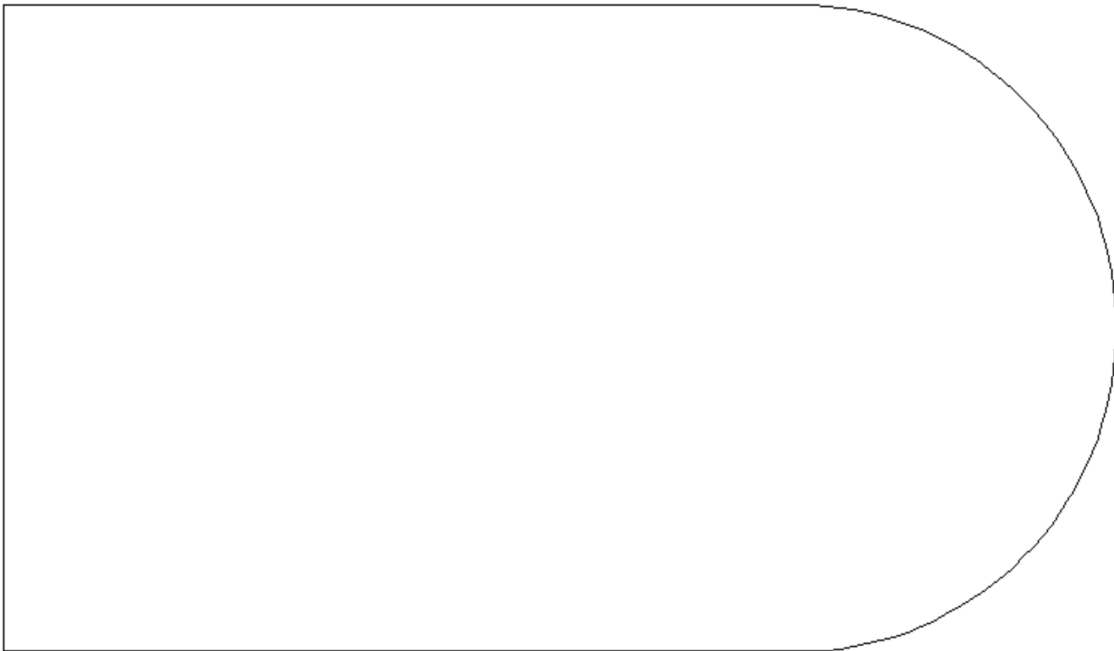
Step #2. Define the correct Units by pressing the icon  **Units** and select *Meters (m)* and *KiloNewtons* in the *Units Form*.



Step #3. Create an AutoCAD/BricsCAD Polyline that will describe the outline of the tank. Type the following commands at the command prompt (each command is followed by an <ENTER>):

```
PLINE  
0,0,0  
@10,0  
ARC  
@4,4  
@-4,4  
LINE  
@-10,0  
CLOSE
```

The polyline shown in the picture below is created after entering Zoom and Extents at the command prompt.



Step #4. Create a Line along the height of the tank: Switch to the isometric view of the structure by pressing the small house icon in AutoCAD's Cube or BricsCAD's Chair.



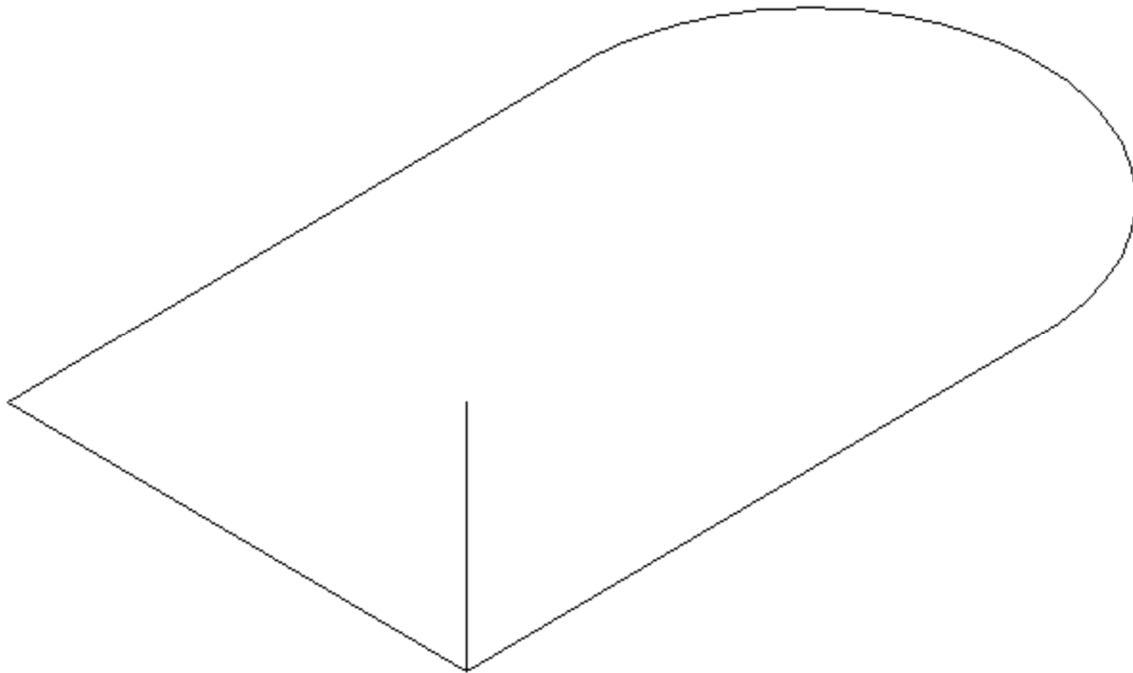
Type the following commands at the command prompt (each command is followed by an <ENTER>):

```
LINE
0,0,0
@0,0,4
<ENTER>
```



The line shown at the picture below is created.

Click on the TOP icon of the AutoCAD's Cube / BricsCAD's Chair in order to switch back to floor plan view.



4.4. Create the bottom of the tank

Step #5. Generate the Finite Elements inside the polyline, at the bottom of the Tank: Click on the icon  **2D Area**, under the “2D” Drop Button, located in Meshing at Ribbon Area, and

when the prompt message *Poly Select Boundary Polyline or Circle* appears, click on the Polyline that you have created in the previous step.

Select Mesh Properties

Generate

Material Concrete

Element Attributes

Type SBHT6 Thickness 0.20

Mesh Geometry

External Boundary obj-581

External Boundary Edge Size 0.000000

Maximum Element Edge Size 0.50

Minimum Element Edge Size 0.000000

Mesh Quality High

Internal Boundaries

Internal Joints

Add + Remove -

Multi+ Multi -

Add Remove

Spacing Extrude Direction

Uniform 4

Variable

Defined by Curve, Size: 3.242418

Sweep Function

Labeling

More >>

Preview Clear Create Close

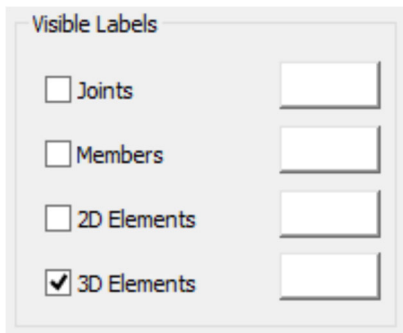
The Select Mesh Properties form appears where you have to enter:

- Material : *Concrete*
- Type: *SBHT6*, meaning triangular elements having 6 degrees of freedom per node
- Thickness: *0.20*
- Boundary Maximum Element Edge Size: *0.50*
- Mesh Quality: *High*

You can press the Preview button to see the finite elements as they will be generated.

Press the Create button to create the finite elements and joints on the bottom of the tank.

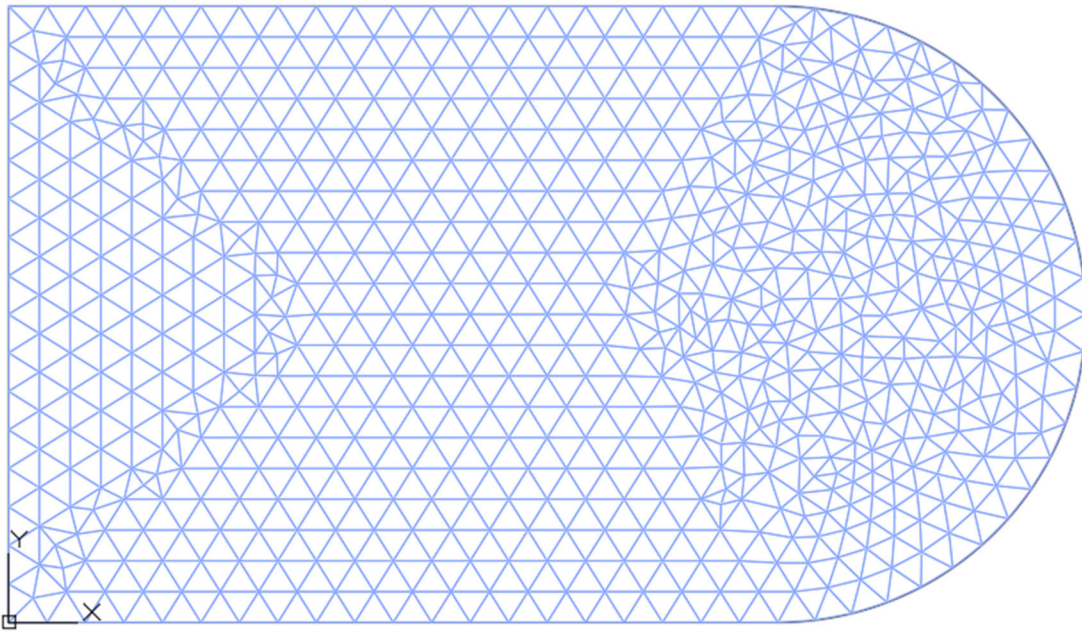
Step #6. Turn OFF labeling and view mesh:



Click on the icon  **Options** in the ribbon bar and then uncheck the Visible Labels option for Joints, Members and 2D Elements.

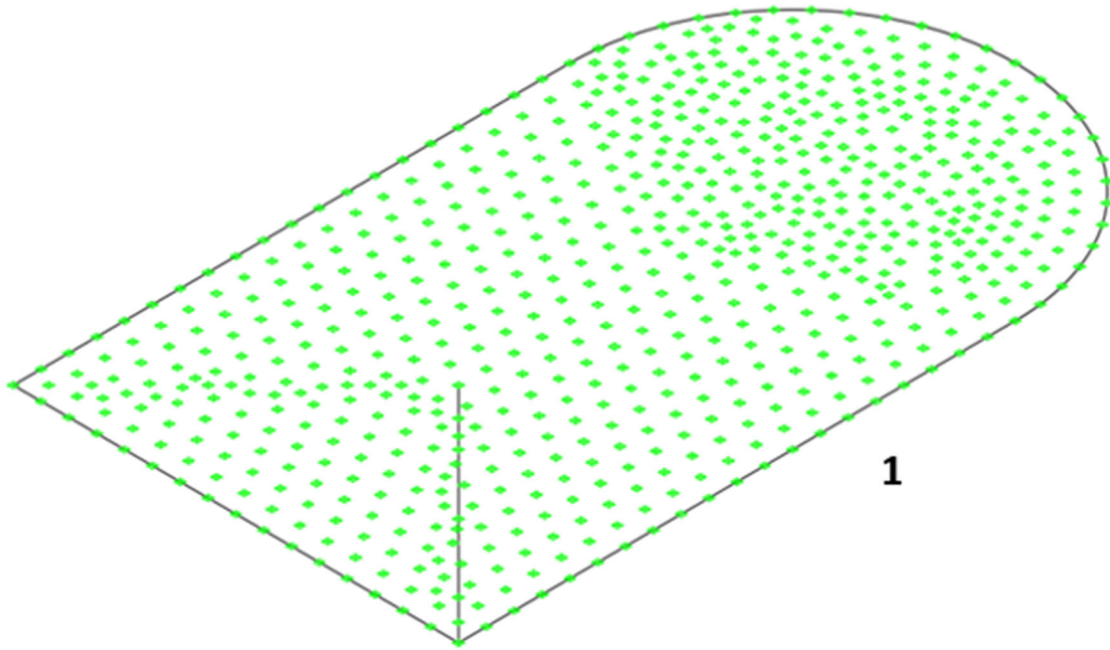
Now labeling is turned off and it is easier and faster to control the model.

The finite element mesh, without labeling, is shown in the image below.

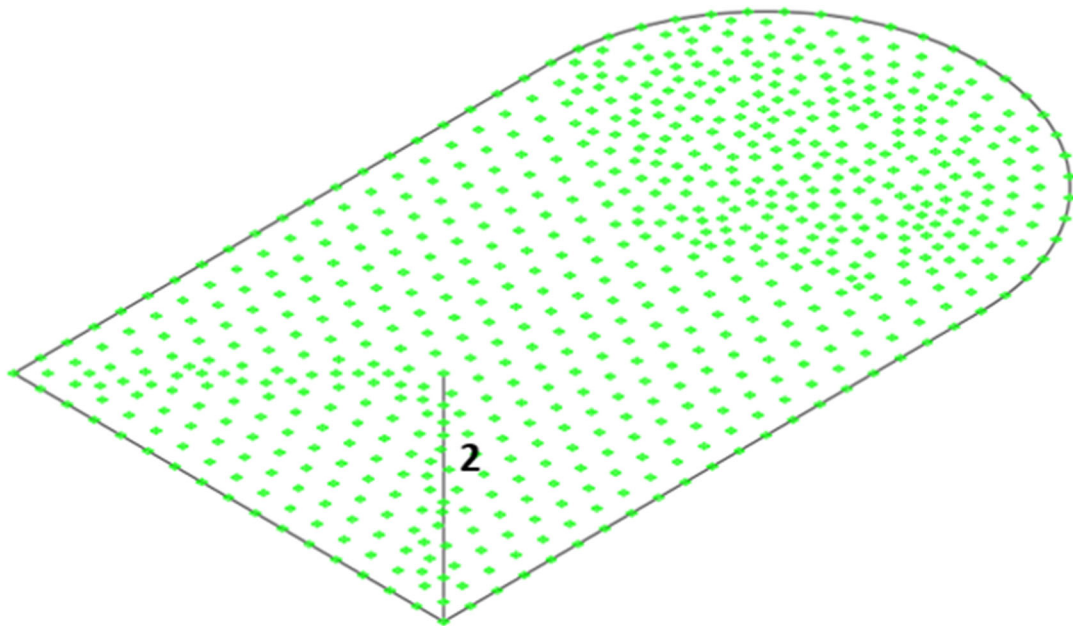


4.5. Create the walls of the tank

Step #7. Generate the finite elements that will model the Wall of the Tank by extruding the polyline: Switch to Isometric view and click on the icon  **3D Extrude**, under the "2D" Drop Button, located in Meshing at Ribbon Area (Note: the display of joints and elements previously created is automatically turned off to make selection of the polyline and extrude line easier). When the prompt message *Select Line, Arc, Circle or PolyLine to be Extruded* appears, click on the Polyline that you have created in a previous step, as shown in the following picture (Click #1).



When the prompt message *Select Extrude Direction Curve (Line or Arc)* appears, click on the line that you have created in a previous step, as shown in the picture below (Click #2).



Select Mesh Properties

Generate

Material
Concrete

Element Attributes

Type
SBHQ6
Thickness
0.20

Mesh Geometry

External Boundary
obj-581

External Boundary Edge Size
0.000000

Maximum Element Edge Size
0.50

Minimum Element Edge Size
0.000000

Mesh Quality
Very Low

Internal Boundaries

Add +
Remove -
Multi+
Multi -

Internal Joints

Add
Remove

Spacing Extrude Direction

☒ Uniform
8

☐ Variable
...

☐ Defined by Curve, Size:
3.242418

☐ Sweep Function

Labeling

More >>

Preview
Clear
Create
Close

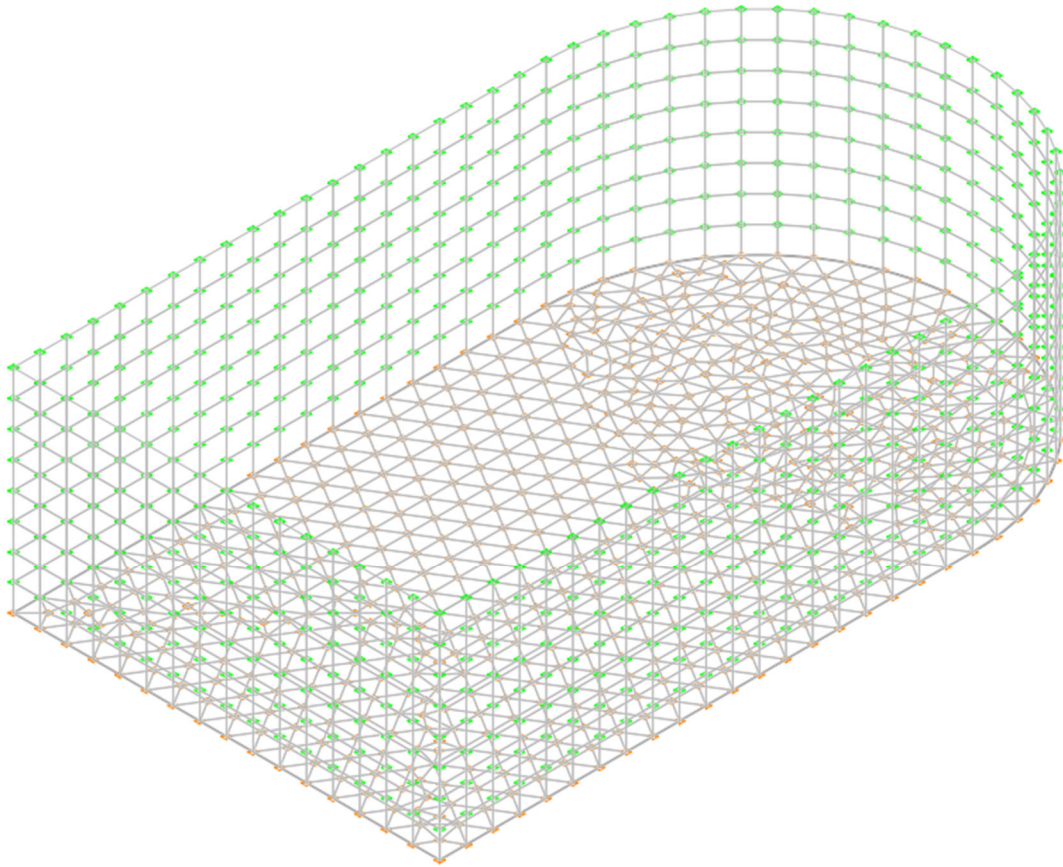
The Select Mesh Properties form appears where you have to enter:

- Material : *Concrete*
- Type: *SBHQ6*, meaning quad elements having 6 degrees of freedom per node
- Thickness: *0.2*
- Maximum Element Edge Size: *0.5*
(Note: This must be the same as when the mesh in the bottom of the tank was created so the mesh will be the same along the polyline)
- Spacing Extrude Direction: *Uniform* and select 8 spaces in the pulldown

You can press the Preview button to see the finite elements as they will be generated.

Press the Create button to generate the finite elements and joints on the wall of the tank

The finite element mesh is presented in the following image.



Step #8. Check for duplicate joints: Since both meshing functions described above generated joints along the polyline, pairs of joints having the same coordinates exist in the model and they have to be merged together. In order to check for joints having the same coordinates, click on the icon  Joints Duplicates, under the “Check” Drop Button, located in Find/Change/Check at Ribbon Area.

For the *Merge Tolerance* <0.001000>, just press <ENTER> to accept the default value.

The Duplicate Joints form appears where you can see the list of joints having the same coordinates. Make sure that Merge option is checked for all joint pairs and press OK.

×

Merge Joints

List

Joint	Duplicate	Merge
1	281	☒
1	363	☒
2	282	☒
3	283	☒
4	284	☒
5	285	☒
6	286	☒
7	287	☒
8	288	☒
9	289	☒
10	290	☒
11	291	☒
12	292	☒
13	293	☒
14	294	☒
15	295	☒
16	296	☒
17	297	☒
18	298	☒
19	299	☒
20	300	☒
21	301	☒
22	302	☒
23	303	☒
24	304	☒
25	305	☒
26	306	☒

Select All

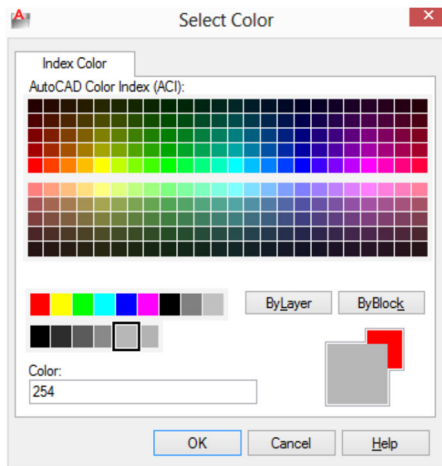
Unselect All

OK

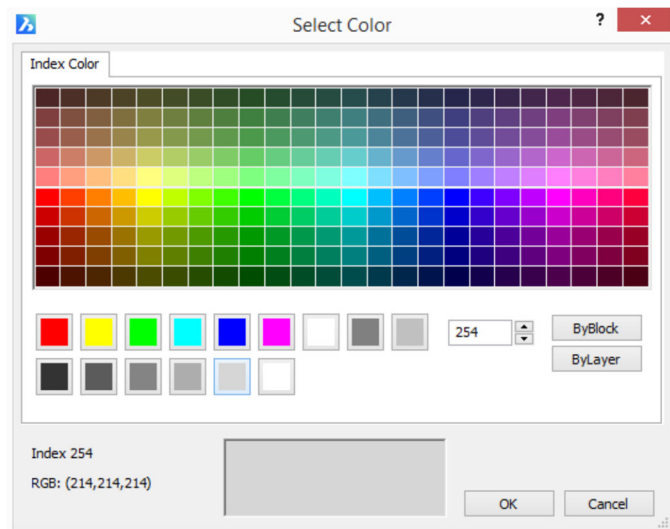
Cancel

By entering the same command again for the 2nd time, you should get the notification that *0 duplicate joints found* .

Step #9. Switch to 3D View: Press the house icon (AutoCAD cube) or Top Front Left (BricsCAD chair) to change the view to Isometric, and type Z and E (Zoom, Extents). Click on the icon **Options** to set a different color for 2D finite elements. When the Display Options form appears, click on the white button next to “2D Element”, to define a different color. Using this form you can also define Object Sizes in the current length units.



Select Color (AutoCAD)



Select Color (BricsCAD)

Press OK to close the Color Options Dialog. The elements will now have the color that you selected.

Press the icon  (and then type *Shade*) to display the 3D solid view of the model, replacing the wireframe view:

Display Options

×

Visible Objects

☒ Joints

☒ Members

☒ 2D Elements

☒ 3D Elements

Visible Labels

☐ Joints

☐ Members

☐ 2D Elements

☒ 3D Elements

Label Settings - Font Sizes

Joints :

0.250000

Members :

0.250000

2D Elements :

0.250000

3D Elements :

0.250000

Annotation (pts) :

10.000000

Annotation Format:

Decimal

Decimal Places :

2

Object Sizes

Joint :

0.050000

Load Arrowhead (pts):

10.00000

Display Members / Elements

Shrink Factor :

1.0

☐ Do Not Display Thickness in 3D

Members As:

Analytical

Scale Factors

Concentrated Load (pts) :

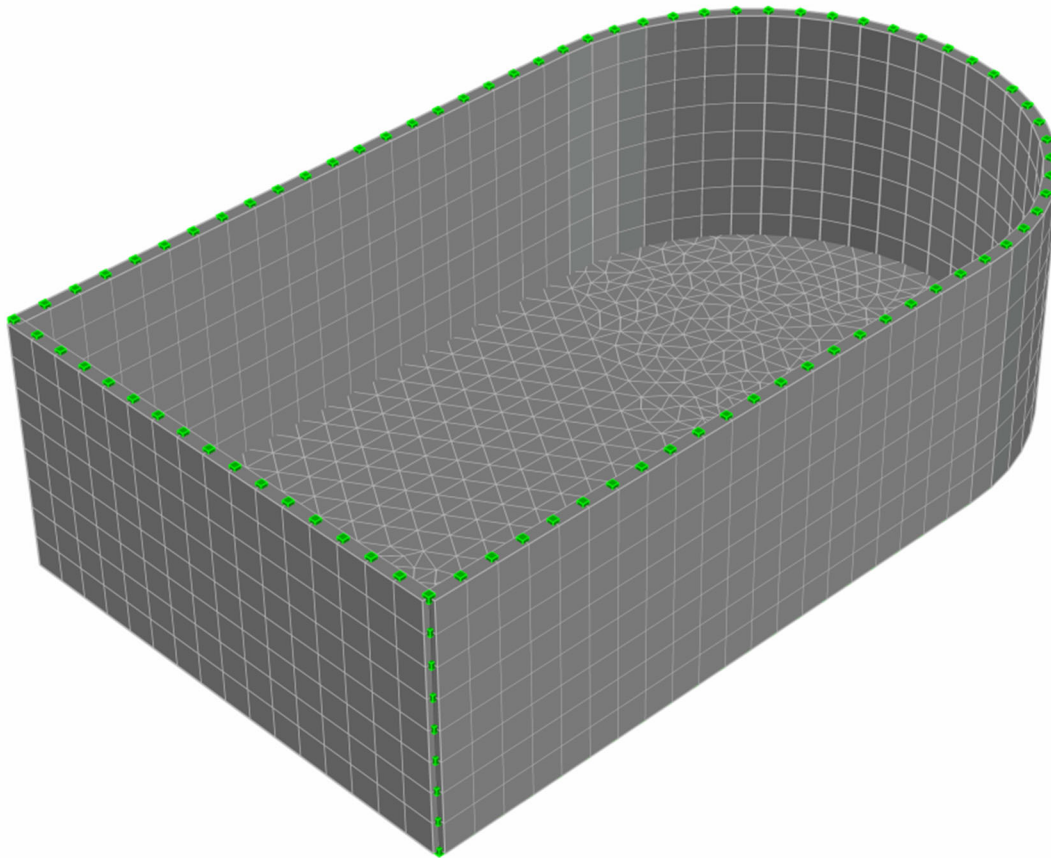
72.000000

Distributed Load (pts) :

72.000000

OK

Cancel



Press the icon  **Frame** to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

Step #10. Save your Model: In order to save your model, use AutoCAD's/BricsCAD's save command and store the DWG using any filename that you want.

4.6. Create Supports

Step #11. Support the joints at the base of the model:

Switch to the FRONT View, by clicking on Front on AutoCAD's cube or BricsCAD's chair.

Click on the icon  **Support** and select the window by clicking at points 1 and 2 in the following image. All the bottom joints are selected and press ENTER to finish the selection.



Restraints & Spring values

Quick Selection : Pin ▼

Restraint	Spring	Restraint	Spring
☒ Fx		☐ Mx	
☒ Fy		☐ My	
☒ Fz		☐ Mz	

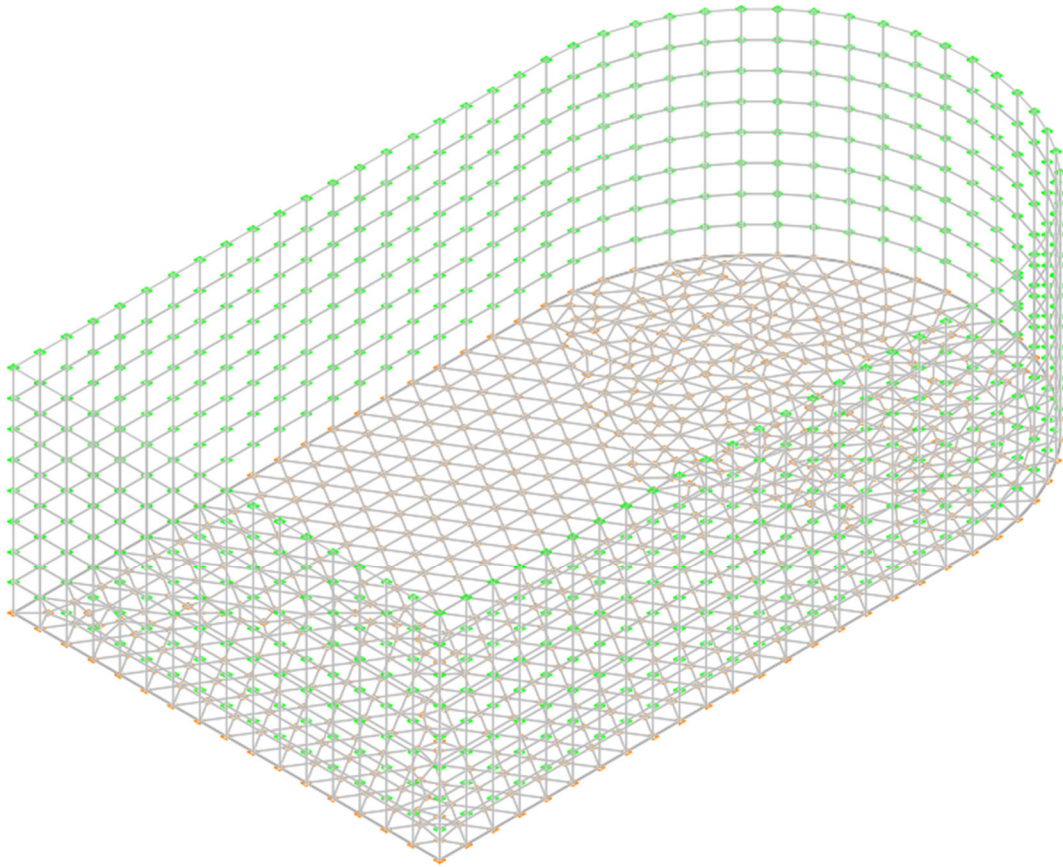
The Joint Properties [Multiple Selection] form appears.

Using the Quick Selection, select Pin and note that Fx, Fy and Fz are automatically checked.

Press OK.

Press the Isometric (Top Front Left) icon to change the view to Isometric, and type Z and E (Zoom, Extents).

All the bottom joints are now pinned and have an orange color instead of green to indicate that they are supported.



4.7. Check the model

Step #12. Check for duplicate joints: In order to check for joints having the same coordinates, click on the icon  **Joints Duplicates**, under the “Check” Drop Button, located in Find/Change/Check at Ribbon Area.

For the Merge Tolerance <0.001000>, just press <ENTER> to accept the default value.

You should get the notification that *0 duplicate joints found*.

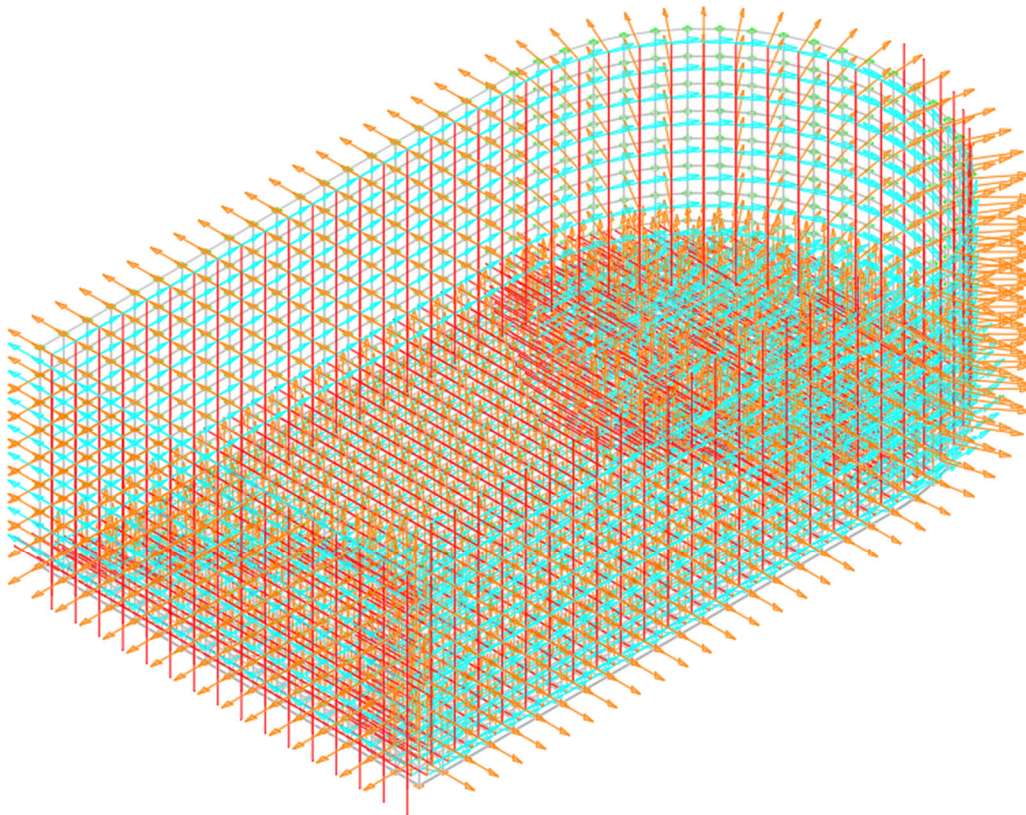
Step #13. Check for floating joints: In order to check for joints not connected to the model, click on the icon  **Joints Floatings**, under the “Check” Drop Button, located in Find/Change/Check at Ribbon Area. If your model was created as described so far, you should get a notification that *0 floating joints found*.

Note: You can also run all other checks of the same drop list, to check for Interference Joints, Duplicate Shells, Duplicate Names and Database Integrity. You should not get any errors or warnings.

Step #14. View Planar Axes for Finite Elements: In order to check the direction of the planar axes of the Finite elements, click on the icon  **Shell Planar Axes** in the “GTS Display” Ribbon area

and then *Enter Legend Coordinates(x,y,z)*: or click at the point where you want the legend to be displayed.

In the legend, the X axis is displayed in cyan, Y axis in red and Z axis in yellow (AutoCAD) or orange (BricsCAD). The size of the arrow and its arrowhead is controlled by the value given in Display Options > Object Sizes > Load Arrowhead and the size of the legend font is controlled by the value given in Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in [2.6.66](#)).



To clear the arrows select  **Clear** from the GTS Display Ribbon Tab.

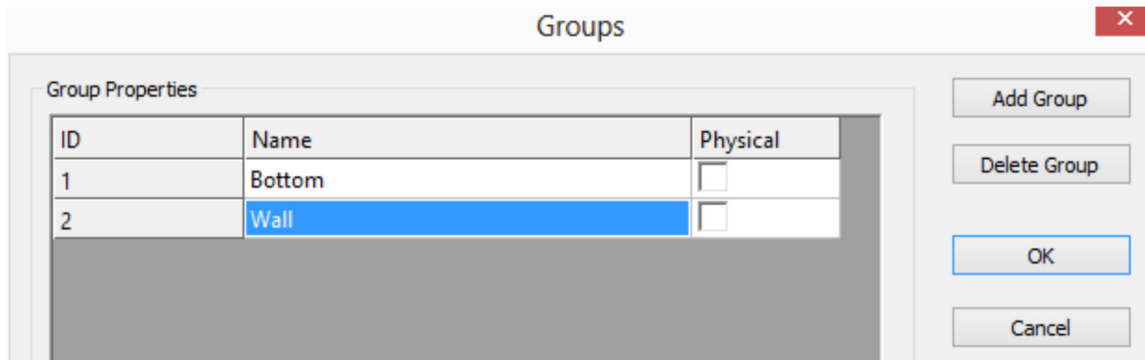
4.8. Define Groups

Step #15. Create Group Names: It is optional to define Groups in your model but it is strongly recommended to do so since it will be easier to control the display and selection for parts of your structure.



List

In the Groups panel of GTS CAD Modeler ribbon, click on the icon  and the Group dialog appears.



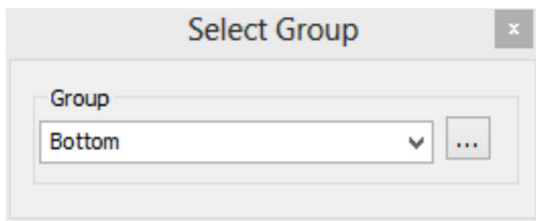
Press the *Add Group* button and enter `Bottom` as *Name* of the group.

Press the *Add Group* button and enter `Wall` as *Name* of the group.

Press OK to close the form.

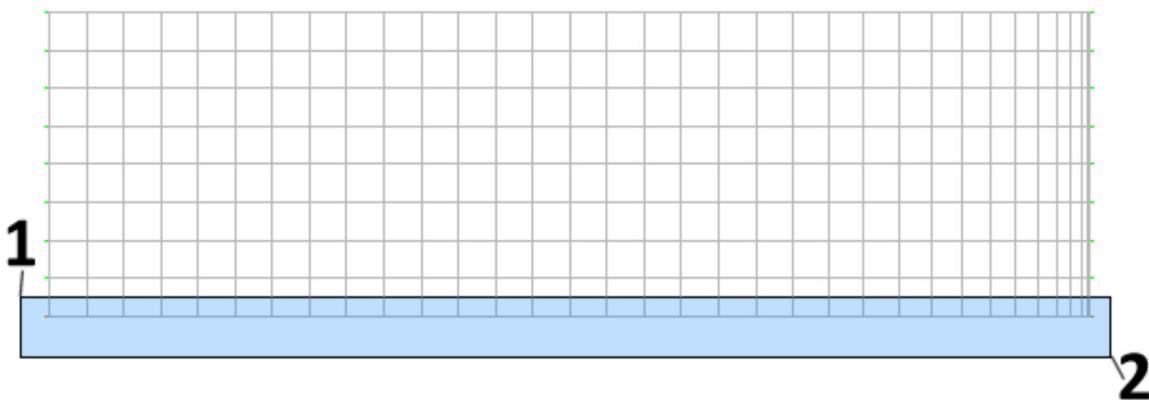
Step #16. Add the elements in the bottom of the tank to the Group Bottom:

Switch to the FRONT View, by clicking on Front on AutoCAD's cube or BricsCAD's chair.



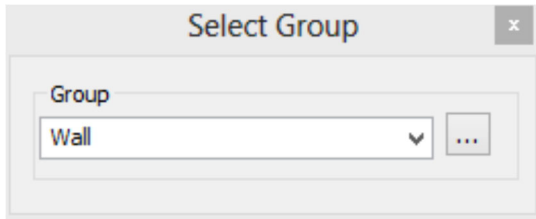
Click on the icon  in the Group panel and make sure that the Group Bottom is selected as the active group.

Make a selection by clicking at the points 1 and 2 of the following image and press the Enter key.



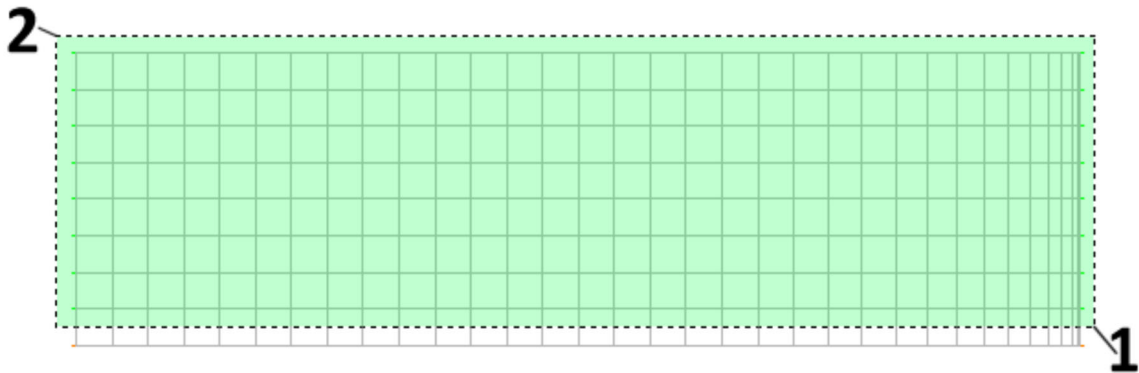
All the shell elements located in the bottom of the tank are now selected and added to group "Bottom".

Step #17. Add the elements in the tank wall to the Group Wall.



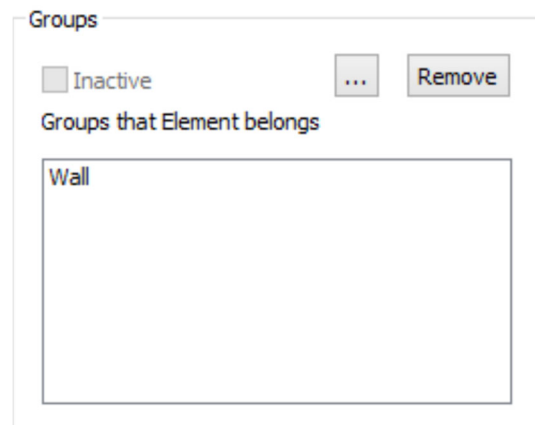
Click on the icon  +Shells and make sure that the Group Wall is selected as the active group.

Make a selection by clicking at the points 1 and 2 of the following image and press the Enter key.



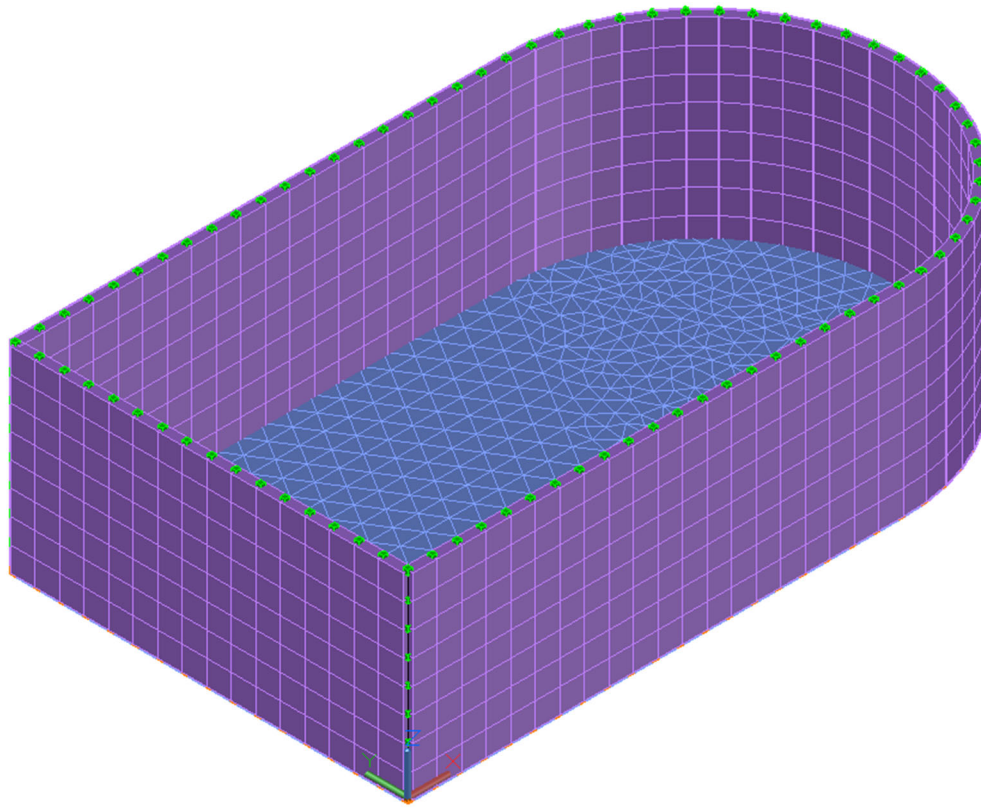
All shell elements located in the wall of the tank are now selected and added to group "Wall".

To verify that the group is applied correctly, you can double click on a element in the wall and in the Shell Properties form, you will be able to see its Group definition in the Shell Properties form.



Press the house icon  to change the view to Isometric, and type Z and E (Zoom, Extents).

Click at the icon  to display the 3D solid view as shown in the following image and save your model.

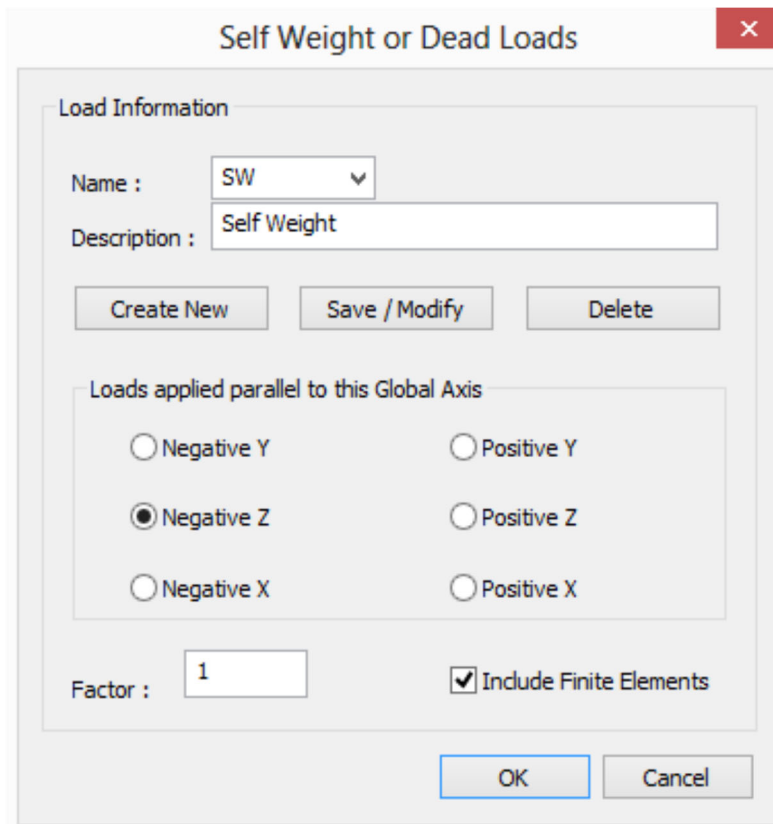


4.9. Define Loads

Step #18. Define Self Weight: Click on the icon  **Self Weight** and the Self Weight dialog appears. Enter:

- *SW* as Name
- *Self Weight* as the Load Description
- *Negative Z* as the Global Axis for the direction of the load
- 1.0 as Factor
- Check Include finite elements

and press OK to create the new loading and close the dialog.



Self Weight or Dead Loads

Load Information

Name :

Description :

Loads applied parallel to this Global Axis

☐ Negative Y
 ☐ Positive Y

☒ Negative Z
 ☐ Positive Z

☐ Negative X
 ☐ Positive X

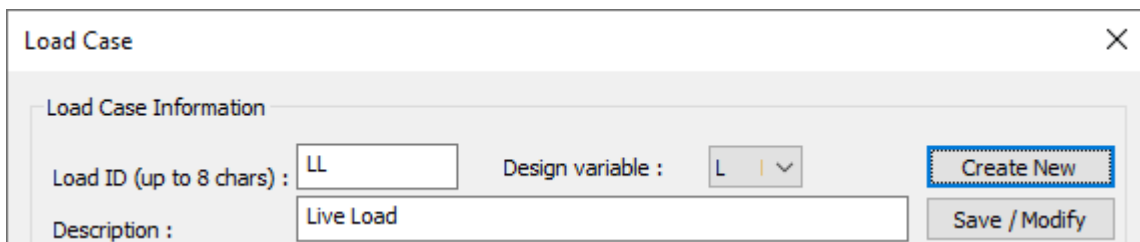
Factor : ☒ Include Finite Elements

Step #19. Define Load Cases: Click on the icon  **Load Cases** and the Load dialog appears.

Enter:

- *LL* as Name
- *Live Load* as the Load Description
- *L* as Design Variable

and press Create New.



Load Case

Load Case Information

Load ID (up to 8 chars) : Design variable :

Description :

Enter:

- *PL* as Name
- *Pressure Load* as the Load Description

and press Create New.

Load Case [X]

Load Case Information

Load ID (up to 8 chars) : Design variable :

Description :

Load ID List :

Press Exit to close the dialog.

Step #20. Apply Live Loads: Live Loads will be applied only to the bottom of the tank , therefore click on the icon  Colors . Select the 2nd Tab in order to colorize elements by their group and make only the Group Bottom visible and press OK.

Sections **Groups**

Categories

Groups	Color	Visible
Bottom	161	☒
Wall	191	☐
UnGrouped data	256	☐

Click on the icon  Shell in the Loads Panel. Using a full window, select all entities that appear on screen and press <ENTER> to finish with the selection.

The Element Properties [Multiple Selection] form appears having the tab “Element Loads” active.

Click on *LL* at the “Empty Load Cases” list box and then enter:

- *Surface* as the Force Type
- *Z* as Direction
- *Global* as System
- *Uniform* as Load Distribution
- *-40* as v1

Press **Create New** and the number 476 appears next to the \$ symbol the Empty Load Cases list box. This is a notification that 476 elements are loaded in Load Case LL.

Element Properties [Multiple Selection]

Model
Element Loads

Applied Load Cases

Empty Load Cases

LL \$[0]
PL \$[0]

Loading

Load Case : LL

Create New

Description : Live Load

Save / Modify

Loads in this Load Case :

▼

Delete

Force Type

☐ Body
☒ Surface
☐ Edge

▼

Direction

☐ X
☐ Y
☒ Z

System

☐ Local
☐ Planar
☒ Global
☐ Projected

Load Distribution

☒ Uniform
☐ Variable

Values (B or S)

v1 : -40
v2 :
v3 :
v4 :

OK

Cancel

Apply

Help

Press OK to close the dialog.

By double clicking on one element and then selecting the Element Loads tab and clicking on LL at the Applied Load Cases List Box, you will be able to view and edit the existing loading values of the specific shell.

To clear the arrows select  Clear from the GTS Display Ribbon Tab.

Step #21. Apply Pressure Load: Pressure Load will be applied only to elements in the Group Wall, therefore click on the icon  Colors . Select the 2nd Tab in order to colorize elements by their group and make only the Group Wall visible and press OK.

Sections
Groups

Categories

Groups	Color	Visible
Bottom	161	☐
Wall	191	☒
UnGrouped data	256	☐

Click on the icon  Shell in the Loads Panel. Using a full window, select all entities that appear on screen and press <ENTER> to finish with the selection.

The Shell Properties [Multiple Selection] form appears having the tab “Element Loads” active.

Click on PL at the “Empty Load Cases” list box and then enter:

- *Surface* as the Force Type
- *Z* as Direction
- *Local* as System
- *Uniform* as Load Distribution
- *5* as v1

Press **Create New** and the number 656 appears next to the \$ symbol the Empty Load Cases list box. This is a notification that 656 elements are loaded in Load Case PL.

The screenshot shows the 'Element Properties [Multiple Selection]' dialog box with the 'Element Loads' tab active. On the left, there are two list boxes: 'Applied Load Cases' (empty) and 'Empty Load Cases' (containing 'LL \$[476]' and 'PL \$[656]'). The 'PL \$[656]' entry is highlighted. The main area contains several configuration sections: 'Loading' (Load Case: PL, Description: Pressure Load, Loads in this Load Case: dropdown), 'Force Type' (radio buttons for Body, Surface, Edge; Surface is selected), 'Direction' (radio buttons for X, Y, Z; Z is selected), 'System' (radio buttons for Local, Planar, Global, Projected; Local is selected), 'Load Distribution' (radio buttons for Uniform, Variable; Uniform is selected), and 'Values (B or S)' (input fields for v1, v2, v3, v4; v1 is 5). At the bottom right are buttons for 'Create New', 'Save / Modify', and 'Delete'. At the bottom of the dialog are 'OK', 'Cancel', 'Apply', and 'Help' buttons.

Press OK to close the dialog.

To clear the arrows select  **Clear** from the GTS Display Ribbon Tab.

Click on the icon  **Colors**. Select the 2nd Tab and make everything visible.

Step #22. Define a Load Combination: Click on the icon  **Combinations** and the Load Combination dialog appears. Enter:

- *CB1* as Name
- *Load Combination 1* for the Description of the Load Combination
- Click on SW, Enter 1.3 as the factor and press ADD>>
- Click on LL, Enter 1.5 as the factor and press ADD>>

- Click on PL, Enter 1.1 as the factor and press ADD>>
- Press Store
- Press Done to close the dialog.

New Form Load or Load Combination

Load Information

Name : CB1 Design variable : ▼

Description : Load Combination 1

Type

☒ Load Combination

☐ Form Load

Combine

SW (Self Weight)
LL (Live Load)
PL (Pressure Load)

ADD >>

Factor : 1.1

Delete Item

SW 1.30000
LL 1.50000
PL 1.10000

STORE V

All Formed Loads or Combinations

Edit Delete

Done

4.10. Create GT STRUDL Input File

Step #23. Create GTI: Click on the icon  and the Create GT STRUDL Input file dialog appears. Keep the default GTI filename, check the options “Perform Stiffness Analysis”, “Read Joint Displacements” and “Read Finite Element Results” as shown in the following image and press OK.

Create GT.STRUDL Input File ✕

Export to GTI

Create GTI File : ...

☒ Export Whole Model
 ☐ Export Portion of the Model
 (will be prompted for Selection)

☒ Perform Stiffness Analysis

Append Other GTI Files/Macros

☐ Copy Commands from GTI Files/Macros (not CINPUT)

Create Commands to Read Results

☒ Read Joint Displacements
☐ Read Member Forces
☐ Read Section Forces Number of Sections:
☐ Read Section Displacements Number of Sections:
☒ Read Finite Element Results
☐ Read Code Check Results

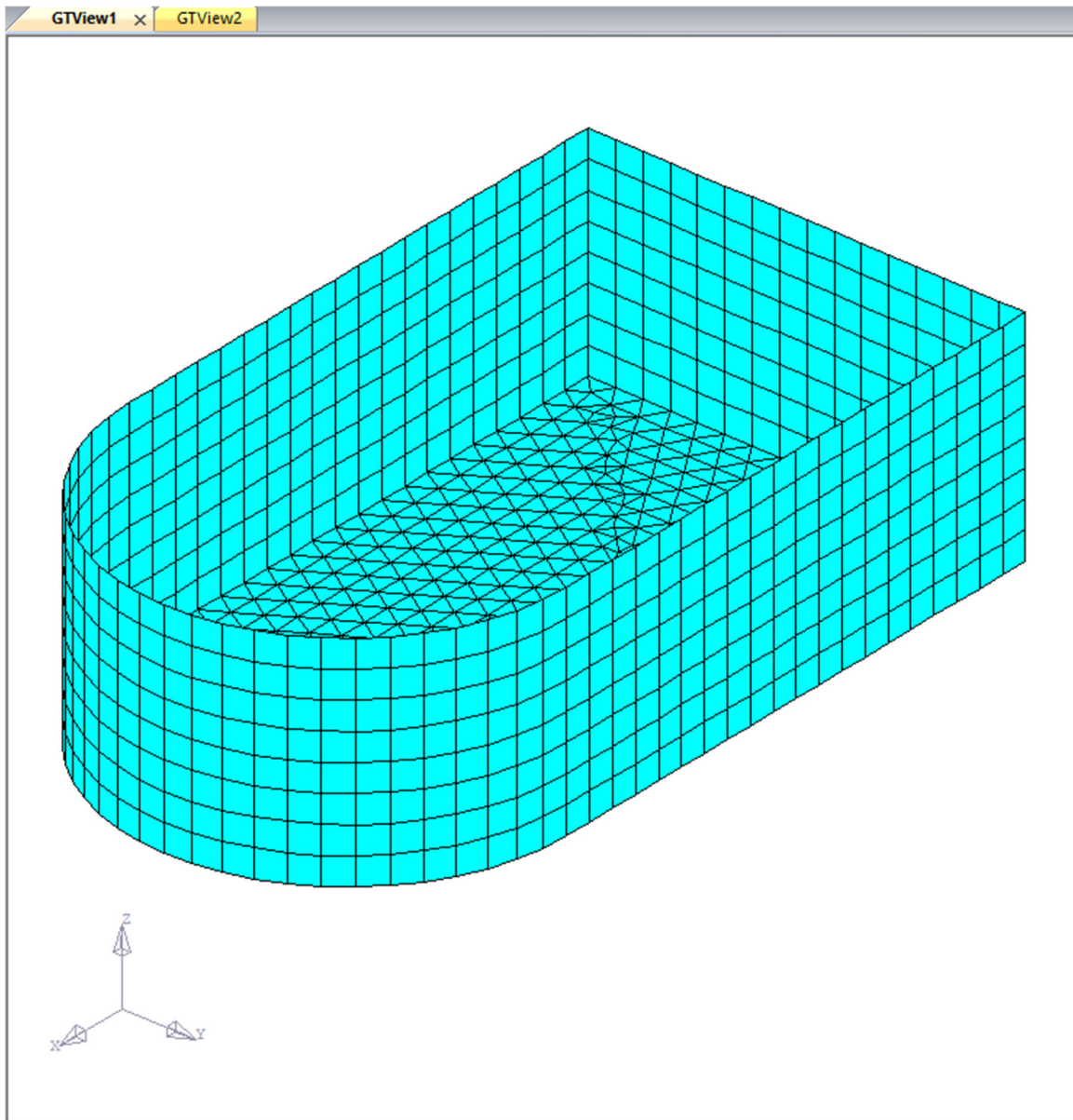
Step #24. View/Edit GTI: Click on the icon  Edit GTI and the GTI file created in the previous step will be opened by the system's default text editor.

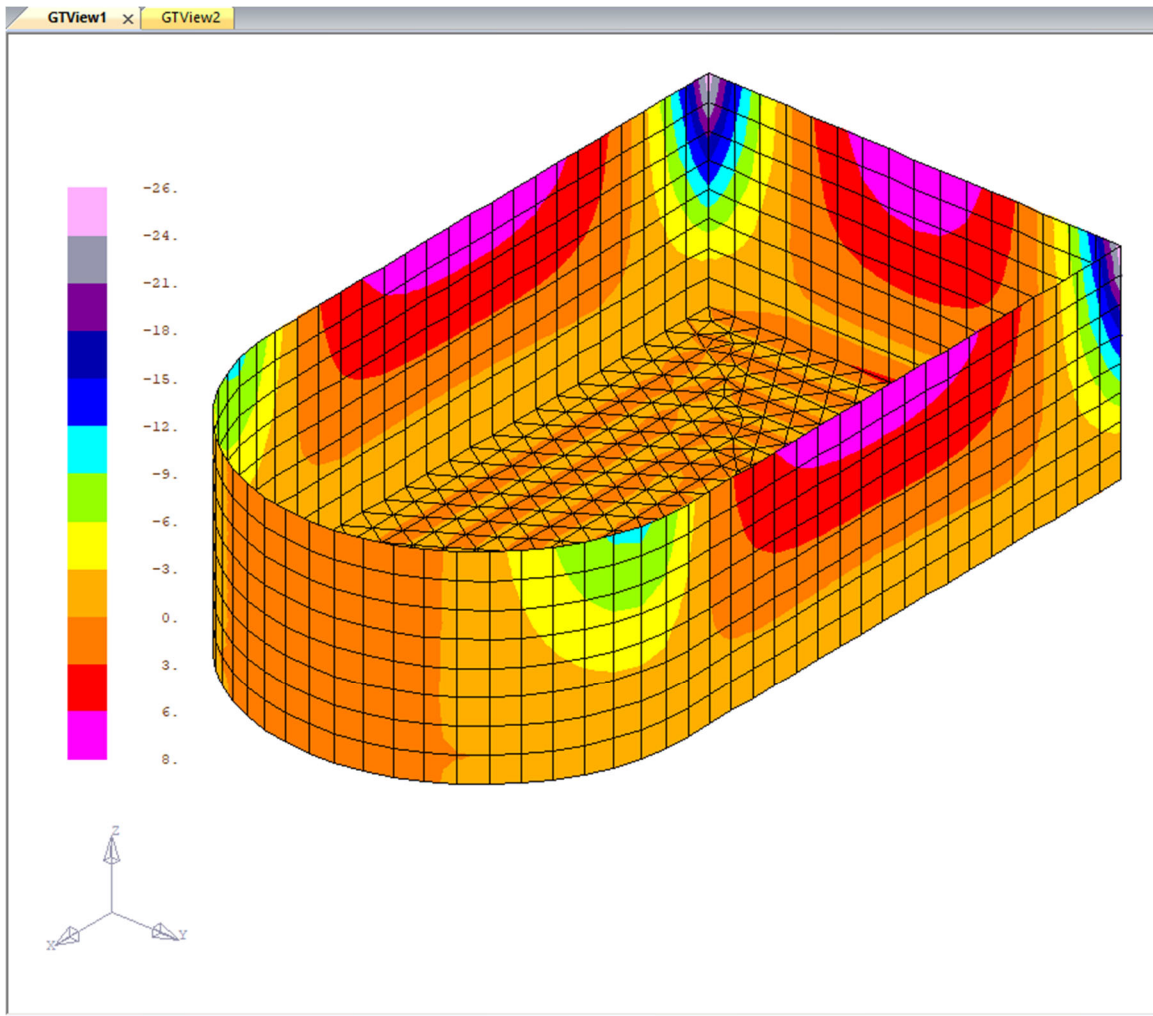
Step #25. Execute GT STRUDL: Click on the icon  and the GTI file created in the previous step will be sent to GT STRUDL main program that is waiting in the background.

Stiffness analysis is automatically performed and DBX result files are automatically created.

In addition, you can enter GTMENU to view the solid model and the results as described in the GTMENU User Guide. You can also click on Results > Finite Element Results > Contour Sresses,

Strains, Displacement and display MXX Bending Resultants for load case PL as shown in the figures on the next page:





Step #26. Read Results from GT STRUDL: Click on the icon  **Read Results** and the Read GT.STRUDL Results dialog appears. Check the options “Read Joint Displacements” and “Read Finite Element Results” as shown below and press OK.

Read GT.STRUDL Results ✕

Read DBX Results

GTI Directory : ...

☒ Read Joint Displacements

☐ Read Member Forces

☐ Read Section Forces Number of Sections:

☐ Read Section Displacements Number of Sections:

☒ Read Finite Element Results

Type of element result : ▼

Surface : ▼

☐ Read Code Check Results

GTI Commands

Corresponding GT STRUDL commands:

DBX BINARY

WRITE REPLACE JOINT RESULTS JOINTS EXISTING

WRITE REPLACE AVERAGE ELEMENT STRESSES TOP MIDDLE BOTOM ELEMENTS EXIS

WRITE REPLACE AVERAGE ELEMENT STRAINS TOP MIDDLE BOTOM ELEMENTS EXIST

WRITE REPLACE AVERAGE RESULTANTS TOP MIDDLE BOTOM ELEMENTS EXISTING

WRITE REPLACE AVERAGE ELEMENT PRINCIPAL STRESSES TOP MIDDLE BOTOM ELEI

WRITE REPLACE AVERAGE ELEMENT PRINCIPAL STRAINS TOP MIDDLE BOTOM ELEM

WRITE REPLACE AVERAGE PRINCIPAL MEMBRANE RESULTANTS TOP MIDDLE BOTOM

OK Cancel

If you get the following error message at the command prompt:

```
ERROR Loading Results: The following DBX files cannot be found:
```

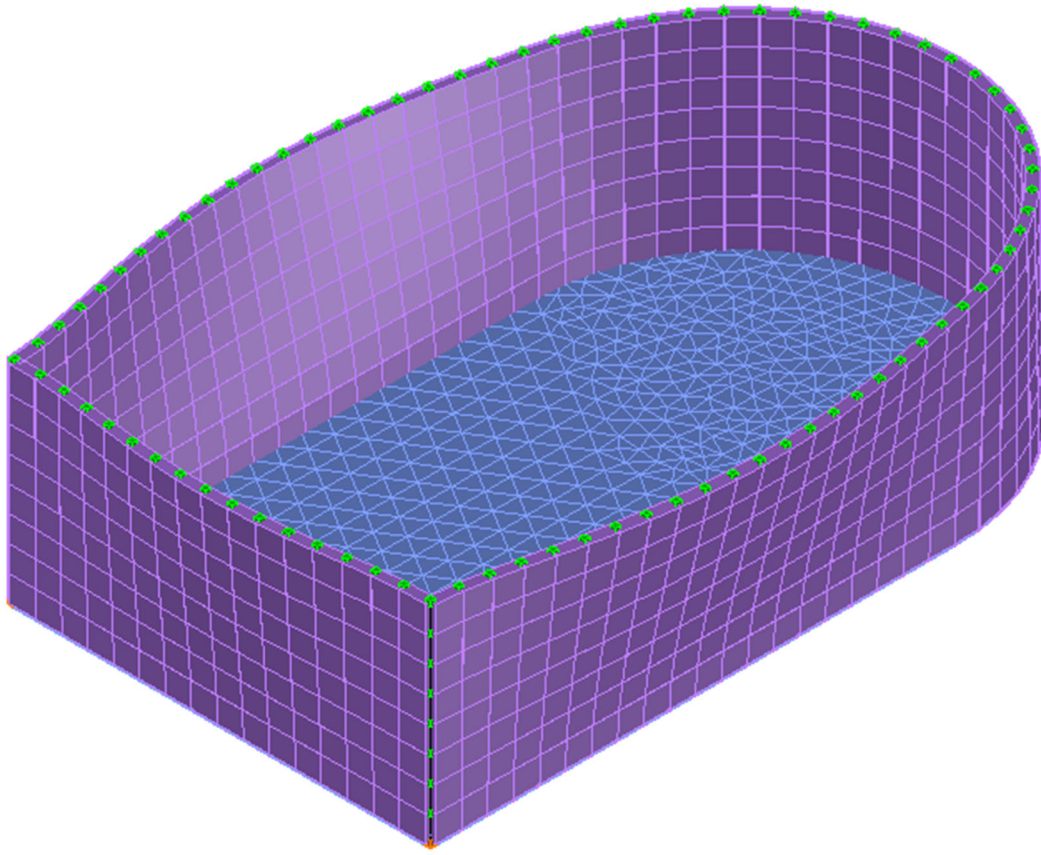
```
STDBX34 - Strains
```

```
STDBX37 - Principal Strains
```

This message informs you that no Strain results are available to be loaded. You can ignore this message, since the elements used (SBHQ6) do not give strains as output results.

4.11. Display Results

Step #27. Show Displacements: On the Menu Bar, click on  **Deformed** (ribbon tab “GTS Display”) and then select PL as the load Case and press ENTER twice. The deformed structure will be drawn as shown below.



Note: You can Annotate the Joint displacements by clicking on the  Annotate Displacements icon, that is located under the  Deformed in the ribbon and the selecting a joint and annotation position.

Click on  Undeformed (ribbon tab “GTS Display”) to return to the original undeformed position of the model.

Another alternative is to click on  Displacements (ribbon tab “GTS Display”).

Displacements

Load Case / Load Combination

PL \$ Pressure load

Display Options

Scale Factor (Values) :
0.100

Font Size (pt) :
10.00

Annotation Format:
Exponential

Decimal Places :
2

☒ Hide Model

Display >>

Annotate >
Legend >

Animation Options

Frames :
7

Animation Speed % :
10

Generate Animation Frames
Animate >>

Clear
Close

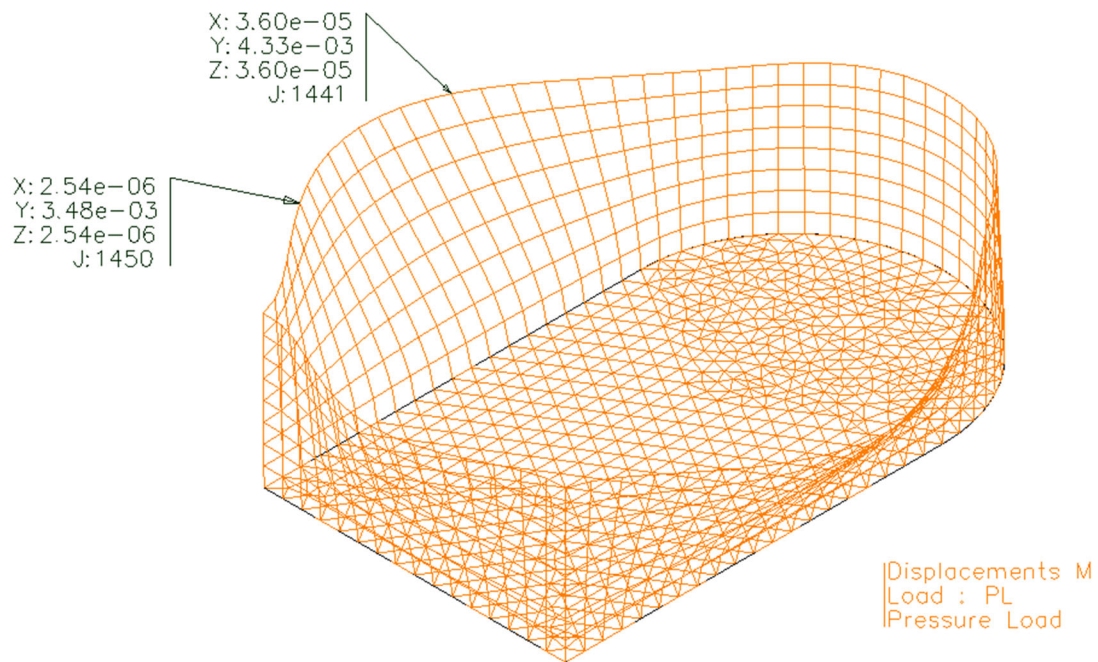
Select:

- *PL* as Load Case
- *0.1* as Scale Factor
- *10.00* as Font Size (default)
- Annotation Format: Exponential
- *Check Hide Model*

Press “Display >>” to view the deformed shape.

Press “Annotate >” and click on any part of the deformed shape curve and then at the position that you want the annotation to be displayed.

Press on “Legend >” and click at any part of the screen to place the legend of the diagram.



In order to display the animation press “Generate Animation Frames” and then “Animate >>”. To terminate the animation press “Stop” button.

In order to exit the command, uncheck “Hide Model”, press “Clear” Button and “Close”.

Step #28. Show Finite Element Results: Click on  Elements (ribbon tab “GTS Display”) and the Element Results Form appears.

Select:

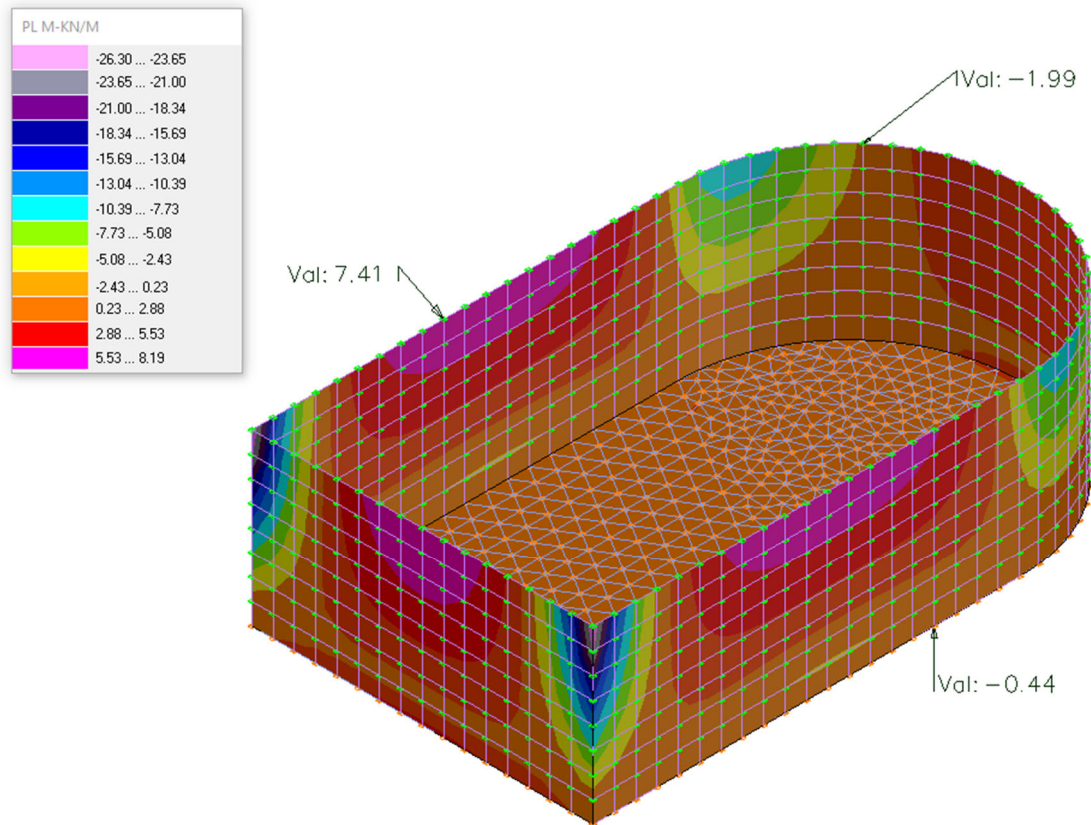
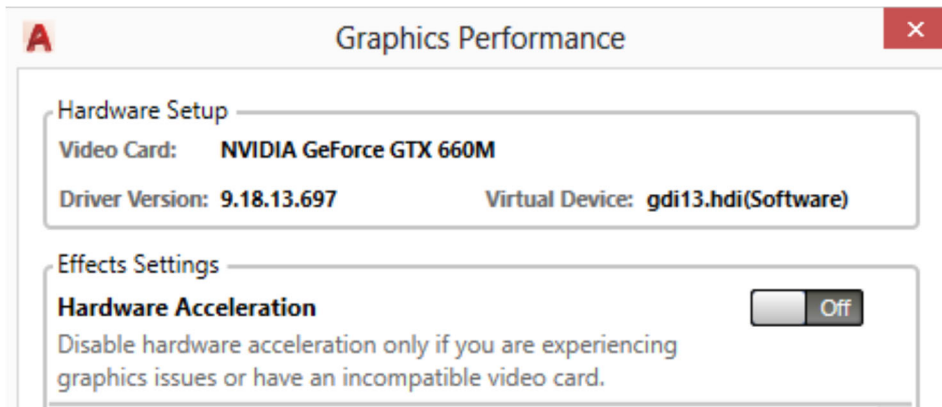
- *PL* as Load Case
- *Resultants* as the Type of element result
- *Mxx* as the Moment Resultant to display
- *Middle* as position (Resultants are only available for the middle surface of a 2D finite element)

and press “*Display >>*”

You can also press the “*Annotate >*” button and select joints to display the corresponding values.

The multi-colored contour image of the structure is displayed and each color corresponds to a range of *Mxx* values as shown in the Legend Form. Type *Shade* to view the contours in shaded mode as shown on the next page.

NOTE: Hardware graphics acceleration may cause AutoCAD to incorrectly display the colors of the contour. In such a case it is recommended that you turn OFF Hardware Acceleration, during displaying the stress contours, by typing the command `GRAPHICSCONFIG`. You can turn it back ON afterwards.



In order to display the animation press “Generate Animation Frames” and then “Animate >>”. To terminate the animation press “Stop” button.

To exit the command, uncheck “Hide Model”, press “Clear” Button and “Close”.

4.12. Results Datasheets

Step #29. In addition to the graphical display of results there is an option to view them in datasheets from the item “Results Datasheets” of the menu/ribbon tab “GTS Display”.

Click on the icon  **Displacements** and the “GTSTRUDL – Joint Displacement Datasheet” dialog appears where you can filter, sort, write results to text file or change results units as shown in figure below;

GTSTRUDL - Joint Displacement Datasheet, display Units: Meters Degrees

File Edit Columns Filter Sort Units Help

Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y
1	SW	0.0000	0.0000	0.0000	0.0000106	-0.0000138
2	SW	0.0000	0.0000	0.0000	-0.0000232	0.0000019
3	SW	0.0000	0.0000	0.0000	-0.0000178	0.0000063
4	SW	0.0000	0.0000	0.0000	-0.0000217	-0.0000060
5	SW	0.0000	0.0000	0.0000	-0.0000299	0.0000074
6	SW	0.0000	0.0000	0.0000	-0.0000240	-0.0000108
7	SW	0.0000	0.0000	0.0000	-0.0000245	0.0000075
8	SW	0.0000	0.0000	0.0000	-0.0000541	0.0000093
9	SW	0.0000	0.0000	0.0000	-0.0000309	-0.0000440
10	SW	0.0000	0.0000	0.0000	-0.0000478	0.0000193
11	SW	0.0000	0.0000	0.0000	-0.0000426	-0.0000289
12	SW	0.0000	0.0000	0.0000	-0.0000393	0.0000310
13	SW	0.0000	0.0000	0.0000	-0.0000511	-0.0000206
14	SW	0.0000	0.0000	0.0000	-0.0000470	0.0000164
15	SW	0.0000	0.0000	0.0000	-0.0000420	-0.0000298
16	SW	0.0000	0.0000	0.0000	-0.0000307	0.0000099
17	SW	0.0000	0.0000	0.0000	-0.0000394	0.0000016
18	SW	0.0000	0.0000	0.0000	-0.0000175	-0.0000173
19	SW	0.0000	0.0000	0.0000	-0.0000129	0.0000066
20	SW	0.0000	0.0000	0.0000	-0.0000321	0.0000080
21	SW	0.0000	0.0000	0.0000	-0.0000101	-0.0000294
22	SW	0.0000	0.0000	0.0000	0.0000268	0.0000065
23	SW	0.0000	0.0000	0.0000	0.0000194	0.0000086
24	SW	0.0000	0.0000	0.0000	0.0000164	0.0000071
25	SW	0.0000	0.0000	0.0000	0.0000193	0.0000075
26	SW	0.0000	0.0000	0.0000	0.0000132	0.0000096
27	SW	0.0000	0.0000	0.0000	0.0000088	0.0000062
28	SW	0.0000	0.0000	0.0000	-0.0000073	0.0000003

Click on the icon  **Stresses** and the “GTSTRUDL – Average Stresses Datasheet” dialog appears where you can filter, sort, write results to text file or change results units as shown in figure below.

GT STRUDL - Average Stresses Datasheet, display Units: KiloNewtons Meters Degrees

File Edit Columns Filter Sort Units Help

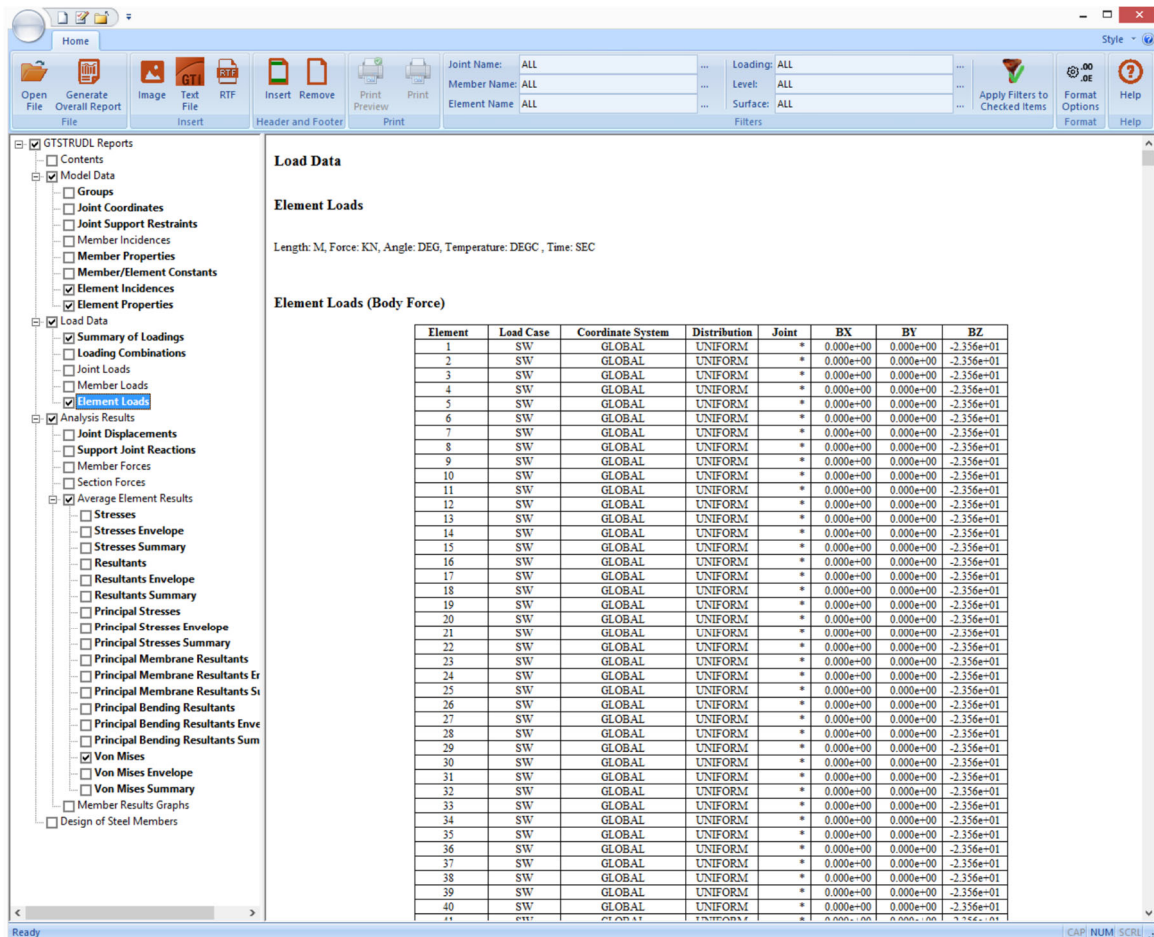
Sorted on: Not sorted Filter: None

Joint	Load	Surface	SXX	SYY	SXY	VonMises
1	SW	Top	-9.072	-79.302	1.757	43.439
1	SW	Middle	-10.035	-68.864	-0.187	37.202
1	SW	Bottom	-10.998	-58.427	-2.131	31.122
1	LL	Top	12.447	-9.762	16.378	19.803
1	LL	Middle	-0.211	-0.052	0.225	0.250
1	LL	Bottom	-12.868	9.658	-15.928	19.530
1	PL	Top	1.229	-32.917	-13.717	23.734
1	PL	Middle	-1.770	-39.660	3.729	22.713
1	PL	Bottom	-4.768	-46.403	21.175	33.166
2	SW	Top	-6.494	-41.199	-3.870	22.486
2	SW	Middle	-6.716	-36.815	-3.170	19.860
2	SW	Bottom	-6.939	-32.431	-2.470	17.255
2	LL	Top	-2.938	-40.908	-8.212	24.250
2	LL	Middle	-0.010	-0.434	1.038	1.067
2	LL	Bottom	2.918	40.039	10.288	24.578
2	PL	Top	26.386	-11.697	-30.198	35.951
2	PL	Middle	-2.172	-8.818	0.775	4.659
2	PL	Bottom	-30.731	-5.938	31.748	35.689
3	SW	Top	-2.867	-41.443	-2.248	23.253
3	SW	Middle	-6.252	-36.609	-2.432	19.733

4.13. Report Builder



Step #57. Click on the icon  to generate the report of this structure by calling Report Builder. Report Builder is launched in a separate window. You can find more information on how creating your report in the GT STRUDL® Report Builder Getting Started Guide (Help Icon on the right top corner on Report Builders' window).



The screenshot shows the GT STRUDL Report Builder window. The left sidebar contains a tree view of report categories, with 'Element Loads' selected. The main area displays the 'Element Loads (Body Force)' report, which includes a table of data for 40 elements.

Load Data

Element Loads

Length: M, Force: KN, Angle: DEG, Temperature: DEGC, Time: SEC

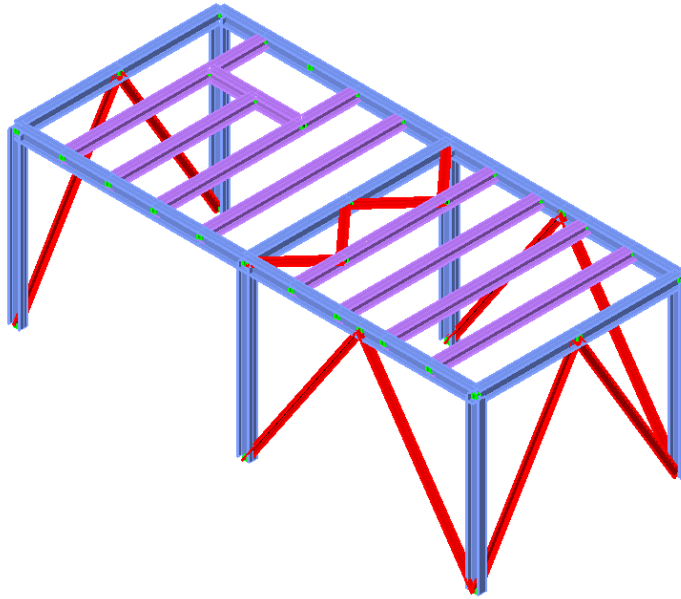
Element Loads (Body Force)

Element	Load Case	Coordinate System	Distribution	Joint	BX	BY	BZ
1	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
2	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
3	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
4	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
5	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
6	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
7	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
8	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
9	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
10	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
11	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
12	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
13	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
14	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
15	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
16	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
17	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
18	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
19	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
20	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
21	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
22	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
23	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
24	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
25	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
26	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
27	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
28	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
29	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
30	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
31	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
32	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
33	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
34	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
35	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
36	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
37	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
38	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
39	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01
40	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01

5. Tutorial Example #3

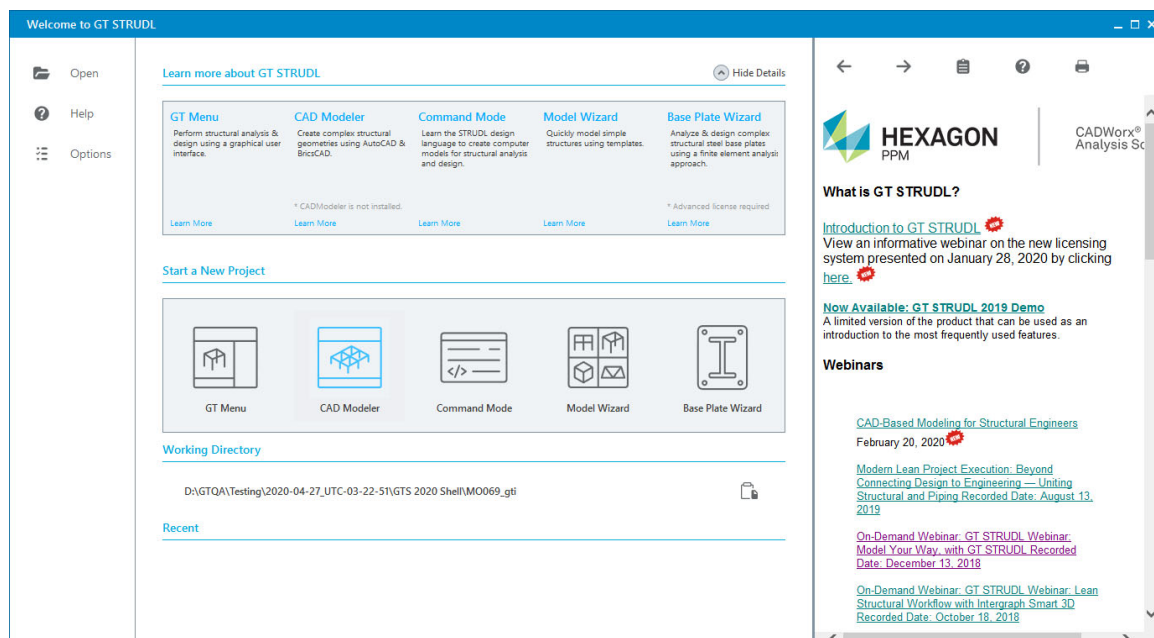
5.1. Introduction

The modeling of a frame structure using CAD Modeler shown below is demonstrated in a step-by-step process.

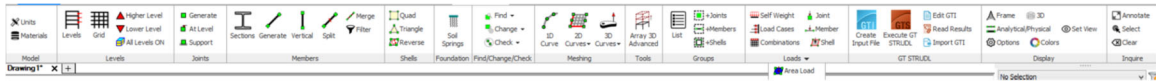


5.2. Open CAD Modeler and start working

Step #1. Launch GT STRUDL by selecting the icon “CAD Modeler” in the Welcome to GT STRUDL dialog shown below. The version of AutoCAD/BricsCAD selected during the installation will be automatically launched, together with CAD Modeler’s menus and ribbons.



Step #2. Make sure that CAD Modeler's ribbons and menus are visible.



If AutoCAD's/BricsCAD's menu is not visible, type `MENUBAR` at AutoCAD's/BricsCAD's command prompt, then 1 and press `<ENTER>`.

If AutoCAD's/BricsCAD's ribbon area is not visible, type `RIBBON` and press `<ENTER>`.

Ribbon commands will be used in this tutorial example. However, since all ribbon commands appear in the menu area, you can use the menu bar as well.

5.3. Define the basic geometry of the model

Step #3. Define the correct **Units** by pressing the icon  and select *Feet (m)* and *Kips* in the *Units Form*.

 A screenshot of the 'Units' dialog box. It contains four sections: 'Length' with options 'Inches (in)', 'Feet (ft)' (selected), 'Meters (m)', 'Centimeters (cm)', and 'Millimeters (mm)'; 'Force' with options 'Pounds (lbs)', 'Kips' (selected), 'Tons', 'Kilograms', 'Metric Tons', 'Newtons', 'KiloNewtons', and 'MegaNewtons'; 'Angles' with options 'Degrees' (selected), 'Radians', and 'Cycles'; and 'Time' with options 'Seconds' (selected), 'Minutes', and 'Hours'. At the bottom, there is a checkbox 'Scale non-structural CAD entities (grids, structural lines, curves etc)' which is checked, and 'OK' and 'Cancel' buttons.


Step #4. Define the one level of the model by pressing the icon . Press the *Add Level* button one time to add a level to your model. Modify the height of the 1st level by selecting the *Height* cell of the 1st Level and entering 15.

Note: Some Edit Boxes appear in yellow background and green fonts, like the one at the Level Heights. You can use mixed units in the yellow edit boxes. For more information about Mixed Units and the valid syntax please read GT STRUDL GT Menu Guide

Make sure that *Z Vertical Axis* option is checked and press `OK` to close the form.

Level Properties

Levels	Height	Elevation	Visible
1	15.000000	15.000000	☒

Options

Add Level

Delete Level

Detect Levels Automatically

Merge Levels

Base Elevation: 0.0000

☒ Z Vertical Axis (else Y)

☐ Update Levels for All Entities

OK Cancel



Step #5. Enter a Grid that will help you enter the columns quickly by clicking on the icon  . The grid is going to have 2 (20.4167 ft) spaces in the horizontal direction (X) and 1 space (18 ft) in the sidelong direction (Y). Enter 20ft-5in in the *Distance* text box and press the *Add* button 2 times. The program will automatically convert the distance of 20ft -5in to 20.4167 ft.

Grid

Placement

☒ Horizontal ☐ Sidelong

Spacing

Distance: 20.4167

Add

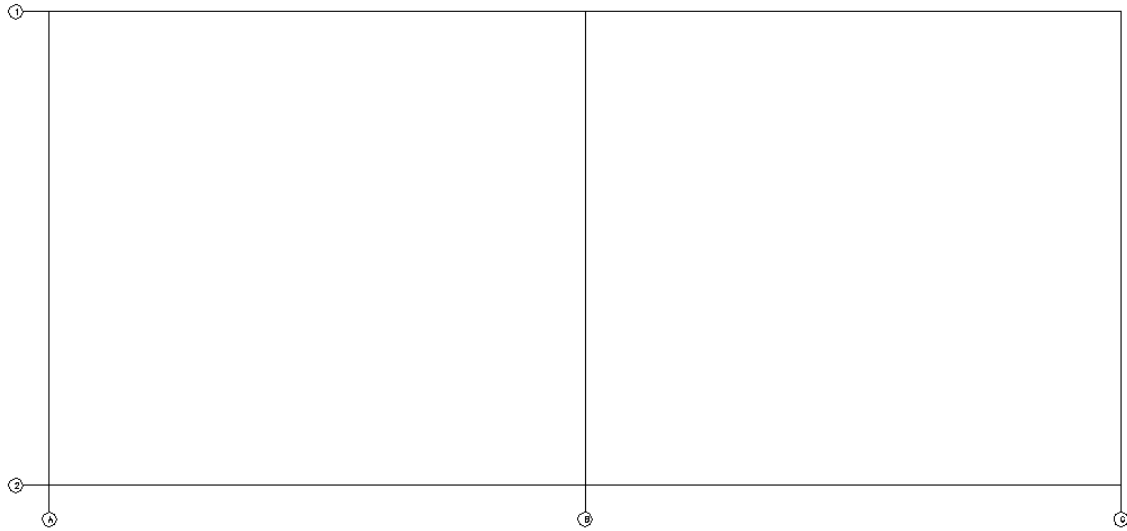
Edit

Delete

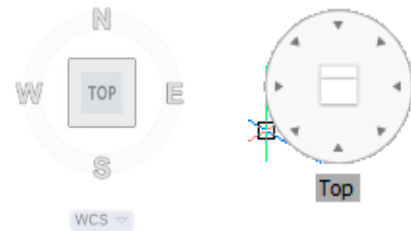
Then, click on *Sidelong* in the *Placement* area to enter the spacing in the Y direction, and enter 18 and *Add*.

By pressing OK you are prompted (command prompt) to enter the Insert Point for the grid. Type 0, 0, 0 and press <ENTER>.

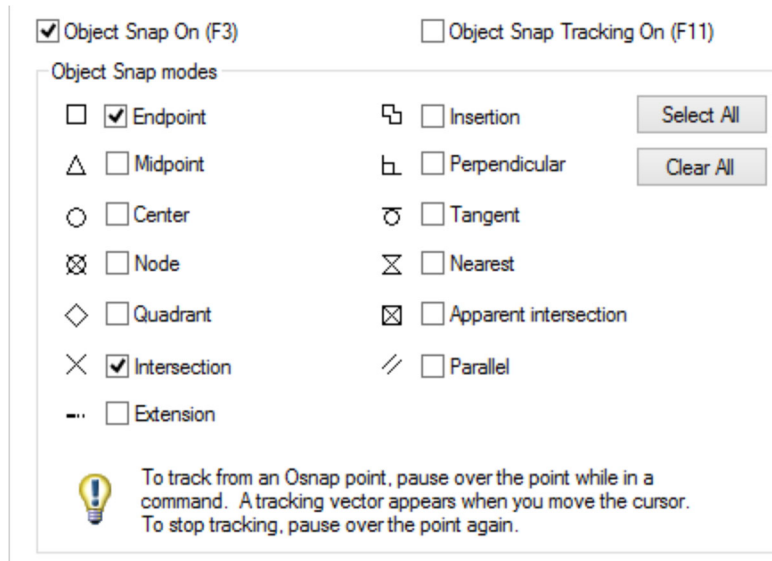
The grid is created, having its upper left corner A-1 at the point 0,0,0.



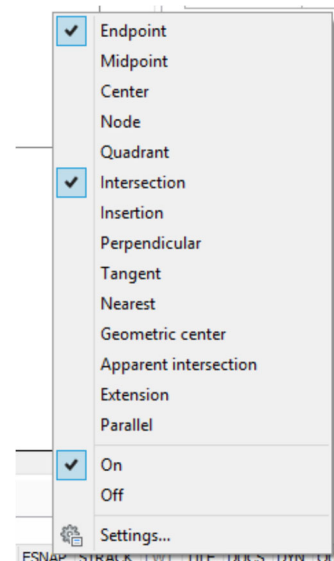
You will be able to view the grid by pressing the Top Icon in AutoCAD's Cube or BricsCAD's Chair, or preferably by typing **Z** (for Zoom), **E** (for Extents) and press **<ENTER>**.



Note: In order to be able to snap at the intersection of the grid lines, while placing columns, make sure that the AutoCAD's/BricsCAD's Object Snap is ON, and the Intersection mode is enabled. Type **OSNAP** for AutoCAD or right click on "ESNAP" setting for BricsCAD to set the various snap settings. Shown below is the ObjectSnap tab in AutoCAD's and BricsCAD's Drafting settings dialog.



AutoCAD's Snap Setting



BricsCAD's Snap Settings

Step #6. Click at *Top Front Right* of the View Cube (AutoCAD) or Chair (BricsCAD) to change the view of the model.



AutoCAD's View Cube for Isometric View

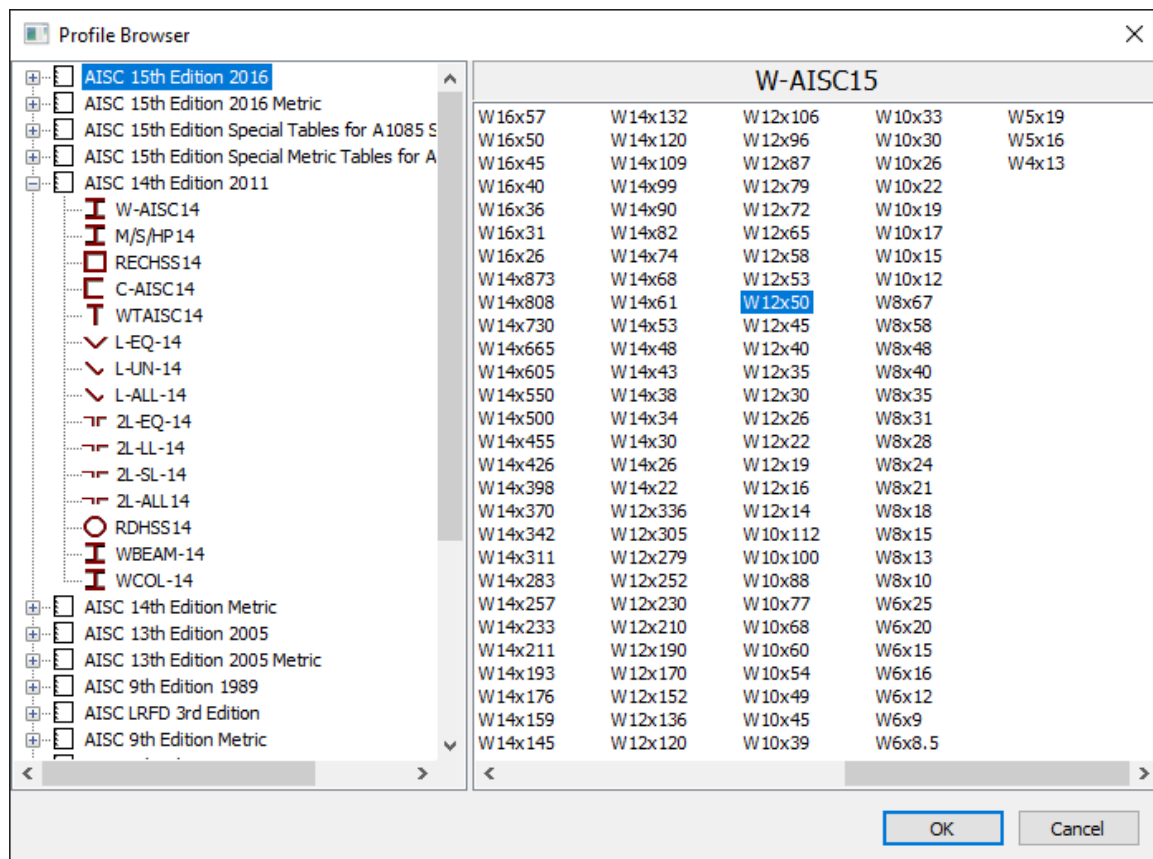


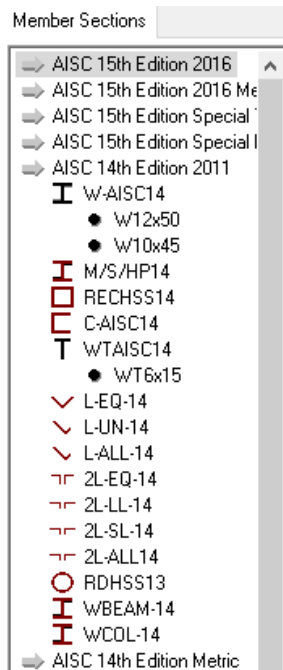
BricsCAD's View Cube for Isometric View



Sections

Enter the cross-section profiles that will be used at the model by pressing the icon [Sections](#). Click on the *AISC 14th Edition 2011* list and then on the *W-AISC14* table of profiles. Select the profile *W12x50* that will be used for the columns, by double clicking on it.





The profile is added to the project and it appears in the left list-box having a black dot in front of it.

Using the same procedure, add 2 additional profiles: *W10x45*, for beams and *WT6x15* for the bracing from the table *WTAISC14*. Press **OK** to close the form.

Note: You can add additional profiles at any time by following this procedure and also view the full list of profiles used in your model and add more profiles if needed (left image).

5.4. Create Columns



Vertical

Step #7. Start entering the columns by clicking on the icon **Vertical**. The dialog *Place Member* appears that helps you to quickly select properties for the members that are going to be entered.

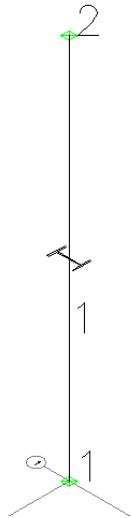
Select *W12x50* as the cross section for the columns. Make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *90*. There is no need to close this dialog manually.

Press the “Place Member(s) >>” button.

Click at the intersection point between line A and line 1 (point A-1) and the column will be placed at this position.

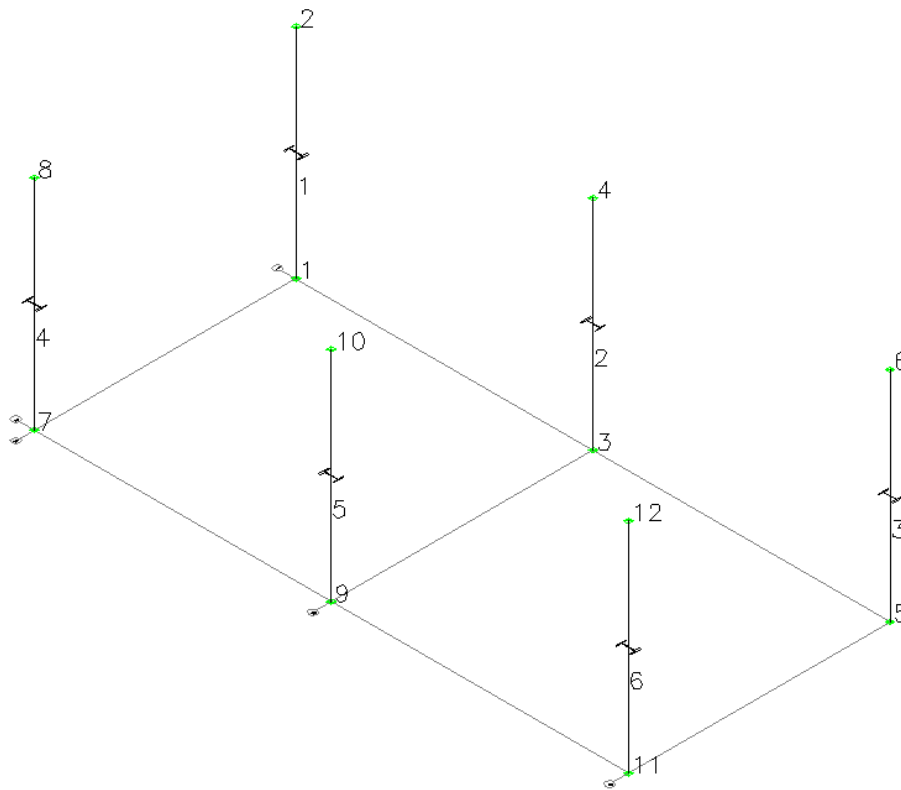
Repeat the same procedure by clicking at the points B-1, C-1, A-2, B-2 and C-2.

When you are done, press **ESC** to exit the *Vertical* column command. The *Place Member* form is automatically hidden.



Note: Each time you create a member, the orientation of the cross section will appear in the middle of the element, unless you clear it with command “Clear” (see 2.6.86).

Step #8. As you can see in the isometric view below, column members 1 to 6 were created together with joints 1 to 12 at their ends. Each column is 15 ft long, as defined in Level Properties (height of the first floor).



5.5. Create beams and girders



Step #9. Start entering the beams, along X axis, by clicking on the icon **Generate** . The *Place Member* form appears.

Cross Section

Table Section:
W12x50 W-AISC14 AISC 14th Edition 2011

or Member Dimensions:
<Select Shape>

or Same as Member:
<Select Existing Member>

Material
Steel

Releases
Fix-Fix

Beta (o)
90

☒ Split Intersecting Members ☒ Physical Member
☒ Split Ending Members

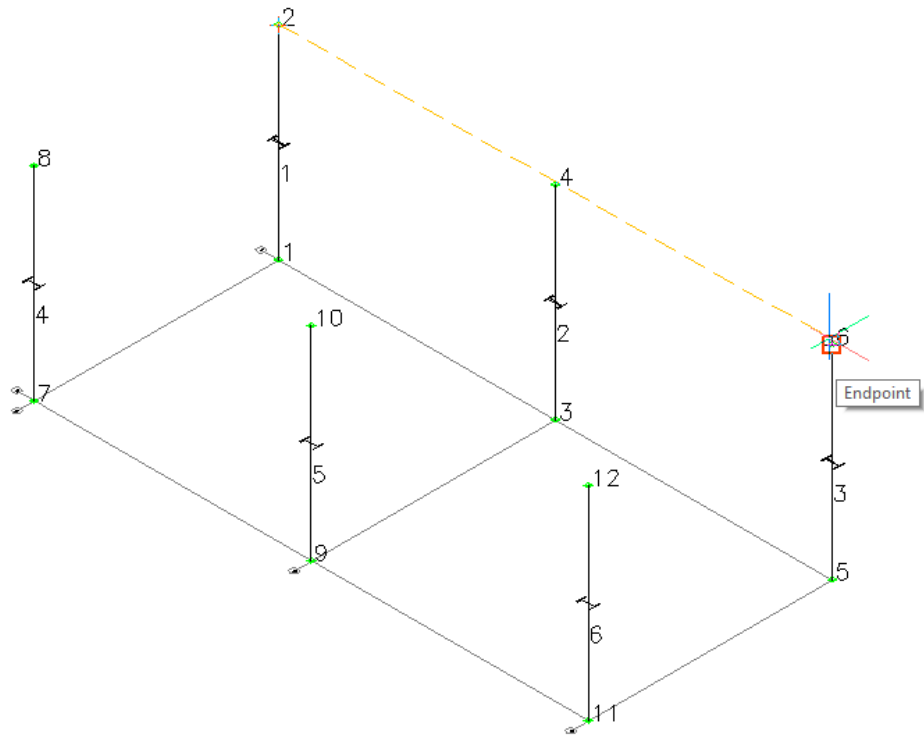
Place Member(s) >>

Select *W12x50* as the cross section and make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *90*. You have to set the Beta angle equal to 90 degrees in order to orientate the local Y axis of the W12x50 cross section along the Z global axis. Make sure that the option *Split Intersecting Members* is checked and then uncheck Split Ending Members. Moreover, check the “Physical Member” option, so that you also define the physical member and not only the analytical members.

Press “Place Member(s) >>” button.

Starting Point (x,y,z) message appears, asking you to enter the coordinates or click on a specific point on screen. Click on Joint 2 at the top joint of column 1 at position A-1.

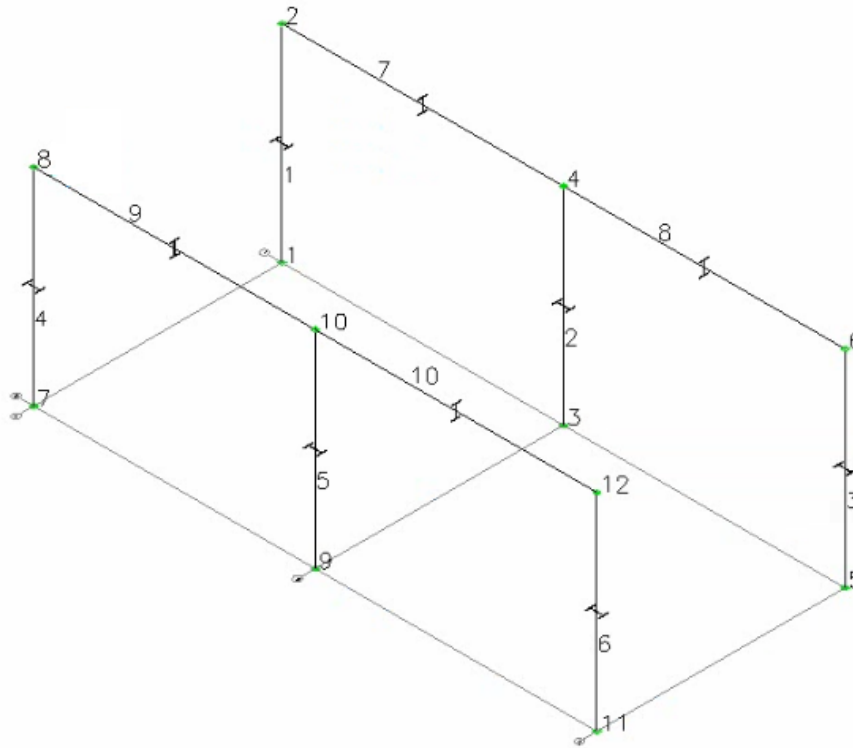
Then you have to define the *Ending Point (x,y,z)* so click on Joint 6, as shown in the picture below.



Members 7 and 8 will be created.

The two beams along X axis were generated with only two clicks of the mouse: at Joints 2 and 6. The beam from Joint 2 to Joint 6, was split into two parts, between Joints 2, 4 and 6, since Joints 4 (column at positions B-1) intersect this member.

Since the command is still active, you are prompted to enter the Starting Point (x,y,z), repeat the same procedure by clicking on Joint 8 (top of column at position A-2), and then click at Joint 12 (top of column at position C-2). Members 9 and 10 will be created.



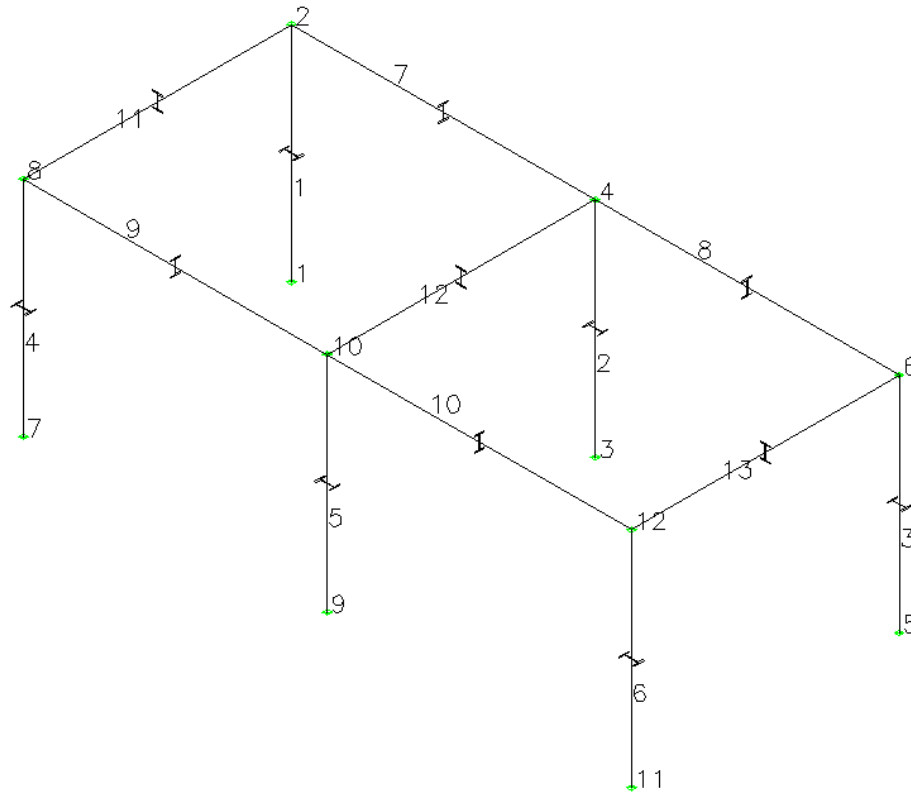
Step #10. Enter the beams, along Y axis. The command Generate Beams should be still active,



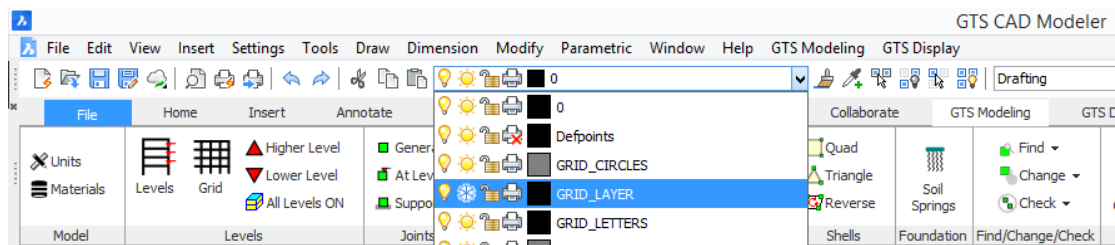
else you can call it again by clicking again on the icon **Generate**. Keep the same settings at the Place Member Form, as in the previous step, regarding the cross section and Beta angle, but do NOT click on *Split Intersecting Members*. Press the “Place Member(s) >>” button.

The prompt message *Starting Point (x,y,z)* appears, asking you to enter the coordinates or click on a specific point on screen. Click on the Joint 8, that is the top of column at position A-2. In order to define the *Ending Point (x,y,z)*, click at Joint 2 (top of column at position A-1). Member 11 is generated.

Repeat the same procedure by clicking on the Joints 10 and 4 to generate Member 12, Joints 12 and 6 to generate Member 13. Then, press ESC to terminate the command.



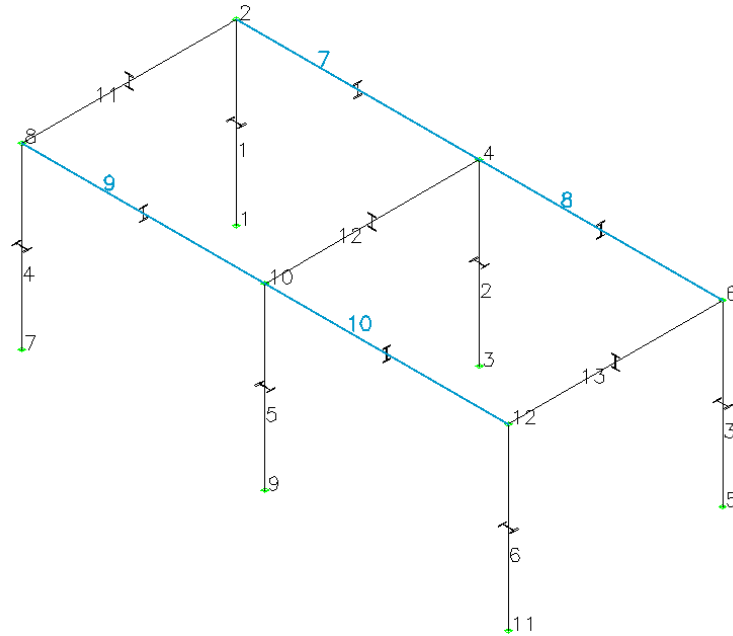
Step #11. Hide Grid: Since the Grid is no longer needed it can be hidden using AutoCAD/BricsCAD commands to freeze the layer GRID_LAYER. This can be done by clicking on the Home Tab and then selecting the GRID_LAYER from the drop down list of layers and clicking on its freeze icon.



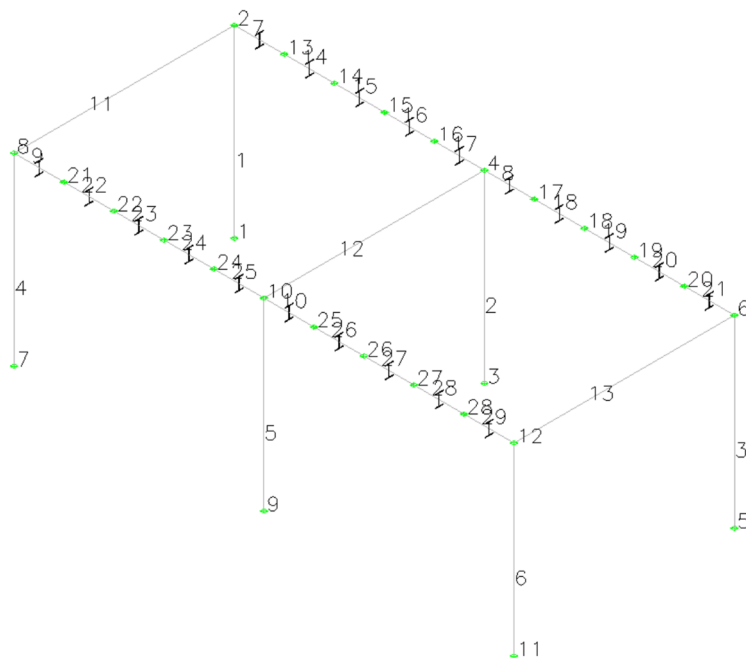


Split

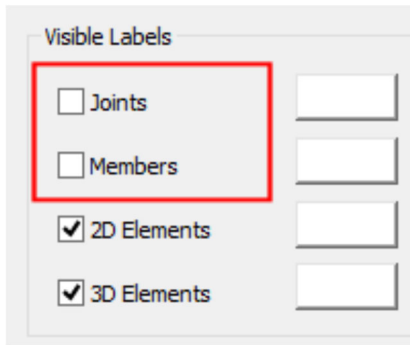
Step #12. Split horizontal members: Click on the icon and click on the members 7, 8, 9 and 10, and then press <ENTER>.



In order to define the Distance for splitting the member or the number of parts (negative number), enter -5 , so that the beams will be split into 5 equal parts.

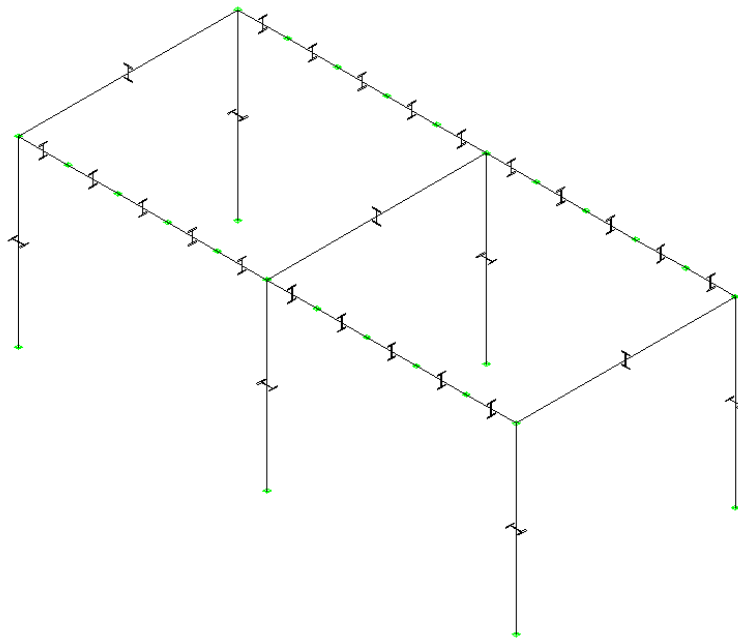


Step #13. Turn OFF labeling:

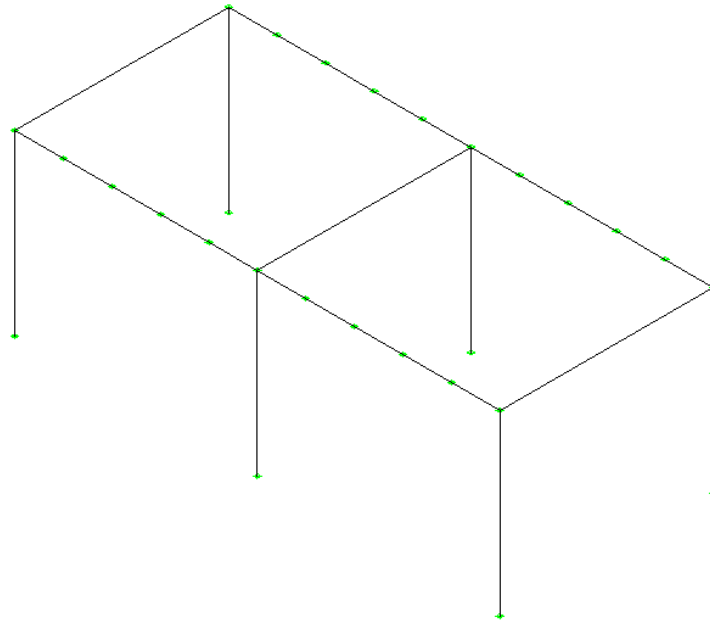


Click on the icon  **Options** in the ribbon bar and then uncheck the Visible Labels option for Joints and Members.

Now labeling is turned off and it is easier and faster to view and control the model.



Press the icon  **Clear** to remove the informative cross section shapes in the middle of each member:



5.6. Create girders



Step #14. Place girder members along Y axis: Click on the icon **Generate** and Place Member form appears.

Cross Section

Table Section:

W10x45 W-AISC 14 AISC 14th Edition 2011
...

or Member Dimensions:

<Select Shape>

or Same as Member:

<Select Existing Member>

Material

Steel
...

Releases

Pin-Pin

Beta (o)

90

☐ Split Intersecting Members
☒ Physical Member

☐ Split Ending Members

Place Member(s) >>

Select *W10x45 W-AISC14 14th Edition 2011* as the cross section and make sure that Material is set to *Steel*, Releases to *Pin-Pin* and Beta angle is *90*. Uncheck *Split Intersecting Members*, Uncheck *Split Ending Members* and Check *Physical Member*

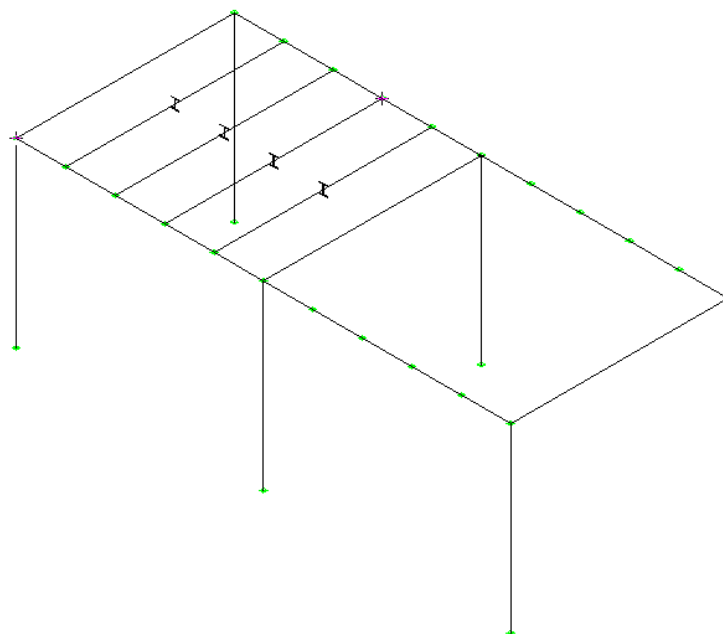
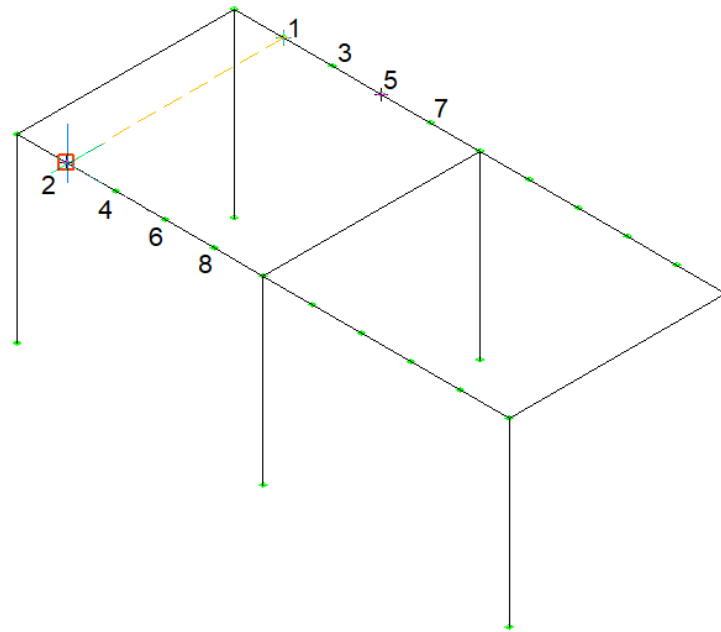
Press the “Place Member >>” button.

Click on the Joint located at the point 1 of the following image. Click on the joint at point 2 and the girder member is generated.

Having the command still active, click on the Joints at points 3 and 4 and another girder member is generated.

Continue by clicking on Joints at points 5 and 6 and another girder member is generated.

Continue by clicking on Joints at points 7 and 8 and another girder member is generated.

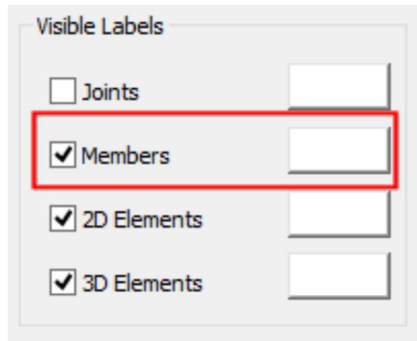


Press *ESC* to terminate the Generate Beam Command.

Press the icon  **Clear** to remove the informative cross section shapes in the middle of each member.

Step #15. Switch between analytical and physical view.

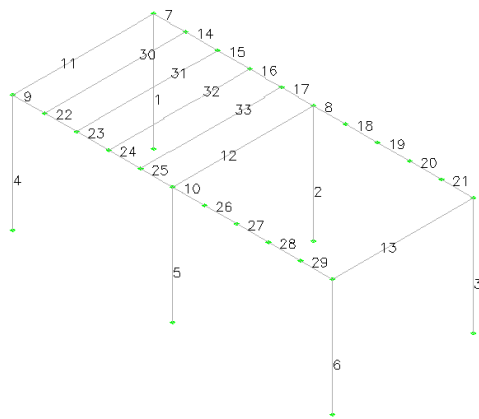
Turn ON labeling:



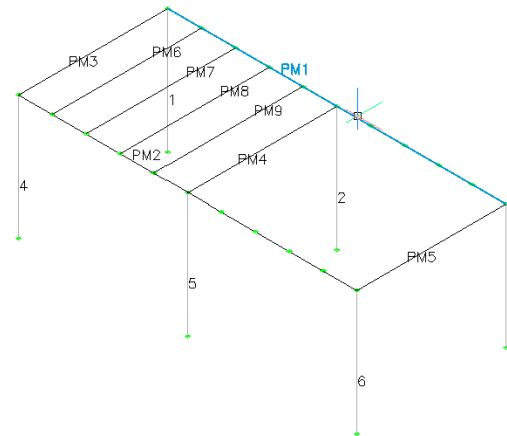
Click on the icon  **Options** in the ribbon bar and then check the Visible Labels option for Members and press OK

The labels of all members are visible now.

Press the icon  **Analytical/Physical** located in Display at Ribbon Area to switch from Analytical to Physical View as displayed in the picture bellow.



Analytical View



Physical View

Physical Members, with the prefix PM, were generated while placing the analytical member, since the "Physical Member" option was checked. By double clicking on a Physical Member, in example PM1, you are able to see its properties, including the set of analytical members that define it. In example for PM1 the set is: 7,14,15,16,17,8,18,19,20,21 as created by placing the beam, and splitting it afterwards.

Physical Member Properties

Model Section Properties Member Loads

General

Name : PM1

Level : 1

Type - Incidences

Type : SPACE FRAME

Start : 2

End : 6

Beta Angle : 90

Physical Member : PM1

Groups

☐ Inactive ... Remove

Analytical Members

7 14 15 16 17

Section Properties

Section : W12x50 W-AISC14 AISC 14th

Ax : 0.101389 Ix : 8.24653e-005

Ay : 0.0313472 Iy : 0.00271508

Az : 0.0478815 Iz : 0.018856

Sy : 0.00804397 Ey : 0

Sz : 0.0371528 Ez : 0

Yd : 1.01667 Yc : 0.508333

Zd : 0.673333 Zc : 0.336667

Shape Code : 1.0

Material Properties

Material : Steel

E : 4.176e+006 Density : 0.489543

G : 1.58399e+006 CTE : 1.17e-005

Poisson : 0.3

Releases & Elastic Connection spring values

Quick Selection :

Start	Spring	End	Spring
☐ Fx	0	☐ Fx	0
☐ Fy	0	☐ Fy	0
☐ Fz	0	☐ Fz	0
☐ Mx	0	☐ Mx	0
☐ My	0	☐ My	0
☐ Mz	0	☐ Mz	0

End Sizes OR Member Eccentricities (Offsets)

Start	End
Sizes : 0	0
X : 0	X : 0
Y : 0	Y : 0
Z : 0	Z : 0

OK Cancel Apply

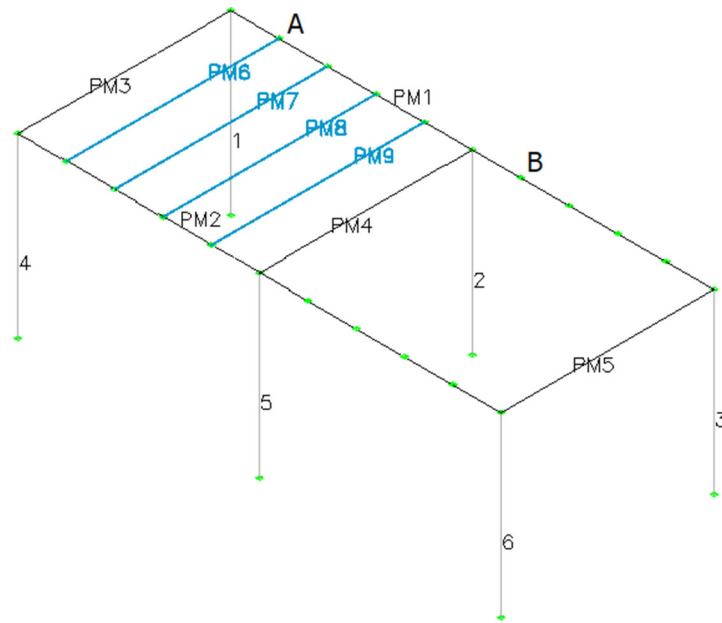
Step #16. Copy Physical Members

Having the physical member view ON the last 4 physical members created will be copied. If the COPY command is performed on physical members then the physical member is copied with its set of analytical members. If the COPY command is performed on analytical members, then only the analytical members are copied, without the physical member definition.

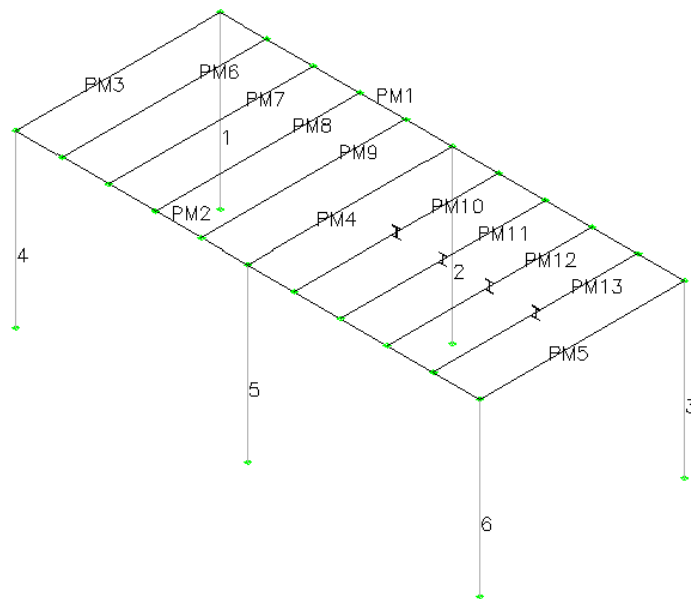
Type AutoCAD/BricsCAD COPY command and when asked to *Select objects/entities*: click on the members PM6, PM7, PM8 and PM9. You will get a notification about the selected entities and press <ENTER>.

In order to *Enter base point*: click on Point A of the following image.

In order to *Enter second point*: click on Point B of the following image.



Press ESC to exit from copy command. Another four girders, as physical members PM10, PM11, PM12 and PM13, are generated.

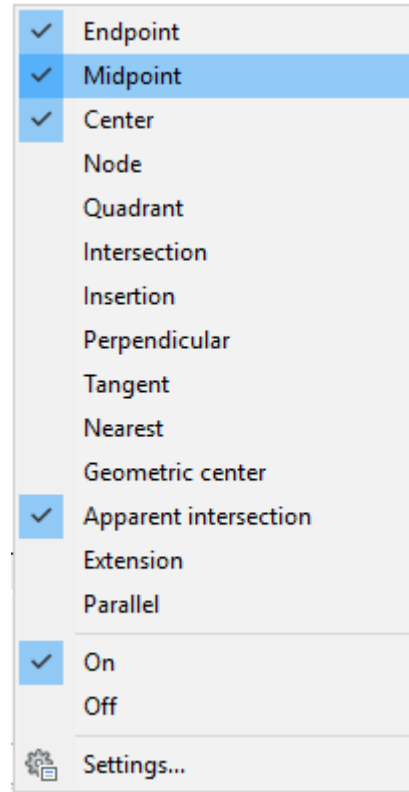
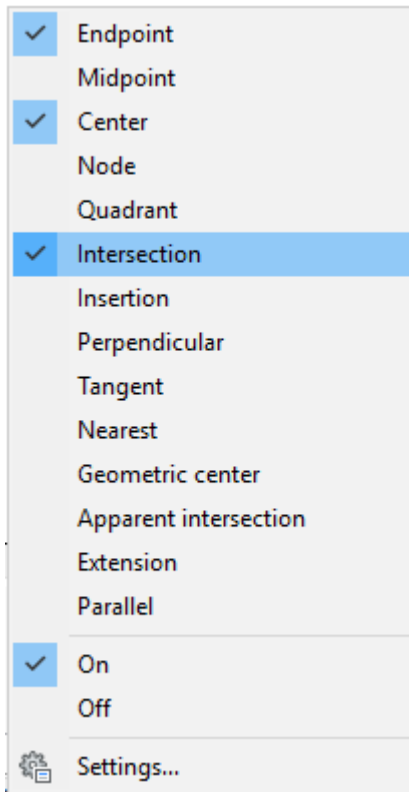


Switch back to analytical view by pressing the icon  Analytical/Physical and turn OFF member labeling (as explained in Step #13).

Press the icon  Clear to remove the informative cross section shapes in the middle of each member.

Step #17. Place bracing members.

Set ESNAP to Midpoint and not Intersection: Right-click at ESNAP at the bottom of the screen and deselect “Intersection” and select “Midpoint” option.



Click on the icon **Generate** and Place Member form appears.

Cross Section

Table Section:
 WT6x15 WTAISC14 AISC 14th Edition 2011 ...

or Member Dimensions:
 <Select Shape> ...

or Same as Member:
 <Select Existing Member> ...

Material
 Steel ...

Releases
 Pin-Pin ...

Beta (o)
 0 ...

☒ Split Intersecting Members ☐ Physical Member

☒ Split Ending Members

Place Member(s) >>

Select *W16x15 WTAISC14 14th Edition 2011* as the cross section and make sure that Material is set to *Steel*, Releases to *Pin-Pin* and Beta angle is *0*.

Make sure that the option “Split Intersecting Members” is ON, so that common joints will be created along the previously created Y-direction girders. Uncheck Physical Member option.

Press the “Place Member >>” button.

Click on the Joint located at the point 1 of the following image. Click on point 3, that will snap to the middle of the member, and the bracing member is generated.

Having the command still active, click on the Joints at points 2 and 3 and another member is generated.

Continue by clicking on Joint at point 4 and midpoint 6 and another member is generated.

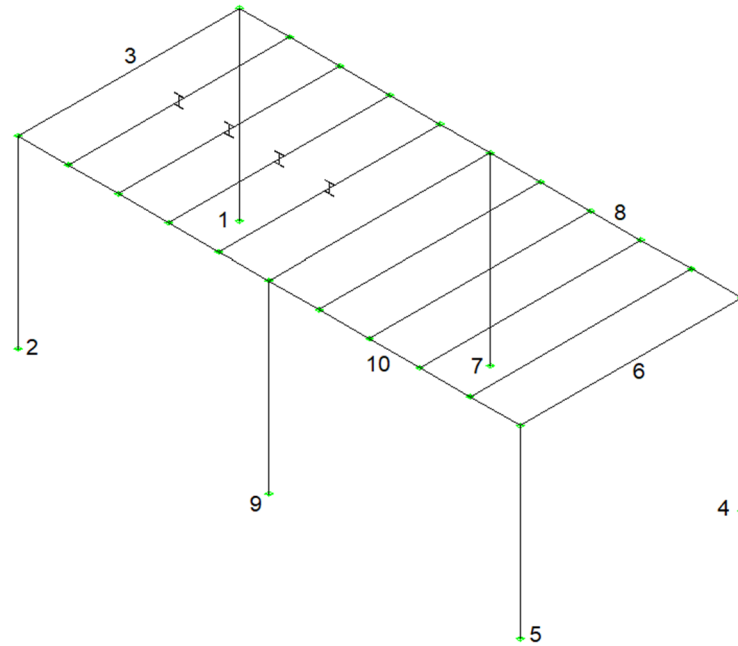
Continue by clicking on Joints at points 5 and 6 and another member is generated.

Continue by clicking on Joint at point 7 and midpoint 8 and another member is generated.

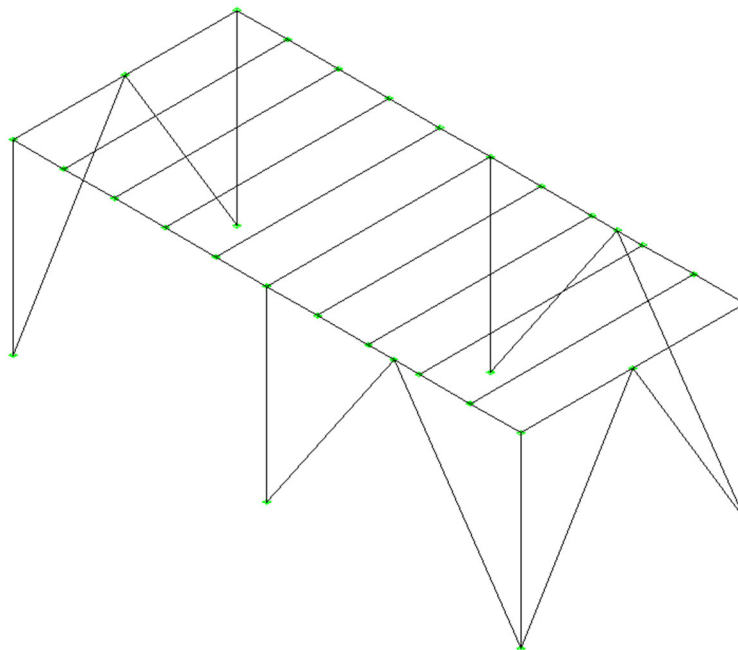
Continue by clicking on Joints at points 4 and 8 and another member is generated.

Continue by clicking on Joint at point 9 and midpoint 10 and another member is generated.

Continue by clicking on Joints at points 5 and 10 and another member is generated.



Press *ESC* to terminate the Generate Beam Command and click on the icon  **Clear** to remove the informative cross section shapes in the middle of each member.



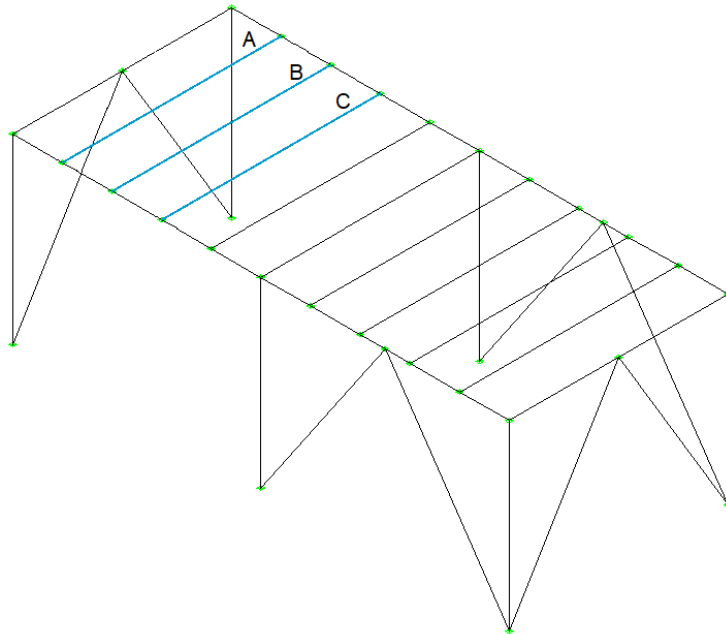
Step #18. Split Girders

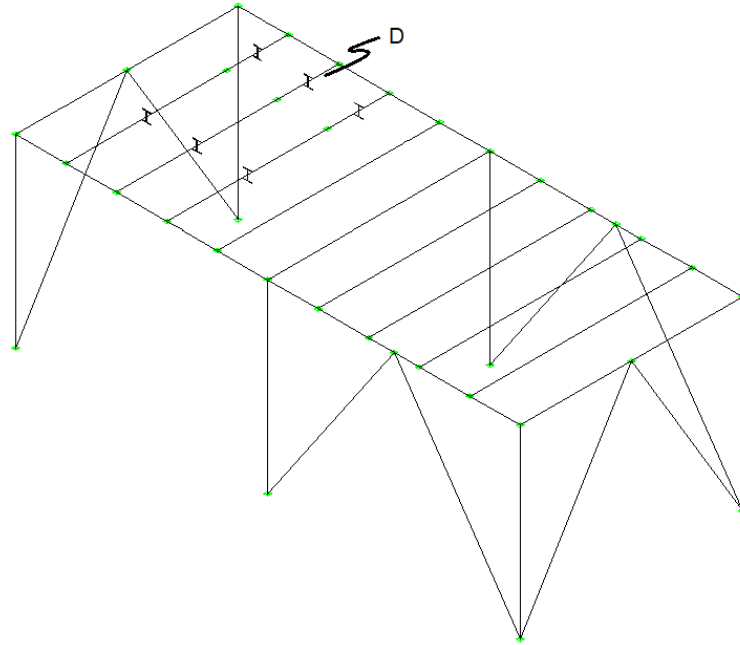


Split

Click on the icon  and click on the members marked as A, B and C, and then press <ENTER>. In order to define the *Distance for splitting the member or the number of parts (negative number)*, enter 5, so that the beams will be split at location 5 ft (current length units) from the start of the members.

Note: Split command can be used in two different ways. If you enter a negative value (eg. -5), the member is split into equal parts (eg. 5 equal parts). The value (absolute part) that you entered is used to set the number of equal parts. If you enter a positive value (eg 5), then the member is always split at two parts, at the point that is 5ft (current length units) from the start of the member. The value that you entered is used to set the distance from start.





Select member marked in D and delete it, by pressing DEL key and then and click on the icon  **Clear** to remove the informative cross section shapes in the middle of each member.

Step #19. Add additional members along X



Click on the icon **Generate** and Place Member form appears.

Cross Section

Table Section:
W10x45 W-AISC14 AISC 14th Edition 2011 ...

or Member Dimensions:
<Select Shape> ...

or Same as Member:
<Select Existing Member> ...

Material
Steel ...

Releases
Pin-Pin ...

Beta (o)
90 ...

☒ Split Intersecting Members ☒ Physical Member

☒ Split Ending Members

Place Member(s) >>

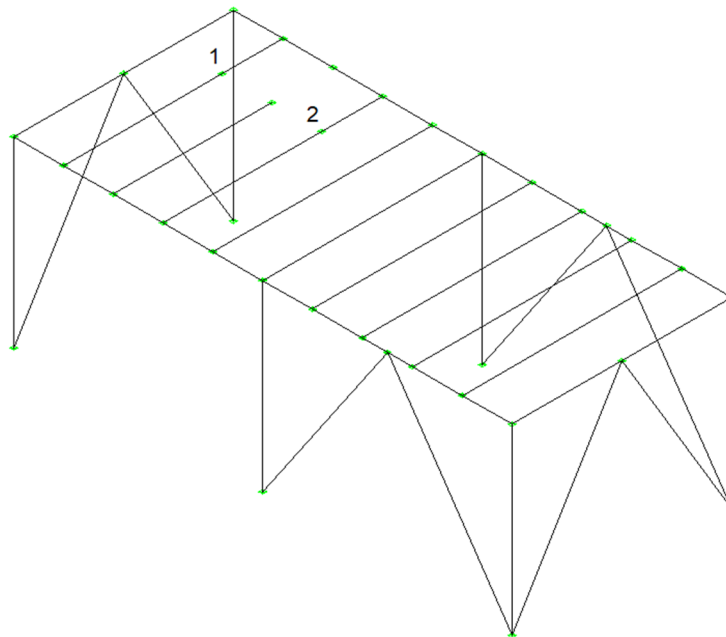
Select *W10x45 W-AISC14 14th Edition 2011* as the cross section and make sure that Material is set to *Steel*, Releases to *Pin-Pin* and Beta angle is *90*.

Make sure that the option “Split Intersecting Members” is ON, so that common joints will be created along the previously created Y-direction girders. Check Physical Member option.

Press the “Place Member >>” button.

Click on the Joint located at the point 1 of the following image. Click on the Joint at point 2 and two members are generated along X axis.

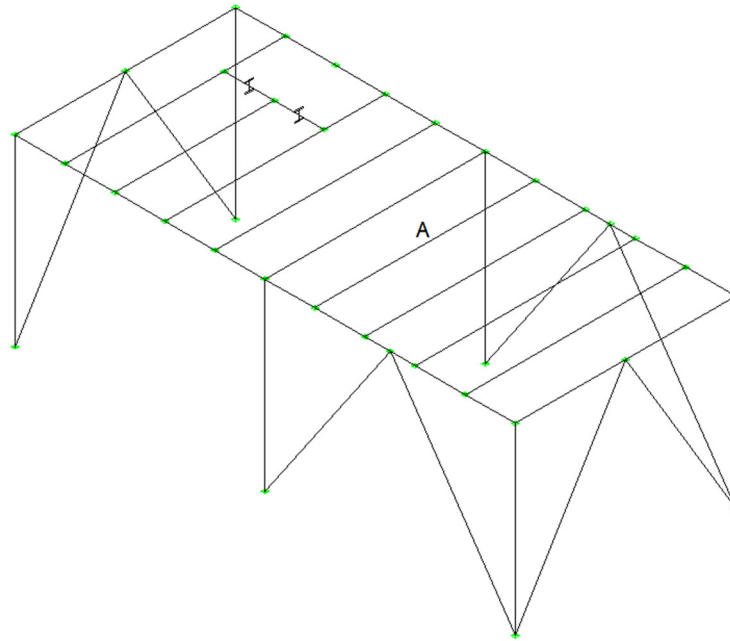
Press *ESC* to terminate the Generate Beam Command.



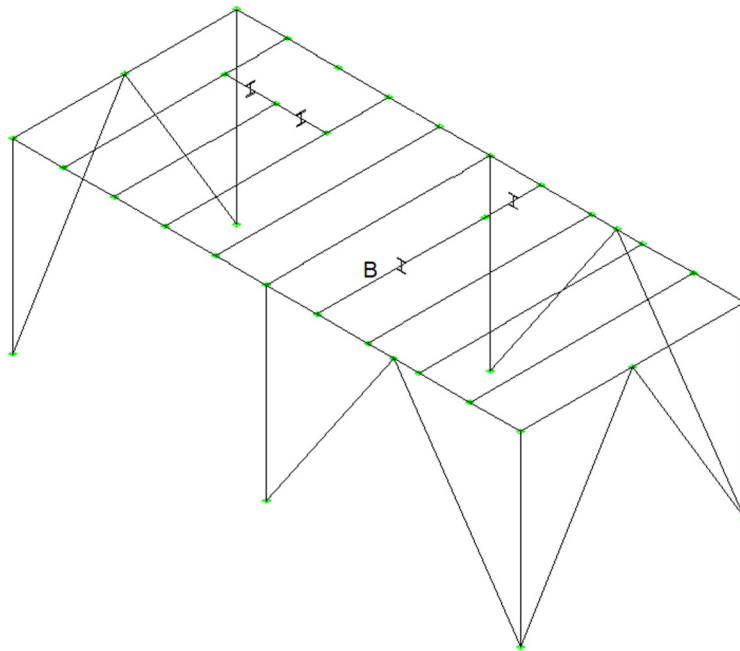


Split

Step #20. Split Girders: Click on the icon and click on the member marked in A (of the following image) and then press <ENTER>.



In order to define the *Distance for splitting the member or the number of parts (negative number)*, enter 4 . 5, so that the beam will be split at the location 4.5ft from the start of the member (see Notes of Step #18 for more information about the split command).



Split

Again, click on the icon **Split** and click on the member B (of the previous image) and then press <ENTER>. In order to define the *Distance for splitting the member or the number of parts (negative number)*, enter 9, so that the beam will be split at the location 9ft from the start of the member (see Notes of Step #18 for more information about the split command).



Clear

Click on the icon **Clear** to remove the informative cross section shapes in the middle of each member.



Generate

Step #21. Create additional braces: Click on the icon **Generate** and Place Member form appears.

Cross Section

Table Section:
 WT6x15 WTAISC14 AISC 14th Edition 2011 ...

or Member Dimensions:
 <Select Shape> ...

or Same as Member:
 <Select Existing Member> ...

Material
 Steel ...

Releases
 Pin-Pin ...

Beta (o)
 0 ...

☒ Split Intersecting Members ☐ Physical Member

☒ Split Ending Members

Place Member(s) >>

Select *WT16x15 WTAISC14 14th Edition 2011* as the cross section and make sure that Material is set to *Steel*, Releases to *Pin-Pin* and Beta angle is *0*.

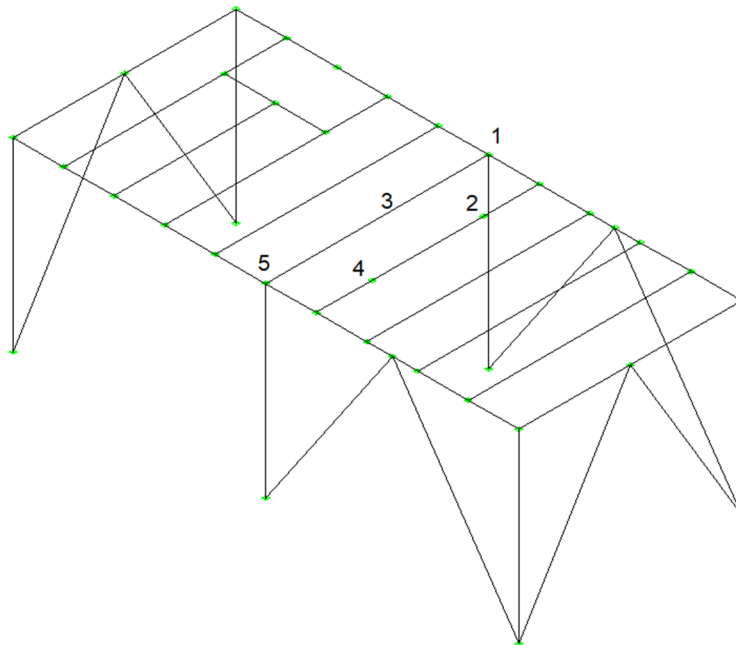
Make sure that the option “Split Intersecting Members” is ON and “Split Ending Members” is also ON. Uncheck Physical Member option.

Press the “Place Member >>” button.

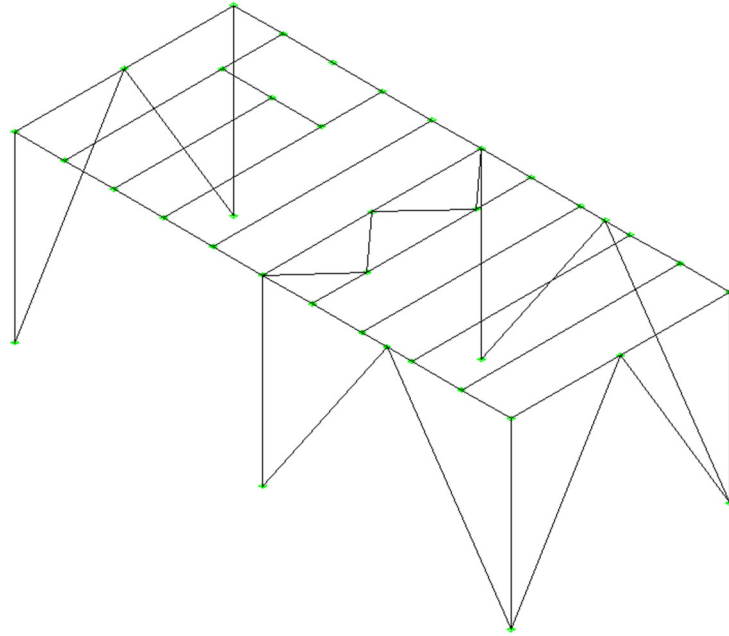
Click on the Joint located at the point 1 of the previous image. Click on the Joint at point 2 and the brace member is generated.

Having the command still active, click on the Joint at point 2 and midpoint 3 and another brace member is generated.

Repeat the same procedure to generate two more members from points 3 to 4 and 4 to 5.

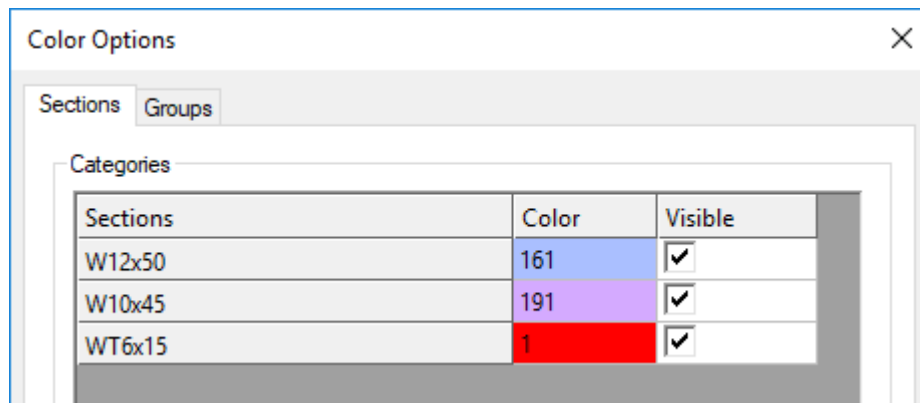


Press *ESC* to terminate the Generate Beam Command and click on the icon  **Clear** to remove the informative cross section shapes in the middle of each member. The created model is displayed in the following image.

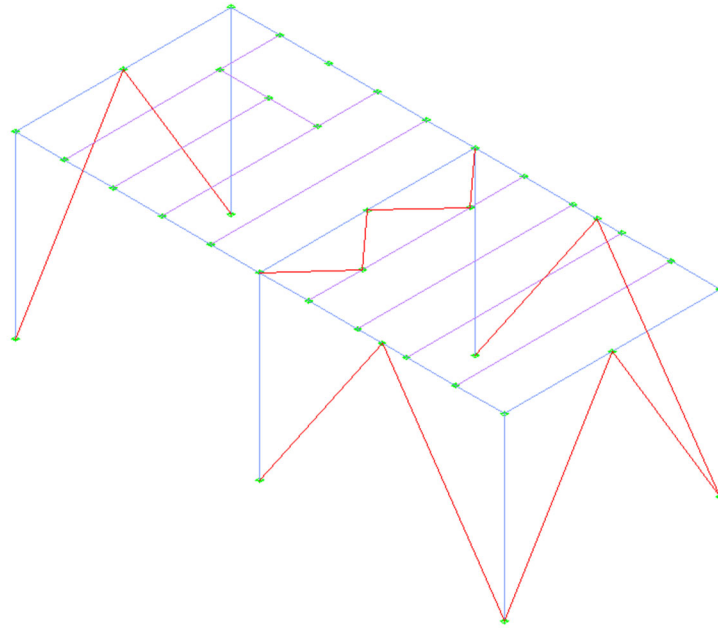


Step #22. Switch to 3D View

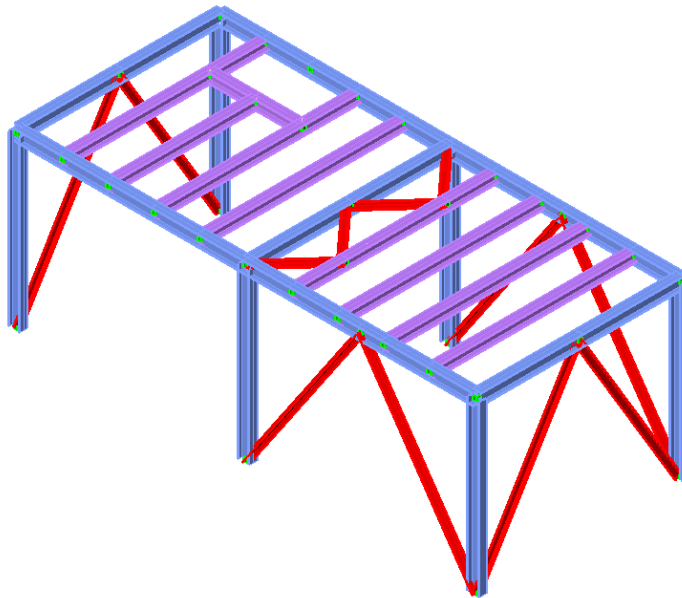
Click on the icon  Colors to set different colors for each profile.



Press OK to close the Color Options Dialog. Each cross section will now have a different color.



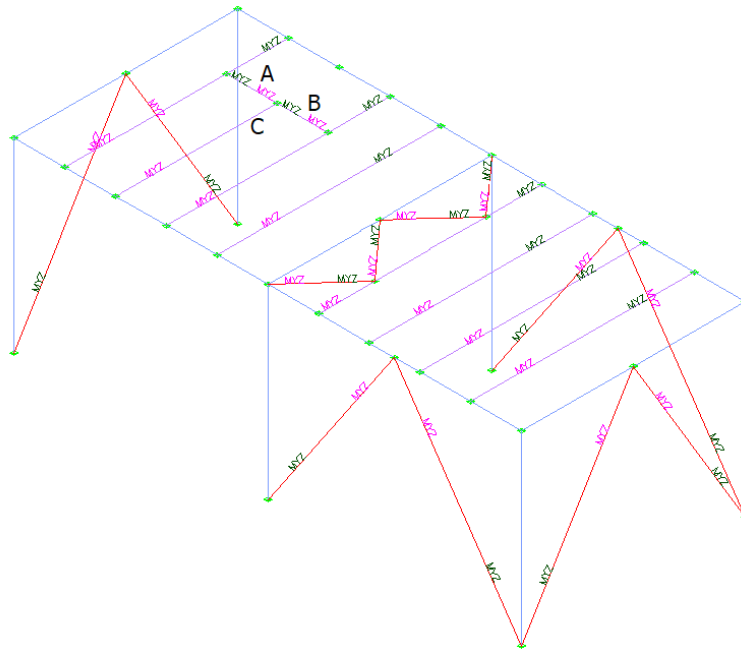
Press the icon  to display the 3D solid view of the model, replacing the wireframe view:



Press the icon  **Frame** to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

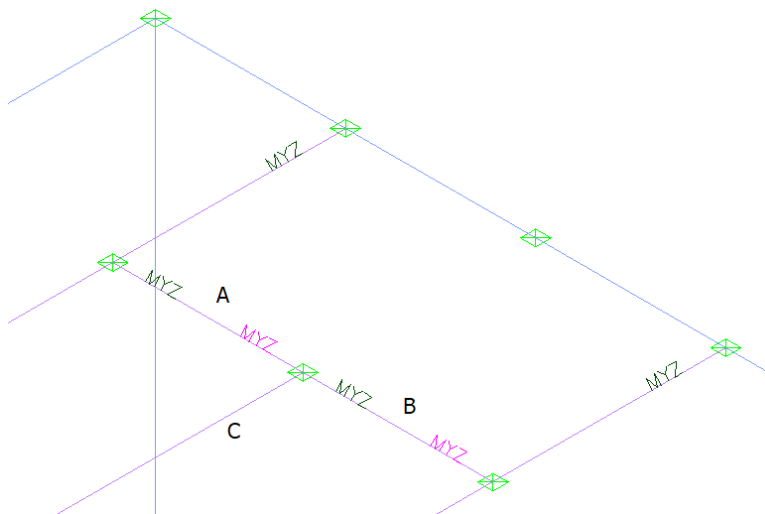
Step #23. Save your Model: In order to save your model just use AutoCAD's or BricsCAD's save command and store the DWG using any filename that you want.

Step #24. Change member releases: Press the  Releases icon (in GTS Display Tab of the ribbon) to show releases for each member.



Click on the icon  Clear to clear the annotations and zoom at the area of the following image.

Again, press the  Releases icon to show releases for each member. Click on the icon  Clear to clear annotations.



Select **member marked in A** by double clicking on it and uncheck the My and Mz Releases of the member at its End. Note that this member belongs to Physical Member PM14.

Member Properties

Model Section Properties Member Loads Member Temperature Loads Member Distortions

General

Name : 53

Level : 1

Type - Incidences

Type : SPACE FRAME

Start : 33

End : 34

Beta Angle : 90

Physical Member : PM14

Groups

☐ Inactive ... Remove

Groups that Member belongs

PM14

Section Properties

Section : W10x45 W-AISC14 AISC 14th

Ax : 0.0923611 Ix : 7.28202e-005

Ay : 0.0245486 Iy : 0.00257524

Az : 0.0460408 Iz : 0.0119598

Sy : 0.00769676 Ey : 0

Sz : 0.0284144 Ez : 0

Yd : 0.841667 Yc : 0.420833

Zd : 0.668333 Zc : 0.334167

Shape Code : 1.0

Material Properties

Material : Steel

E : 4.176e+006 Density : 0.489543

G : 1.58399e+006 CTE : 1.17e-005

Poisson : 0.3

Releases & Elastic Connection spring values

Quick Selection :

Start	Spring	End	Spring
☐ Fx	0	☐ Fx	0
☐ Fy	0	☐ Fy	0
☐ Fz	0	☐ Fz	0
☐ Mx	0	☐ Mx	0
☒ My	0	☐ My	0
☒ Mz	0	☐ Mz	0

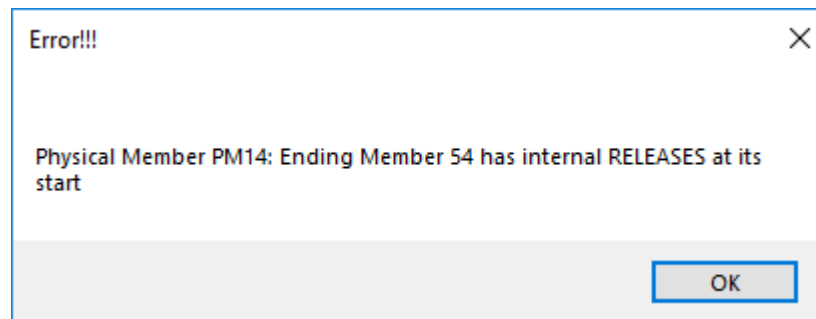
End Sizes OR Member Eccentricities (Offsets)

Start	End
Sizes : 0	0
X : 0	X : 0
Y : 0	Y : 0
Z : 0	Z : 0

OK Cancel Apply

By pressing OK, you get an error message, saying that the Physical Member PM14, has its analytical member 54 (is the member marked in B in the image above) with internal releases at its start. Physical members are now allowed to have internal releases at their internal joints but only at their starting and ending joints.

Note: This error can also be located using the Check Physical Member command as well, as explained in [2.6.26](#) and this command will be used in a following step.



In order to fix the physical member PM14, select **member marked in B** by double clicking on it and uncheck the My and Mz Releases of the member at its Start and press OK.

Member Properties

Model Section Properties Member Loads Member Temperature Loads Member Distortions

General

Name : 54

Level : 1

Type - Incidences

Type : SPACE FRAME

Start : 34

End : 35

Beta Angle : 90

Physical Member : PM14

Groups

☐ Inactive ... Remove

Groups that Member belongs

PM14

Section Properties

Section : W10x45 W-AISC14 AISC 14th

Ax : 0.0923611 Ix : 7.28202e-005

Ay : 0.0245486 Iy : 0.00257524

Az : 0.0460408 Iz : 0.0119598

Sy : 0.00769676 Ey : 0

Sz : 0.0284144 Ez : 0

Yd : 0.841667 Yc : 0.420833

Zd : 0.668333 Zc : 0.334167

Shape Code : 1.0

Material Properties

Material : Steel

E : 4.176e+006 Density : 0.489543

G : 1.58399e+006 CTE : 1.17e-005

Poisson : 0.3

Releases & Elastic Connection spring values

Quick Selection :

Start	Spring	End	Spring
☐ Fx	0	☐ Fx	0
☐ Fy	0	☐ Fy	0
☐ Fz	0	☐ Fz	0
☐ Mx	0	☐ Mx	0
☐ My	0	☒ My	0
☐ Mz	0	☒ Mz	0

End Sizes OR Member Eccentricities (Offsets)

Start	End
Sizes : 0	0
X : 0	X : 0
Y : 0	Y : 0
Z : 0	Z : 0

OK Cancel Apply

No errors are reported now.

Select **member marked in C** by double clicking on it and check the My and Mz Releases of the member at its Start. Note that this member does not belong to any Physical Member. It was belonging to PM7, but when a part of it was deleted (Step #18) the PM7 definition was deleted as well. Press OK to close the properties form.

Member Properties

Model | Section Properties | Member Loads | Member Temperature Loads | Member Distortions

General

Name : 51
Level : 1

Type - Incidences

Type : SPACE FRAME
Start : 34
End : 18
Beta Angle : 90
Physical Member

Groups

☐ Inactive ... Remove

Groups that Member belongs to

Section Properties

Section : W10x45 W-AISC14 AISC 14th ...

Ax : 0.0923611 Ix : 7.28202e-005
Ay : 0.0245486 Iy : 0.00257524
Az : 0.0460408 Iz : 0.0119598
Sy : 0.00769676 Ey : 0
Sz : 0.0284144 Ez : 0
Yd : 0.841667 Yc : 0.420833
Zd : 0.668333 Zc : 0.334167
Shape Code : 1.0

Material Properties

Material : Steel ...

E : 4.176e+006 Density : 0.489543
G : 1.58399e+006 CTE : 1.17e-005
Poisson : 0.3

Releases & Elastic Connection spring values

Quick Selection :

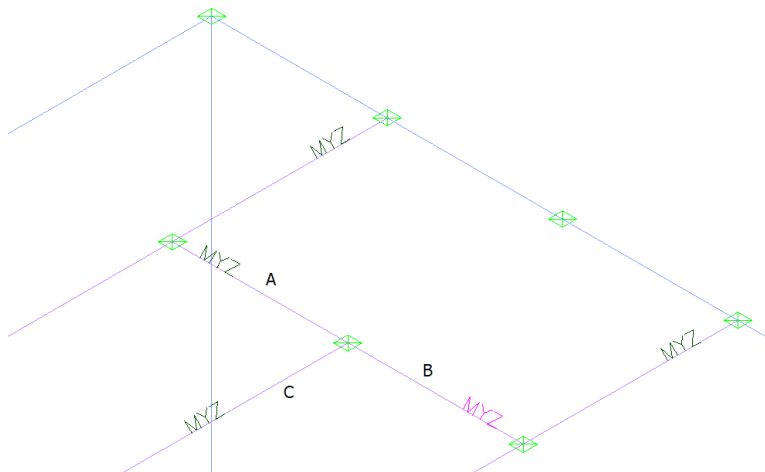
	Start	Spring	End	Spring
☐ Fx		0	☐ Fx	0
☐ Fy		0	☐ Fy	0
☐ Fz		0	☐ Fz	0
☐ Mx		0	☐ Mx	0
☒ My		0	☒ My	0
☒ Mz		0	☒ Mz	0

End Sizes OR Member Eccentricities (Offsets)

	Start	End
Sizes :	0	0
X :	0	0
Y :	0	0
Z :	0	0

OK Cancel Apply

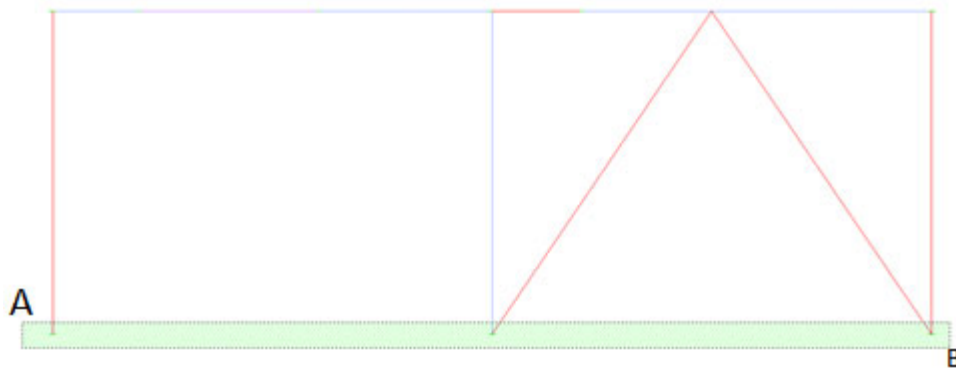
Press the  Releases icon to show releases for each member.



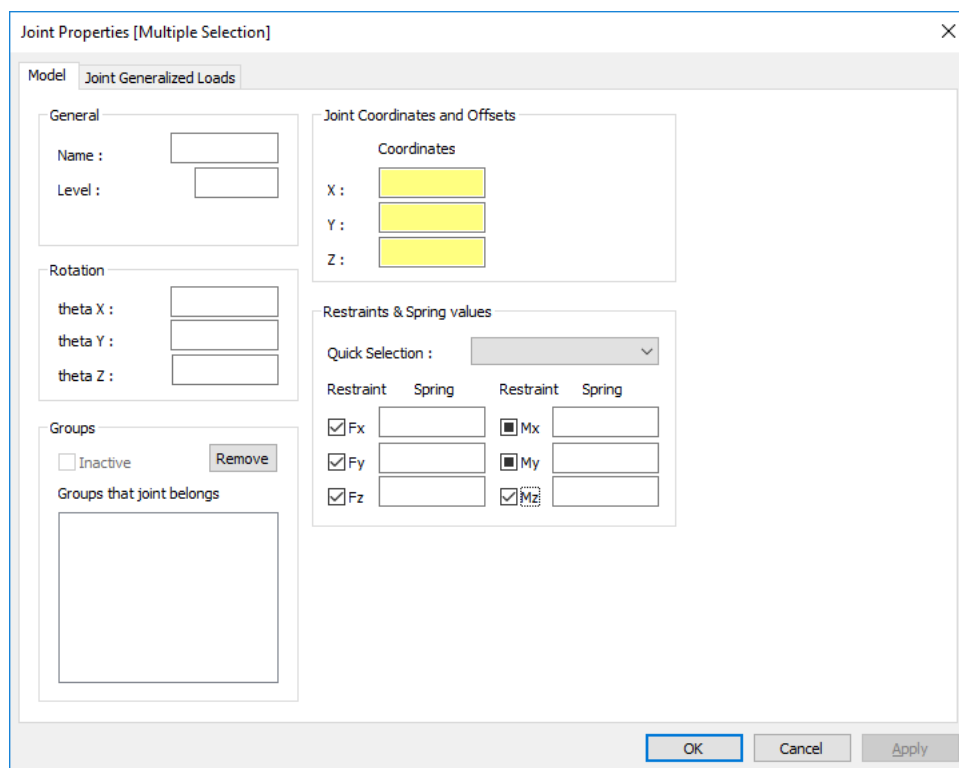
Click on the icon  Clear to clear annotations.

5.7. Define supports

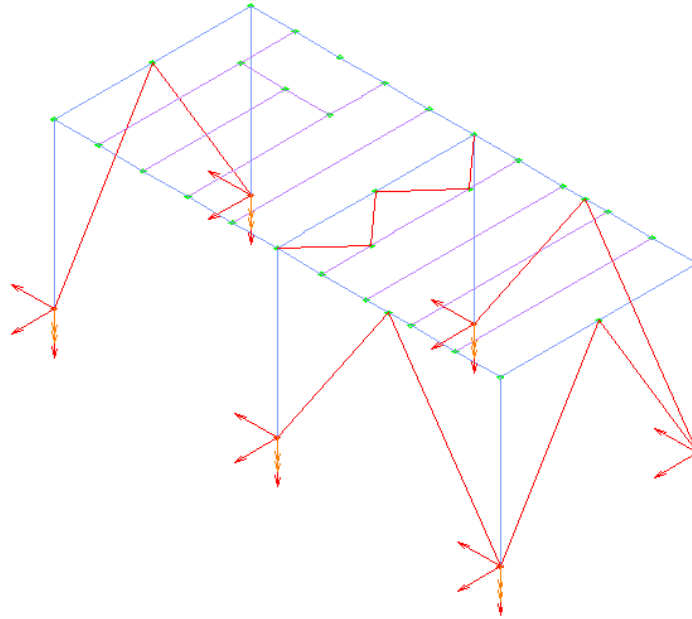
Step #25. Create supports: Use AutoCAD's Cube or BricsCAD's' Chair to change the view to Front. Press the icon  **Support** (in GTS Modeling Tab of the ribbon), select all Joints at the bottom, by clicking at points A and B of the following image, and press ENTER.



At the Joint Properties [Multiple Selection] dialog box select Fx, Fy, Fz and Mz as supports for the selected joints and press OK.



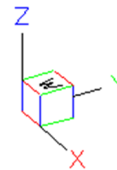
Switch the view to Top Front Right (as in Step #6), and press the icon  Joint Supports (in GTS Display Tab of the ribbon) to view Joints supports.



Click on the icon  Clear to clear annotations and symbols.

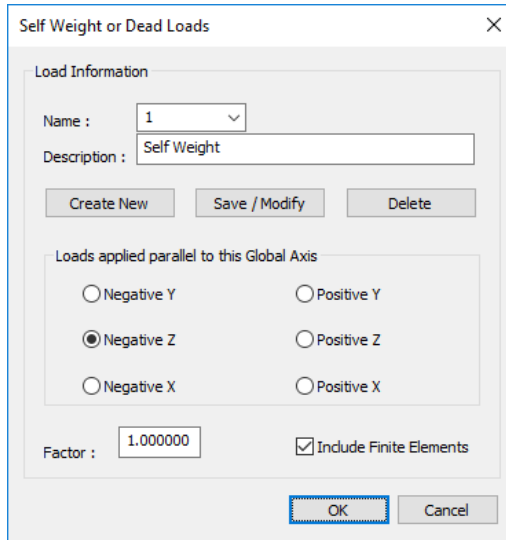
5.8. Define Loads

Step #26. Change UCS: At the command window, type *UCS* and then give *w*, to change to World UCS.



Step #27. Define self-weight: Click on the icon  **Self Weight** to define self-weight load case. Give a Name and a Description. Select Negative Z.

Make sure that the option “Include Finite Elements” is ON.



The dialog box titled "Self Weight or Dead Loads" contains the following fields and options:

- Load Information** section:
 - Name :** A dropdown menu with "1" selected.
 - Description :** A text field containing "Self Weight".
 - Buttons: "Create New", "Save / Modify", and "Delete".
- Loads applied parallel to this Global Axis** section:
 - Radio buttons for: Negative Y, Positive Y, Negative Z (selected), Positive Z, Negative X, and Positive X.
- Factor :** A text field containing "1.000000".
- Include Finite Elements:** A checked checkbox.
- Buttons: "OK" and "Cancel".

5.9. Perform analysis

Step #28. Overview the model and run checks.

Check for duplicate joints: Click on the icon  **Joints Duplicates**, under the “Check” Drop Button, located in Find/Change/Check at Ribbon Area. For the *Merge Tolerance* <0.001000>, just press <ENTER> to accept the default value. You should get the verification that:

```
0 duplicate joints found
```

Check for floating joints: In order to check for joints not connected to the model, click on the icon  **Joints Floatings**, under the “Check” Drop Button, located in Find/Change/Check at Ribbon Area. If your model was created as described so far, you should get a notification:

```
0 floating joints found
```

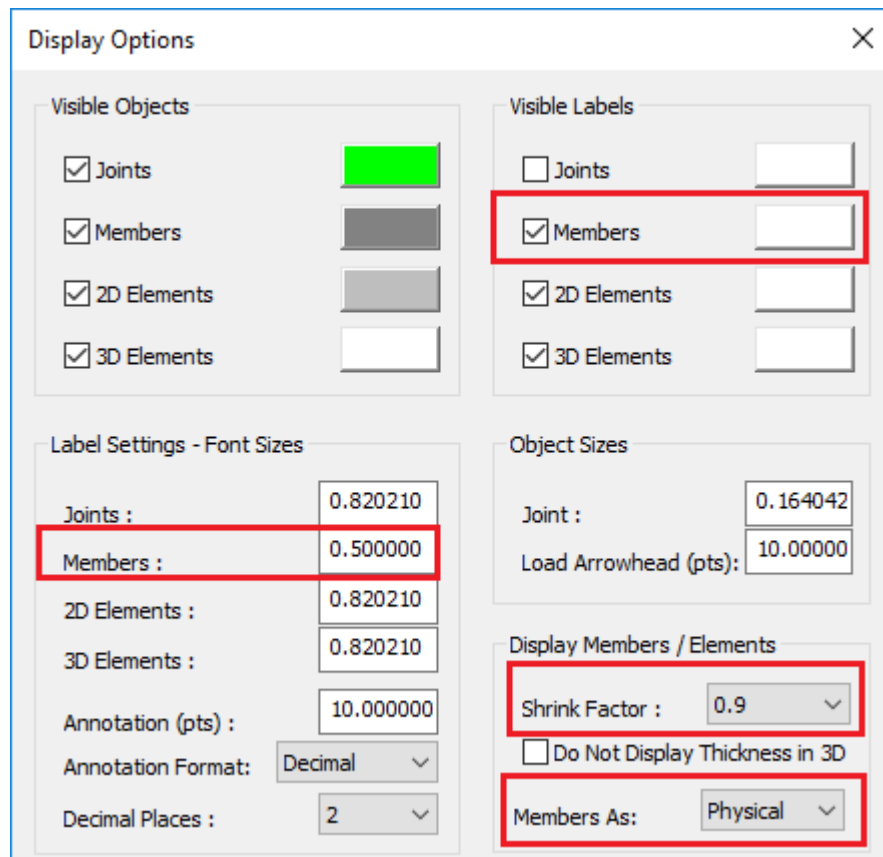
Check Physical Members: Click on the icon  **Physical Members**, under the “Check” Drop Button, located in Find/Change/Check at Ribbon Area.). This check applies to Physical Members having two or more analytical members. The output should be (press F2 to extend the command line height and see all of it.

```
Physical Member PM1: OK
Physical Member PM2: OK
Physical Member PM3: OK
Physical Member PM5: OK
Physical Member PM6: OK
Physical Member PM8: OK
Physical Member PM10: OK
```


Physical Member PM14: OK
0 Errors Found

Note: You can also run all other checks of the same drop list, to check for Interference Joints, Duplicate Members, Zero Length Members, Duplicate Names and Database Integrity. You should not get any errors or warnings.

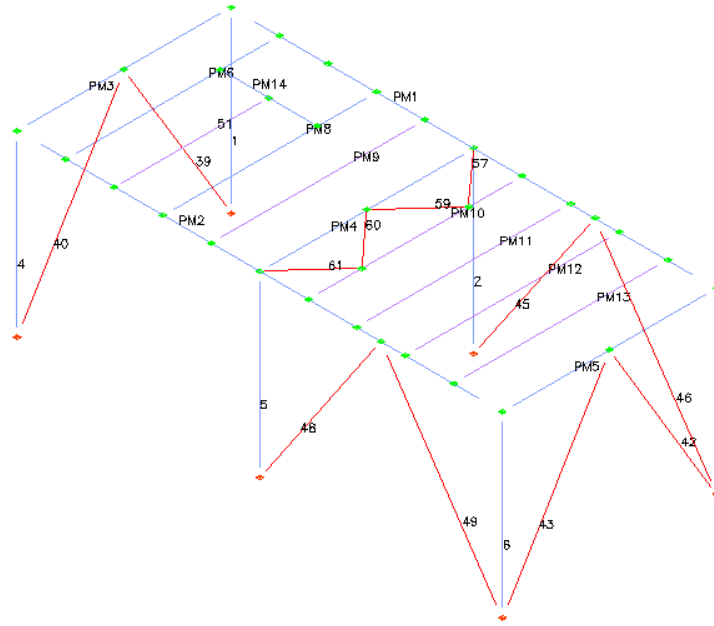
View physical members: Click on the icon  **Options** in the ribbon bar and in the Display Options Form check Visible Labels for Members, set Font Size for Members equal to 0.5, Set Shrink Factor to 0.9 and display members as “Physical”.

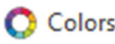


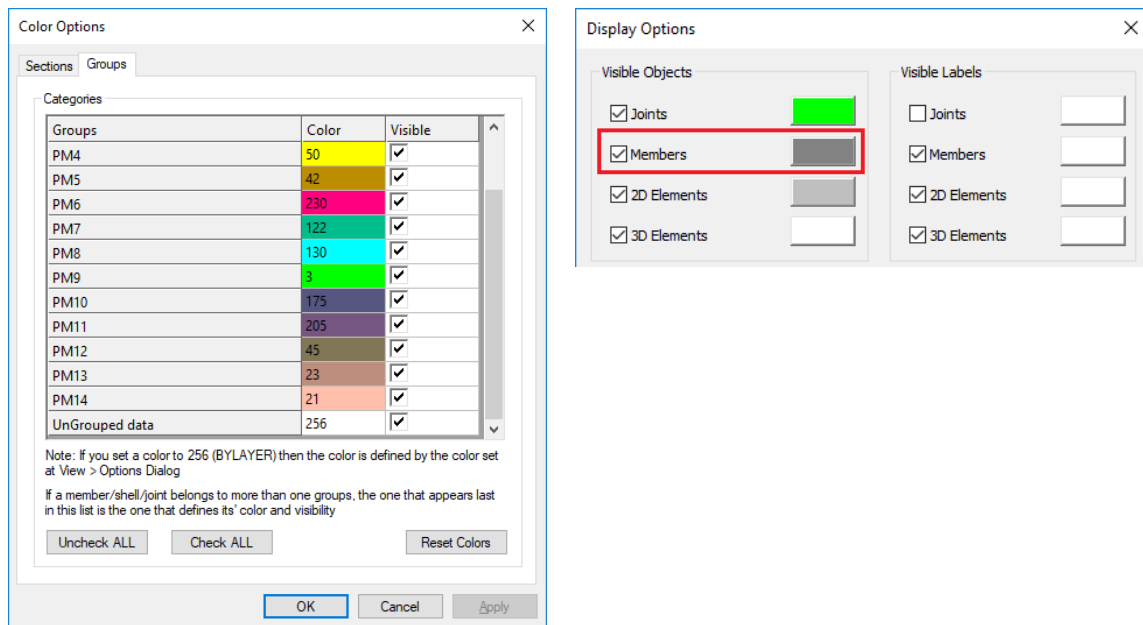
The screenshot shows the 'Display Options' dialog box with the following settings highlighted by red boxes:

- Visible Objects:** Joints, Members, 2D Elements, and 3D Elements are all checked.
- Visible Labels:** Members is checked.
- Label Settings - Font Sizes:** Members font size is set to 0.500000.
- Object Sizes:** Joint size is set to 0.164042 and Load Arrowhead (pts) is set to 10.00000.
- Display Members / Elements:** Shrink Factor is set to 0.9.
- Members As:** Set to Physical.

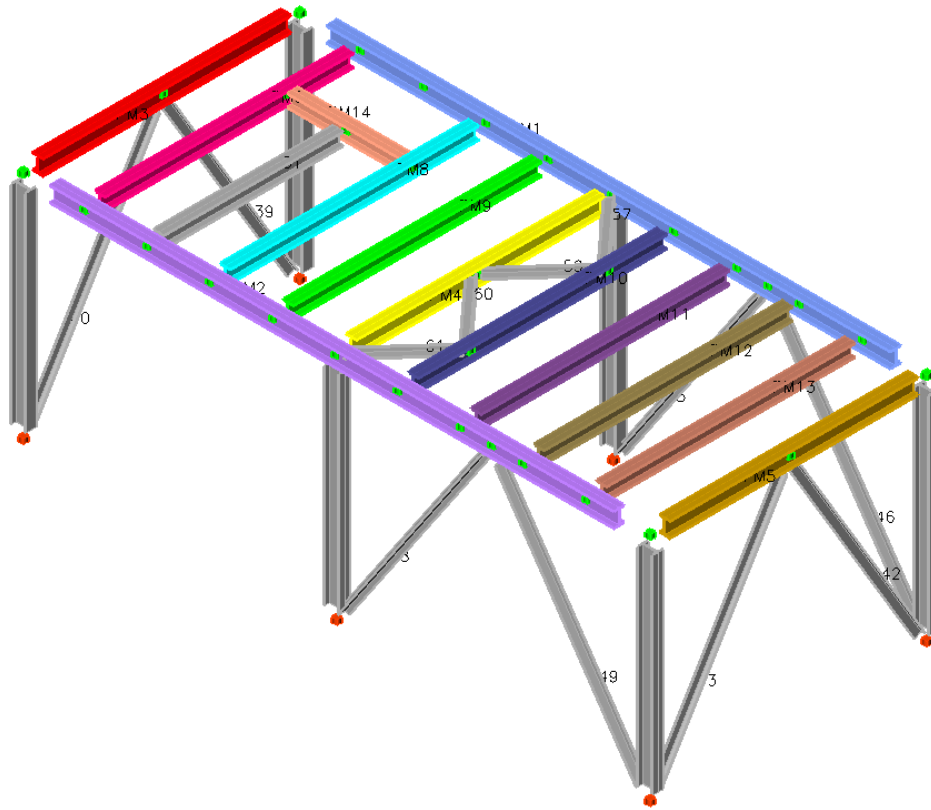
Press OK to close the form and to see the physical members in wireframe view.



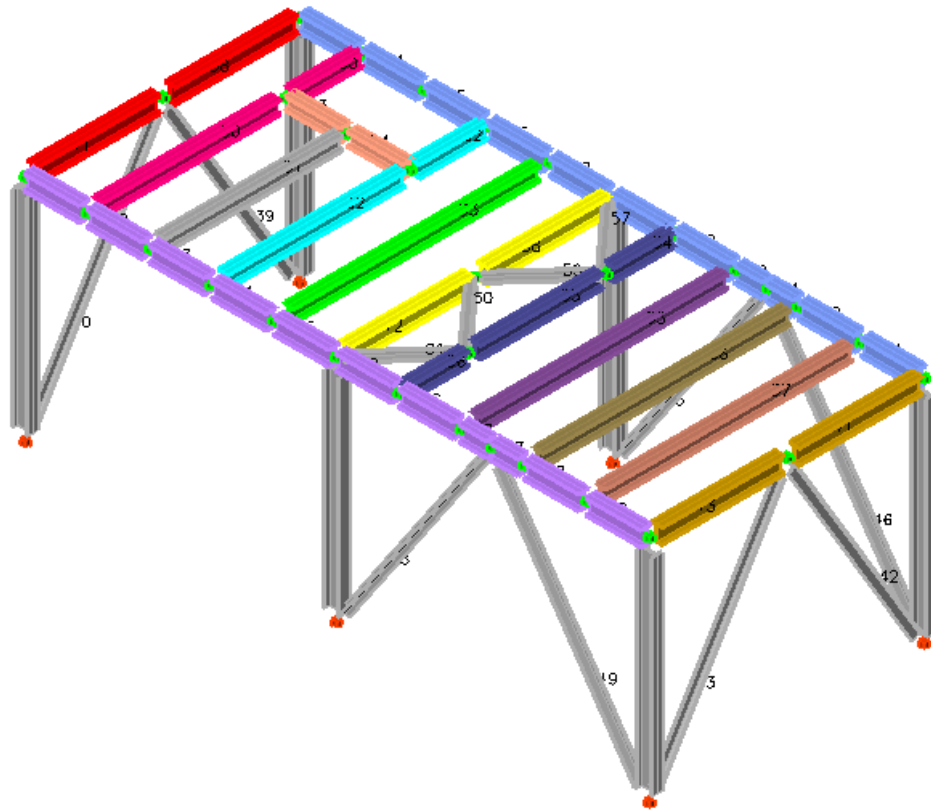
Click on the icon  and select the 2nd Tab in order to colorize members by their group. Member do not belonging to any Physical Member are in the “Ungrouped data”. Their default color is BYLAYER (256), meaning use the color from Display Options Form, but you can set any other color. Press OK to close the Color Options Form.



Now each physical member has a different color and you can also press icon  to display the 3D solid view, as in the image bellow.



By clicking on the icon  Analytical/Physical , you can switch between physical view (image above) and analytical view (image below). Note the difference between the physical view, where each physical member is a single object, and the analytical view, where you can control each analytical member individually.



Click on the icon  **Options** and turn off labels for members, set shrink factor back to 1.0, display members as “Analytical” and press OK.

Also click on the icon  **Colors** and select the 1st Tab in order to colorize members by their section and press OK.

Press the icon  **Frame** to switch back to wireframe view.

Step #29. Create input file: Click on the icon  to create GTI input file. At the dialog box “Create GT.STRUDL Input File” check the following and click OK.

- Perform Stiffness Analysis
- Read Joint Displacements
- Read Member Forces
- Read Section Forces
- Read Section Displacements

Create GT.STRU DL Input File

Export to GTI

Create GTI File : D:\GTSTRU DL\CAD_Modeler\Examples\Example3\Example3.gti ...

☒ Export Whole Model
 ☐ Export Portion of the Model
 (will be prompted for Selection)

☒ Perform Stiffness Analysis

Append Other GTI Files/Macros

☐ Copy Commands from GTI Files/Macros (not CINPUT)

Create Commands to Read Results

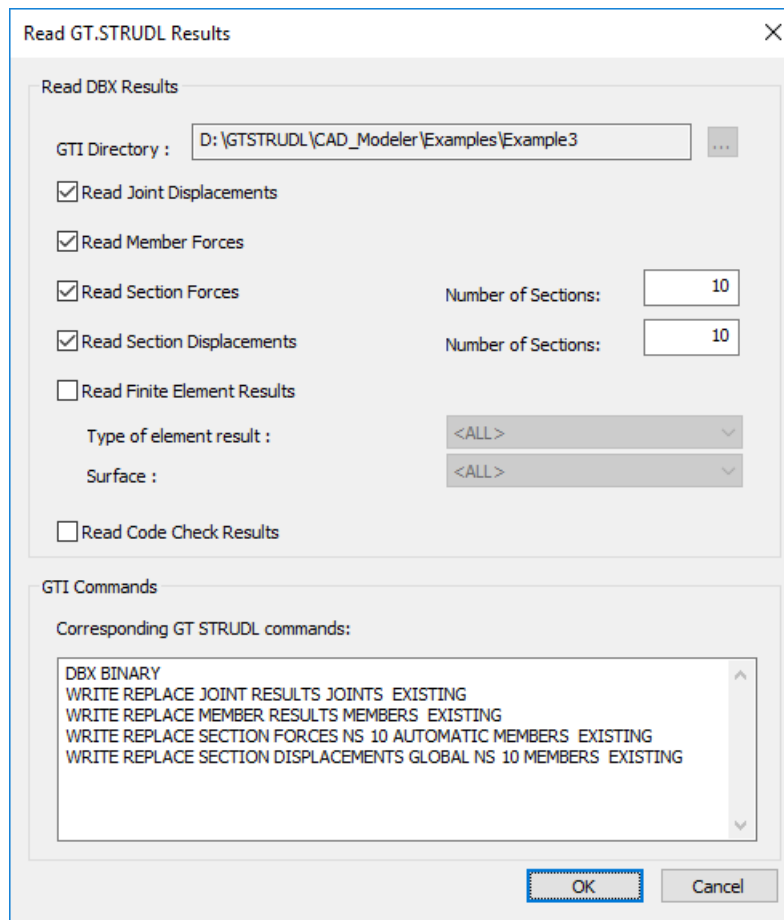
☒ Read Joint Displacements
☒ Read Member Forces
☒ Read Section Forces Number of Sections: 10
☒ Read Section Displacements Number of Sections: 10
☐ Read Finite Element Results
☐ Read Code Check Results

OK Cancel

Step #30. View/edit input file: Click on the icon  Edit GTI to view/edit GTI input file.

5.10. Read analysis results

Step #32. Read GT.STRUDL results: Click on the icon  **Read Results** to read GT.STRUDL results.



The dialog box is titled "Read GT.STRUDL Results". It contains two main sections: "Read DBX Results" and "GTI Commands".

Read DBX Results:

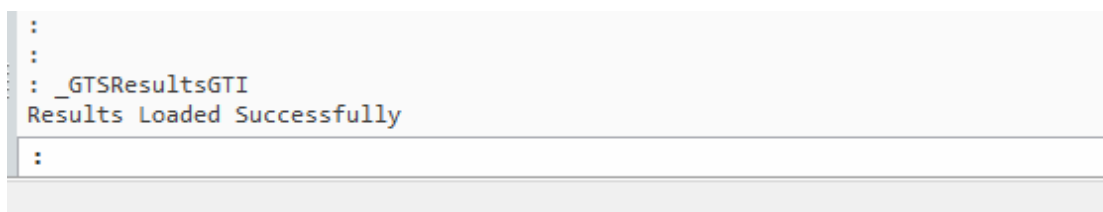
- GTI Directory : D:\GTSTRUDL\CAD_Modeler\Examples\Example3
- ☒ Read Joint Displacements
- ☒ Read Member Forces
- ☒ Read Section Forces (Number of Sections: 10)
- ☒ Read Section Displacements (Number of Sections: 10)
- ☐ Read Finite Element Results
- Type of element result : <ALL>
- Surface : <ALL>
- ☐ Read Code Check Results

GTI Commands:

Corresponding GT STRUDL commands:

```
DBX BINARY
WRITE REPLACE JOINT RESULTS JOINTS EXISTING
WRITE REPLACE MEMBER RESULTS MEMBERS EXISTING
WRITE REPLACE SECTION FORCES NS 10 AUTOMATIC MEMBERS EXISTING
WRITE REPLACE SECTION DISPLACEMENTS GLOBAL NS 10 MEMBERS EXISTING
```

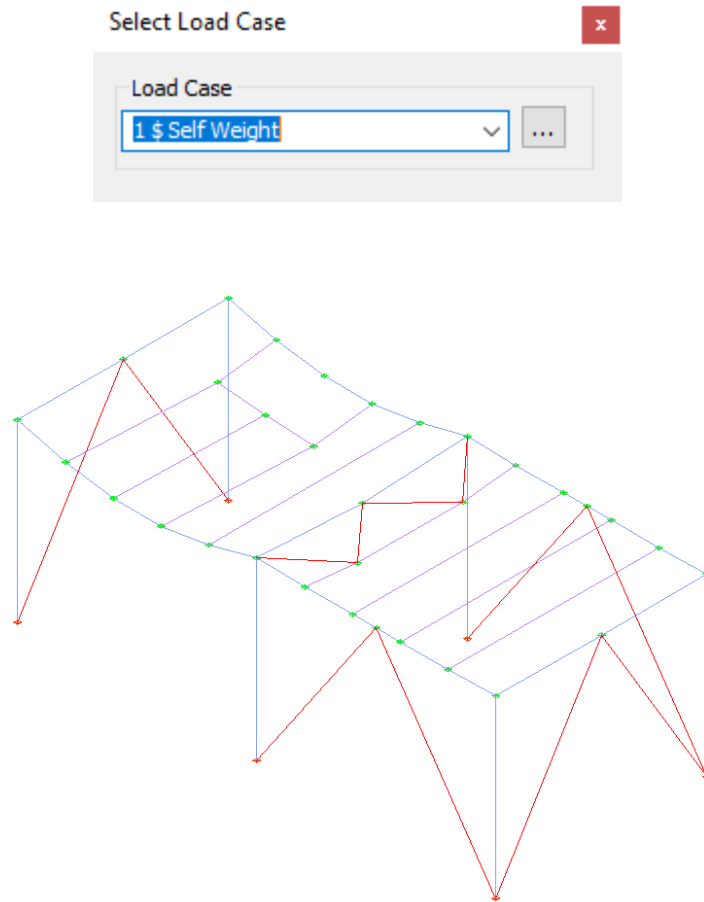
Buttons: OK, Cancel



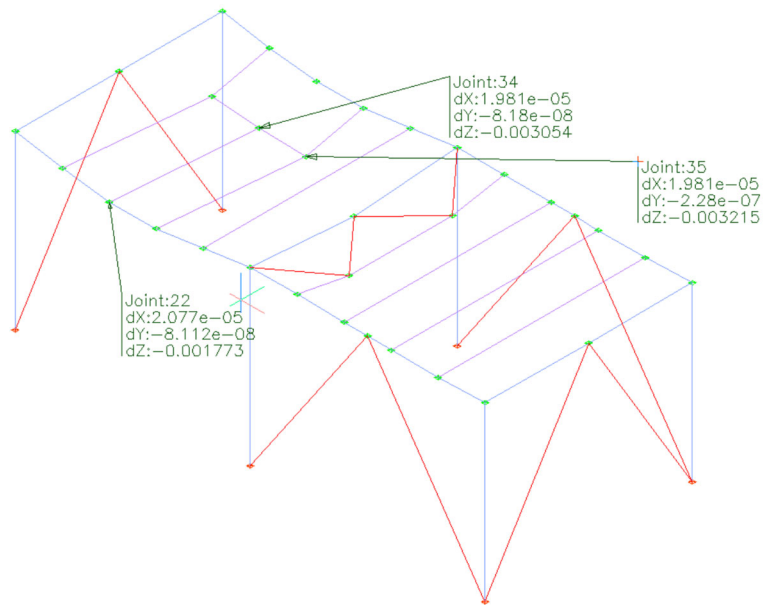
```
:
:
: _GTSResultsGTI
Results Loaded Successfully
:
```

5.11. Display analysis results

Step #33. Graphical display of analysis results: Click on the icon  **Deformed**, under the "Deformation" Drop Button, located in "Display Results" at Ribbon Area, to display deformed shape of the model for Self Weight load case and press ENTER two times.



Step #34. Annotate displacements: Click on the icon  **Annotate Displacements**, under the “Deformation” Drop Button, located in “Display Results” at Ribbon Area, to annotate displacements of the deformed model at specific Joints of interest.



Click on the icon  **Clear** to clear additional generated symbols. Click on the icon  **Undeformed** to display undeformed model again.

Step #35. Display Section Displacements: Click on  **Displacements** (ribbon tab “GTS Display”).

Displacements

Load Case / Load Combination

1 \$ Self Weight

Display Options

Scale Factor (Values) :
0.100

Font Size (pt) :
10.00

Annotation Format:
Exponential

Decimal Places :
2

☒ Hide Model

Display >>

Annotate >
Legend >

Animation Options

Frames :
7

Animation Speed % :
1

Generate Animation Frames
Animate >>

Clear
Close

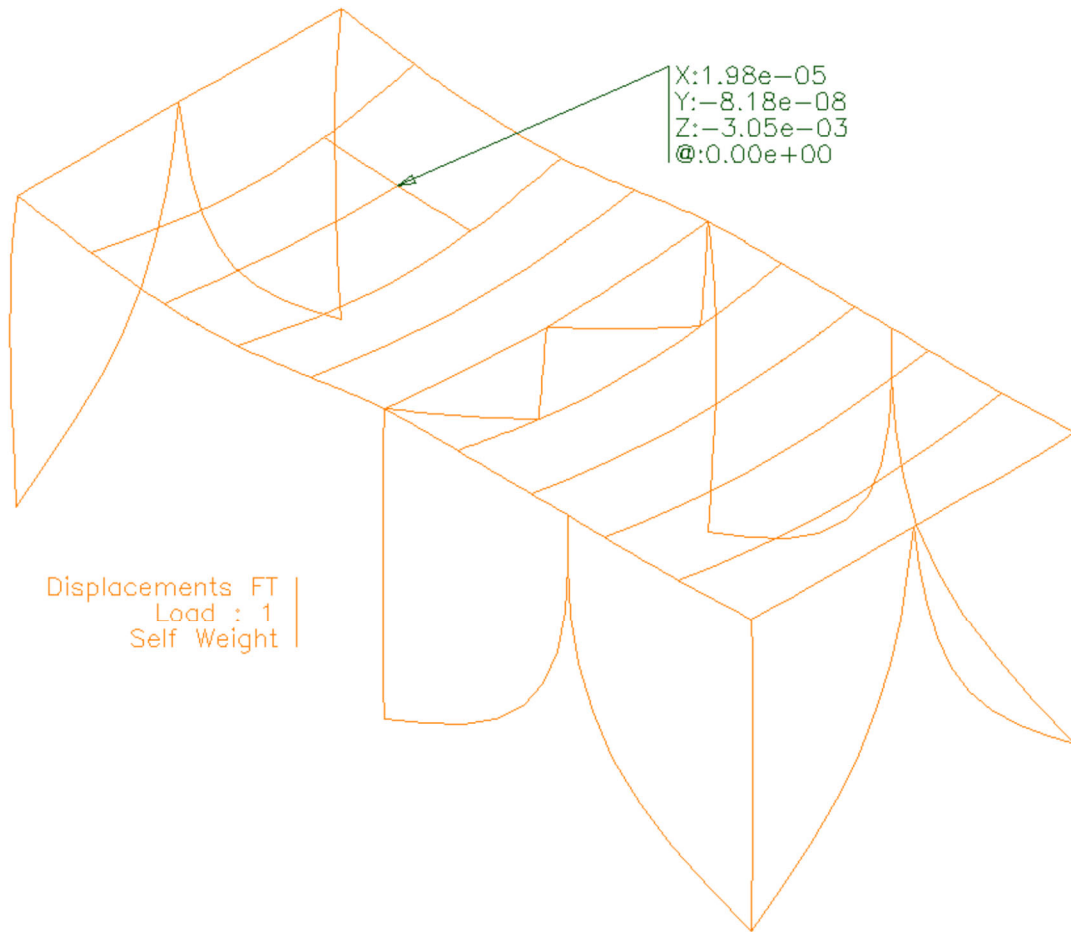
Select:

- 1 as Load Case
- 0.1 as Scale Factor
- 10.00 as Font Size (default)
- Annotation Format: Exponential
- *Check Hide Model*

Press “Display >>” and zoom at the upper left end of the structure as shown in the image below.

Press “Annotate >” and click on any part of the deformed shape curve and then at the position that you want the annotation to be displayed.

Press on “Legend >” and click at any part of the screen to place the legend of the diagram.



In order to display the animation, press “Generate Animation Frames” and then “Animate >>”. To terminate the animation press “Stop” button.

In order to exit the command, uncheck “Hide Model”, press “Clear” Button and “Close”.

Step #36. View member diagrams: Click on the icon  **Diagrams** to view member diagrams. In the dialog box “Member Diagrams”, select *MZ Moment* as Values, 2 as Decimal Places and click on “Display”.

Member Diagrams

Load Case / Load Combination

1 \$ Self Weight

☐ Envelope

Values

MZ Moment

Display Options

Scale Factor (Values)

0.100

Font Size (pt)

10.00

Annotation Format:

Decimal

Decimal Places :

2

☐ Label Max & Min

☐ Positive Sign

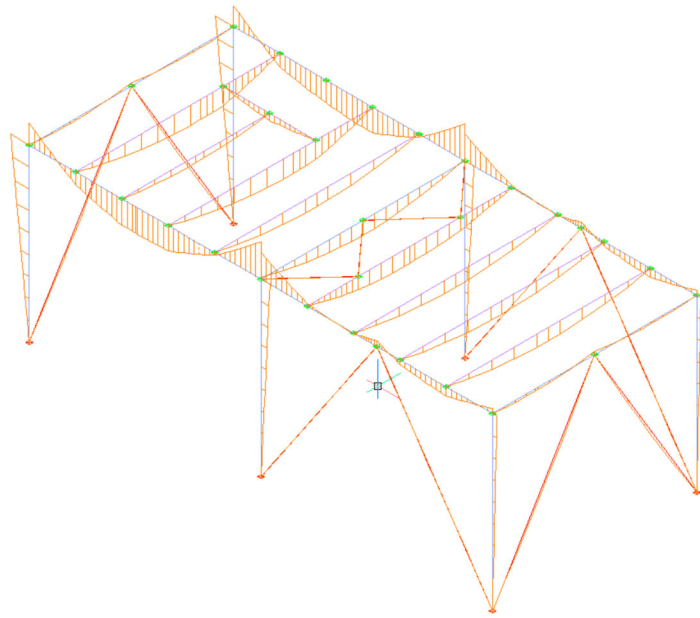
Display >>

Annotate >

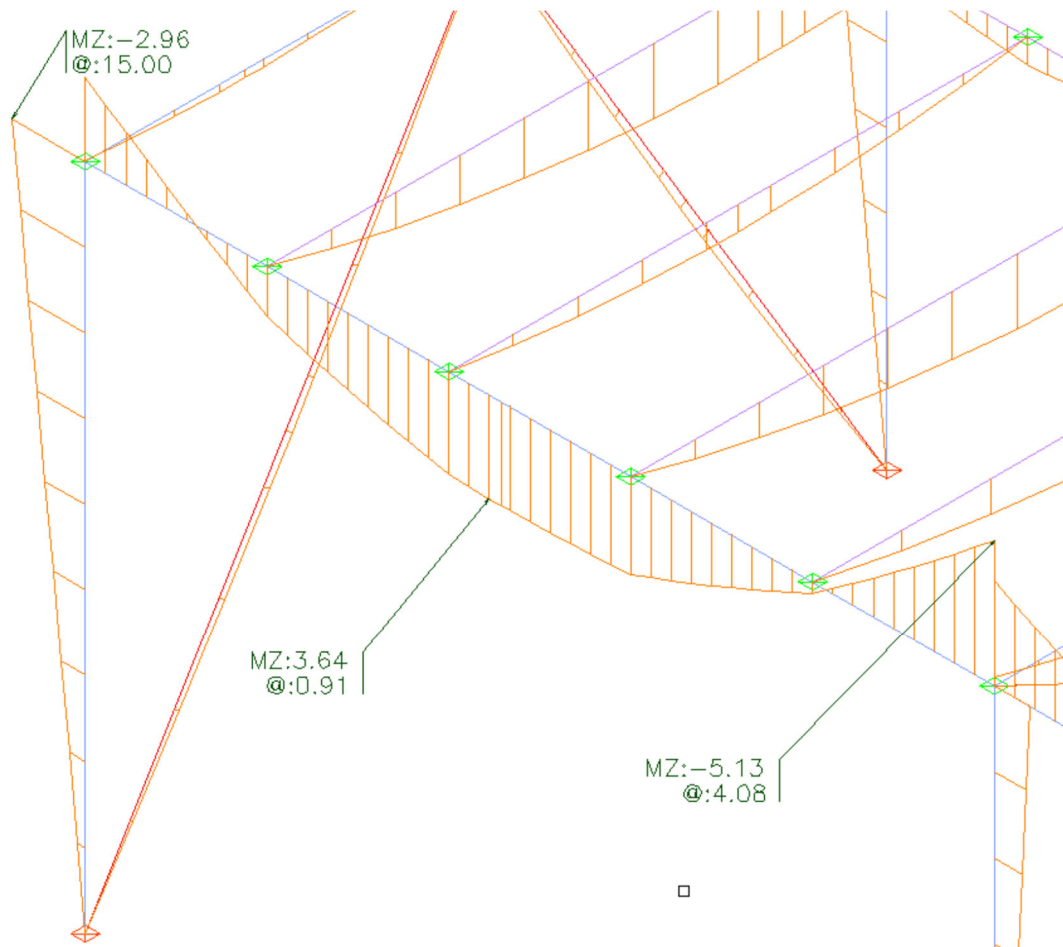
Legend >

Clear

Close



You can also annotate moments at specific Joints by clicking “Annotate” in the dialog box “Member Diagrams” and select the desired precision (decimal places) as well as the annotation format (decimal, exponential, generic).



Select “Clear” and “Close” to clear the diagram/annotations and terminate the command.

Step #37. Display results in datasheets: Click on the icon  **Displacements**, on Results Datasheets area, to view analysis results in datasheets.

GTSTRUDL - Joint Displacement Datasheet, display Units: Feet Degrees									
File	Edit	Columns	Filter	Sort	Units	Help			
Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y	Rotation Z		
1	1	0.0000	0.0000	0.0000	-0.0003399	-0.0048971	0.0000000		
2	1	0.0000	-0.0000	-0.0001	0.0006845	0.0107201	0.0000026		
3	1	0.0000	0.0000	0.0000	-0.0039857	0.0024911	0.0000000		
4	1	0.0000	-0.0000	-0.0001	0.0080282	-0.0051456	0.0000039		
5	1	0.0000	0.0000	0.0000	-0.0004337	0.0012434	0.0000000		
6	1	0.0000	-0.0000	-0.0000	0.0008740	-0.0025264	0.0000005		
7	1	0.0000	0.0000	0.0000	0.0003399	-0.0050672	0.0000000		
8	1	0.0000	0.0000	-0.0001	-0.0006842	0.0111001	0.0000014		
9	1	0.0000	0.0000	0.0000	0.0039877	0.0025457	0.0000000		
10	1	0.0000	0.0000	-0.0001	-0.0080274	-0.0052484	0.0000022		
11	1	0.0000	0.0000	0.0000	0.0004335	0.0012498	0.0000000		
12	1	0.0000	0.0000	-0.0000	-0.0008738	-0.0025280	0.0000019		
13	1	0.0000	0.0000	-0.0011	0.0021532	0.0120130	-0.0000007		
14	1	0.0000	-0.0000	-0.0017	0.0036220	0.0044458	-0.0000022		
15	1	0.0000	-0.0000	-0.0017	0.0050907	-0.0056013	-0.0000005		
16	1	0.0000	-0.0000	-0.0009	0.0065595	-0.0117479	-0.0000049		
17	1	0.0000	0.0000	-0.0001	0.0065356	0.0006018	0.0000072		
18	1	0.0000	0.0000	-0.0001	0.0050429	0.0007610	-0.0000009		
19	1	0.0000	0.0000	-0.0002	0.0036121	0.0013782	-0.0000009		
20	1	0.0000	0.0000	-0.0003	0.0022431	-0.0015496	-0.0000012		
21	1	0.0000	0.0000	-0.0011	-0.0021529	0.0126868	-0.0000010		
22	1	0.0000	-0.0000	-0.0018	-0.0036215	0.0045080	-0.0000018		
23	1	0.0000	-0.0000	-0.0017	-0.0050902	-0.0061676	-0.0000028		
24	1	0.0000	-0.0000	-0.0009	-0.0065588	-0.0120713	0.0000048		
25	1	0.0000	0.0000	-0.0001	-0.0065349	0.0005932	-0.0000016		
26	1	0.0000	0.0000	-0.0001	-0.0050424	0.0007877	0.0000009		
27	1	0.0000	0.0000	-0.0002	-0.0036117	0.0013902	-0.0000014		
28	1	0.0000	0.0000	-0.0003	-0.0022428	-0.0015499	-0.0000004		
29	1	0.0000	-0.0000	-0.0001	0.0000001	0.0106267	0.0000059		
30	1	0.0000	0.0000	-0.0001	0.0000001	-0.0024599	0.0000042		
31	1	0.0000	0.0000	-0.0002	0.0042966	0.0012625	-0.0000008		
32	1	0.0000	0.0000	-0.0002	-0.0042961	0.0012849	-0.0000007		
33	1	0.0000	0.0000	-0.0026	0.0116327	0.0122002	-0.0000086		
34	1	0.0000	-0.0000	-0.0031	0.0116465	0.0042854	-0.0000017		
35	1	0.0000	-0.0000	-0.0032	0.0116604	-0.0057586	0.0000017		
36	1	0.0000	0.0000	-0.0012	0.0107657	-0.0003520	0.0000168		
37	1	0.0000	0.0000	-0.0012	-0.0107855	-0.0003590	-0.0000098		
38	1	0.0000	-0.0000	-0.0010	0.0000003	-0.0037684	0.0000039		

Click on the icon  **Member Forces** to view member forces results in datasheets.

GTSTRUDL - Member End Force Datasheet, display Units: Feet Kips									
File	Edit	Columns	Filter	Sort	Units	Help			
Member	Load	Joint	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z	
1	1	1	2.1786	-0.1908	0.0018	-0.0000	-0.0000	-0.0001	
1	1	2	-1.4341	0.1908	-0.0018	0.0000	-0.0270	-2.8618	
2	1	3	3.6514	0.0933	0.0211	-0.0000	-0.0000	0.0000	
2	1	4	-2.9068	-0.0933	-0.0211	0.0000	-0.3170	1.3994	
3	1	5	1.4975	0.0461	0.0023	-0.0000	-0.0000	0.0000	
3	1	6	-0.7530	-0.0461	-0.0023	0.0000	-0.0345	0.6908	
4	1	7	2.1796	-0.1975	-0.0018	-0.0000	-0.0000	-0.0001	
4	1	8	-1.4351	0.1975	0.0018	0.0000	0.0270	-2.9626	
5	1	9	3.6544	0.0952	-0.0211	-0.0000	0.0000	-0.0000	
5	1	10	-2.9099	-0.0952	0.0211	0.0000	0.3171	1.4282	
6	1	11	1.4984	0.0462	-0.0023	-0.0000	0.0000	0.0000	
6	1	12	-0.7539	-0.0462	0.0023	0.0000	0.0345	0.6923	
7	1	2	0.1908	1.2561	-0.0000	-0.0008	0.0002	2.8618	
7	1	13	-0.1908	-1.0535	0.0000	0.0008	-0.0002	1.8536	
14	1	13	0.1906	0.4071	-0.0000	-0.0008	0.0002	-1.8537	
14	1	14	-0.1906	-0.2044	0.0000	0.0008	0.0000	3.1021	
15	1	14	0.1906	0.2044	-0.0000	-0.0008	-0.0000	-3.1021	
15	1	15	-0.1906	-0.0017	0.0000	0.0008	0.0002	3.5230	
16	1	15	0.1908	-0.6447	0.0002	-0.0008	-0.0002	-3.5229	
16	1	16	-0.1908	0.8474	-0.0002	0.0008	-0.0006	0.4767	
17	1	16	0.1908	-1.2543	-0.0005	-0.0008	0.0006	-0.4766	
17	1	4	-0.1908	1.4570	0.0005	0.0008	0.0014	-5.0588	
8	1	4	0.0975	0.9123	0.0005	0.0008	-0.0011	3.6590	
8	1	17	-0.0975	-0.7096	-0.0005	-0.0008	-0.0008	-0.3476	
18	1	17	0.0976	0.2118	-0.0002	0.0008	0.0008	0.3481	
18	1	18	-0.0976	-0.0091	0.0002	-0.0008	0.0000	0.1030	
20	1	19	0.0460	0.2453	-0.0000	0.0008	0.0000	-0.6225	
20	1	20	-0.0460	-0.0427	0.0000	-0.0008	0.0000	1.2104	
21	1	20	0.0460	-0.3643	-0.0000	0.0008	-0.0000	-1.2104	
21	1	6	-0.0460	0.5669	0.0000	-0.0008	0.0001	-0.6908	
9	1	8	0.1975	1.2571	-0.0000	0.0008	0.0002	2.9627	
9	1	21	-0.1975	-1.0545	0.0000	-0.0008	-0.0000	1.7568	

Click on the icon  **Section Forces** to view section forces results in datasheets.

GTSTRUDL - Section Force Datasheet, display Units: Feet Kips									
Member	Load	Section	Axial	Y shear	Z shear	Torsion	Y bending	Z bending	
1	1	0.000	-2.1786	0.1908	-0.0018	0.0000	0.0000	0.0001	
1	1	0.111	-2.0959	0.1908	-0.0018	0.0000	-0.0030	-0.3179	
1	1	0.222	-2.0132	0.1908	-0.0018	0.0000	-0.0060	-0.6359	
1	1	0.333	-1.9304	0.1908	-0.0018	0.0000	-0.0090	-0.9539	
1	1	0.444	-1.8477	0.1908	-0.0018	0.0000	-0.0120	-1.2719	
1	1	0.556	-1.7650	0.1908	-0.0018	0.0000	-0.0150	-1.5899	
1	1	0.667	-1.6823	0.1908	-0.0018	0.0000	-0.0180	-1.9079	
1	1	0.778	-1.5995	0.1908	-0.0018	0.0000	-0.0210	-2.2258	
1	1	0.889	-1.5168	0.1908	-0.0018	0.0000	-0.0240	-2.5438	
1	1	1.000	-1.4341	0.1908	-0.0018	0.0000	-0.0270	-2.8618	
2	1	0.000	-3.6514	-0.0933	-0.0211	0.0000	0.0000	-0.0000	
2	1	0.111	-3.5686	-0.0933	-0.0211	0.0000	-0.0352	0.1555	
2	1	0.222	-3.4859	-0.0933	-0.0211	0.0000	-0.0704	0.3110	
2	1	0.333	-3.4032	-0.0933	-0.0211	0.0000	-0.1056	0.4665	
2	1	0.444	-3.3205	-0.0933	-0.0211	0.0000	-0.1409	0.6219	
2	1	0.556	-3.2377	-0.0933	-0.0211	0.0000	-0.1761	0.7774	
2	1	0.667	-3.1550	-0.0933	-0.0211	0.0000	-0.2113	0.9329	
2	1	0.778	-3.0723	-0.0933	-0.0211	0.0000	-0.2466	1.0884	
2	1	0.889	-2.9896	-0.0933	-0.0211	0.0000	-0.2818	1.2439	
2	1	1.000	-2.9068	-0.0933	-0.0211	0.0000	-0.3170	1.3994	
3	1	0.000	-1.4975	-0.0461	-0.0023	0.0000	0.0000	-0.0000	
3	1	0.111	-1.4148	-0.0461	-0.0023	0.0000	-0.0038	0.0767	
3	1	0.222	-1.3321	-0.0461	-0.0023	0.0000	-0.0077	0.1535	
3	1	0.333	-1.2494	-0.0461	-0.0023	0.0000	-0.0115	0.2303	
3	1	0.444	-1.1666	-0.0461	-0.0023	0.0000	-0.0153	0.3070	
3	1	0.556	-1.0839	-0.0461	-0.0023	0.0000	-0.0192	0.3838	
3	1	0.667	-1.0012	-0.0461	-0.0023	0.0000	-0.0230	0.4605	
3	1	0.778	-0.9185	-0.0461	-0.0023	0.0000	-0.0269	0.5373	
3	1	0.889	-0.8357	-0.0461	-0.0023	0.0000	-0.0307	0.6141	
3	1	1.000	-0.7530	-0.0461	-0.0023	0.0000	-0.0345	0.6908	
4	1	0.000	-2.1796	0.1975	0.0018	0.0000	0.0000	0.0001	
4	1	0.111	-2.0969	0.1975	0.0018	0.0000	0.0030	-0.3291	
4	1	0.222	-2.0141	0.1975	0.0018	0.0000	0.0060	-0.6583	

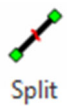
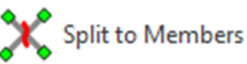
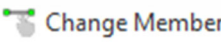
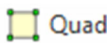
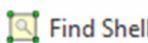
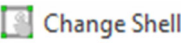
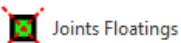
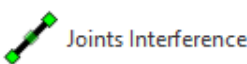
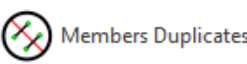
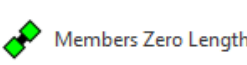
Click on the icon  **Reactions** to view reaction results in datasheets.

GTSTRUDL - Joint Reactions Datasheet, display Units: Feet Kips								
Joint	Load	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z	
1	1	0.191	-0.242	2.709	-0.000	0.000	0.000	
3	1	0.400	-0.021	4.511	-0.000	-0.000	-0.000	
5	1	-0.590	-0.237	2.955	-0.000	-0.000	0.000	
7	1	0.198	0.242	2.710	0.000	-0.000	-0.000	
9	1	0.393	0.021	4.508	0.000	-0.000	0.000	
11	1	-0.591	0.237	2.957	0.000	-0.000	-0.000	

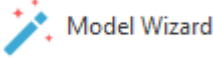
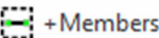
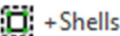
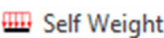
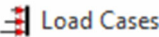
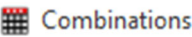
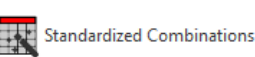
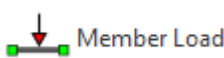
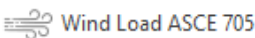
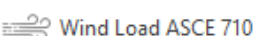
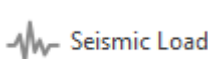
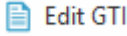
Step #37. Display results in Report Builder: Click on the icon  to view results in Report Builder.

6. Appendix – List of Commands

Command	Icon	Menu	Command Prompt	Link
Units	 Units	<i>GTS Modeling>Units</i>	GTSUnits	2.6.1
Materials	 Materials	<i>GTS Modeling>Materials</i>	GTSMaterials	2.6.2
Sections	 Sections	<i>GTS Modeling>Cross Sections>Table</i>	GTSParams	2.6.9
Prismatic Sections	-	<i>GTS Modeling>Cross Sections>Prismatic</i>	GTSPrismatic	2.6.9
Levels	 Levels	<i>GTS Modeling>Levels</i>	GTSLevels	2.6.3
Higher Level	 Higher Level	-	GTSLevelUp	2.6.3
Lower Level	 Lower Level	-	GTSLevelDown	2.6.3
All Levels ON	 All Levels ON	<i>GTS Display>All Levels ON</i>	GTSSetAllVisible	2.6.3
Grid	 Grid	<i>GTS Modeling>Grid>Create</i>	GTSGrid	2.6.4
Change Grid	 Change Grid	<i>GTS Modeling>Grid>Change</i>	GTSGridChange	2.6.4
Generate Joint	 Generate	<i>GTS Modeling>Joint>Generate Joint</i>	GTSJoint	2.6.5
At Level (Joint)	 At Level	<i>GTS Modeling>Joint>Generate Joint at Level</i>	GTSJointLevel	2.6.5
Find (Joint)	 Find Joint	<i>GTS Modeling>Joint>Find</i>	GTSFJID	2.6.6
Support	 Support	<i>GTS Modeling>Joint>Support</i>	GTSJointSupport	2.6.7
Change (Joint)	 Change Joint	<i>GTS Modeling>Joint>Change</i>	GTSJointChange	2.6.8
Generate (Member)	 Generate	<i>GTS Modeling>Members>Generate Beam Members</i>	GTSBeam	2.6.10

Vertical (Member)	 Vertical	<i>GTS Modeling> Member>Generate Vertical Member</i>	GTSColumn	2.6.10
Find (Member)	 Find Member	<i>GTS Modeling>Member>Find</i>	GTSMID	2.6.11
Split (Member)	 Split	<i>GTS Modeling>Member>Split Member</i>	GTSSplitMember	2.6.12
Split to Crossing Members	 Split to Members	<i>GTS Modeling>Member>Split to Crossing Members</i>	GTSSplitToMembers	2.6.13
Merge (Member)	 Merge	<i>GTS Modeling> Member>Merge Members</i>	GTSMergeMembers	2.6.14
Change (Member)	 Change Member	<i>GTS Modeling> Member>Change</i>	GTSCheckChange	2.6.15
Filter (Members)	 Filter	<i>GTS Modeling> Member>Filter</i>	GTSCheckFilterMembers	2.6.16
Generate Quad	 Quad	<i>GTS Modeling>Shell>Generate quad at joints</i>	GTSShell	2.6.17
Generate Triangle	 Triangle	<i>GTS Modeling>Shell>Generate triangle at joints</i>	GTSShellT	2.6.17
Reverse Incidence Order	 Reverse	<i>GTS Modeling>Shell> Reverse Incidence Order</i>	GTSShellReverse	2.6.18
Find (Shell)	 Find Shell	<i>GTS Modeling>Shell>Find</i>	GTSCheckFind	2.6.19
Change (Shell)	 Change Shell	<i>GTS Modeling>Shell>Generate triangle at joints</i>	GTSShellChange	2.6.20
Locate Duplicates	 Joints Duplicates	<i>GTS Modeling>Checks>Joints Duplicates</i>	GTSCheckDuplicateJoints	2.6.21
Locate Floating	 Joints Floatings	<i>GTS Modeling>Checks>Joints Floatings</i>	GTSCheckFloatingJoints	2.6.22
Joints Interference	 Joints Interference	<i>GTS Modeling>Checks>Joints Interference</i>	GTSCheckInteferenceJoints	2.6.23
Members Duplicates	 Members Duplicates	<i>GTS Modeling>Checks>Members Duplicates</i>	GTSCheckDuplicateMembers	2.6.24
Members Zero Length	 Members Zero Length	<i>GTS Modeling>Checks>Members Zero Length</i>	GTSCheckMembersZeroLength	2.6.25

Physical Members	 Physical Members	<i>GTS Modeling>Checks>Physical Members</i>	GTSCheckPhysicalMembers	2.6.26
Shells Duplicates	 Shells Duplicates	<i>GTS Modeling>Checks>Shells Duplicates</i>	GTSCheckDuplicateShells	2.6.27
Names Duplicates	 Names Duplicates	<i>GTS Modeling>Checks>Names Duplicates</i>	GTSCheckNames	2.6.28
Renumber Names	 Renumber Names	<i>GTS Modeling>Checks>Renumber Names</i>	GTSRenumber	2.6.29
Database Integrity	 Database Integrity	<i>GTS Modeling>Checks>Database Integrity</i>	GTSCheckDatabase	2.6.30
1D Curve (Meshing)	 1D Curve	<i>GTS Modeling>Mesh Generation>1D Along Line or Curve or Circle</i>	GTSMesh1D	2.6.31
2D 2Curves (Meshing)	 2D 2Curves	<i>GTS Modeling>Mesh Generation>2D Between 2 Lines or Curves</i>	GTSMesh2D2L	2.6.32
2D 4Curves (Meshing)	 2D 4Curves	<i>GTS Modeling>Mesh Generation>2D Between 4 Lines or Curves</i>	GTSMesh2D4L	2.6.33
2D Area (Meshing)	 2D Area	<i>GTS Modeling>Mesh Generation>2D Between 4 Lines or Curves</i>	GTSMesh2DPoly	2.6.34
3D Extrude (Meshing)	 3D Extrude	<i>GTS Modeling>Mesh Generation>3D Extrude PolyLine</i>	GTSExtrudePoly	2.6.35
2D 3Curves (Meshing)	 3D 3Curves	<i>GTS Modeling>Mesh Generation>3D Between 3 Lines or Curves</i>	GTSMesh3D3L	2.6.36
Array 3D Advanced	 Array 3D Advanced	-	GTSArrary3D	2.6.37
Soil Springs	 Soil Springs	<i>GTS Modeling>Soil Springs</i>	GTSFoundationSprings	2.6.38
Export STR	 C2	<i>GTS Modeling> CAESAR II>Export STR</i>	GTSExportSTR	2.6.39
Convert Lines/Polylines to Members/Shells	 Convert Lines to Members	<i>GTS Modeling> Convert Lines/Polylines to Members/Shells</i>	GTSDXFRead	2.6.40

Model Wizard		<i>GTS Modeling>Model Wizard</i>	GTSModelWizard	2.6.41
List (Group)	 List	<i>GTS Modeling>Groups>Manage</i>	GTSGroups	2.6.42
+Joints (Group)		<i>GTS Modeling>Groups>Add Joints</i>	GTSGroupJoints	2.6.42
+Members (Group)		<i>GTS Modeling>Groups>Add Members</i>	GTSGroupMembers	2.6.42
+Shells (Group)		<i>GTS Modeling>Groups>Add Shells</i>	GTSGroupShells	2.6.42
Self Weight		<i>GTS Modeling>Loads>Self Weight</i>	GTSSelfWeight	2.6.43
Load Cases		<i>GTS Modeling>Loads>Load Cases</i>	GTSNewLoadCase	2.6.44
Load Combinations		<i>GTS Modeling>Loads>Load Combinations</i>	GTSLoadCombination	2.6.54
Standardized Combinations		<i>GTS Modeling>Loads>Standardized Combinations</i>	GTSLoadCombinationStandardized	2.6.55
Joint Load		<i>GTS Modeling>Loads>Joint Load</i>	GTSJointLoad	2.6.45
Member Load		<i>GTS Modeling>Loads>Member Load</i>	GTSBeamLoad	2.6.46
Shell Load		<i>GTS Modeling>Loads>Shell Load</i>	GTSShellLoad	2.6.47
Area Load		<i>GTS Modeling>Loads>Area Load</i>	GTSAreaLoad	2.6.48
Wind Load ASCE 705		<i>GTS Modeling>Loads>Wind Load ASCE 705</i>	GTSWindLoadsASCE705	2.6.49
Wind Load ASCE 710		<i>GTS Modeling>Loads>Wind Load ASCE 710</i>	GTSWindLoadsASCE710	2.6.50
Seismic Load		<i>GTS Modeling>Loads>Seismic Load</i>	GTSSeismicLoading	2.6.53
Steel Design Parameters		<i>GTS Modeling>Steel Design Parameters</i>	GTSSteelDesignParameters	2.6.56
Create Input File		<i>GTS Modeling>Create Input File</i>	GTSExportGTI	2.6.57
Edit GTI		<i>GTS Modeling>Edit GTI</i>	GTSEditGTI	2.6.58

Execute GT STRU DL		<i>GTS Modeling>Execute</i>	GTSExecuteGTI	2.6.59
Read Results	 Read Results	<i>GTS Modeling>Read Results</i>	GTSTResultsGTI	2.6.60
Import GTI	 Import GTI		GTSGTIRead	2.6.61
Set View	 Set View	<i>GTS Display>Set View</i>	GTSSetView	2.6.62
3D View	 3D	<i>GTS Display>3D Sections</i>	GTSSet3D	2.6.63
Analytical/Physical	 Analytical/Physical	<i>GTS Display>Analytical/Physical</i>	GTSDisplayPhysicalMembers	2.6.64
Frame View	 Frame	<i>GTS Display>Frame</i>	GTSSet1D	2.6.63
Options (View)	 Options	<i>GTS Display>Options</i>	GTSDisplay	2.6.66
Colors	 Colors	<i>GTS Display>Colors</i>	GTSColorView	2.6.65
Annotate	 Annotate	<i>GTS Display>Annotate</i>	GTSAAnnotate	2.6.67
Select	 Select	-	GTSSelect	2.6.68
Display Member Local Axes	 Member Local Axes	<i>GTS Display>Member Local Axes</i>	GTSDisplayLocalAxes	2.6.69
Display Member Releases	 Releases	<i>GTS Display> Member Releases</i>	GTSDisplayReleases	2.6.70
Display Shell Planar Axes	 Shell Planar Axes	<i>GTS Display> Shell Planar Axes</i>	GTSDisplayPlanarAxes	2.6.71
Display Joint Supports	 Joint Supports	<i>GTS Display> Joint Supports</i>	GTSDisplaySupports	2.6.72
Display Joint Loads	 Joint Loads	<i>GTS Display>Joint Loads</i>	GTSDisplayJointLoads	2.6.73
Display Member Loads	 Member Loads	<i>GTS Display>Member Loads</i>	GTSDisplayMemberLoads	2.6.74
Display Shell Loads	 Area Loads	<i>GTS Display>Shell Loads</i>	GTSDisplayElementLoads	2.6.75
Display Area Loads	 Shell Loads	<i>GTS Display>Area Loads</i>	GTSDisplayAreaLoads	2.6.76
Deformed Structure	 Deformed	<i>GTS Display>Deformed Structure</i>	GTSDisplayJointDisplacements	2.6.77
Undeformed Structure	 Undeformed	<i>GTS Display> Udeformed Structure</i>	GTSDisplayJointDisplacements	2.6.77
Annotate Joint Displacements	 Annotate Displacements		GTSAAnnotateJointDisplacements	2.6.78

Displacements	 Displacements	<i>GTS Display> Displacements</i>	GTSDisplaySectionDisplacements	<u>2.6.79</u>
Member Diagrams	 Diagrams	<i>GTS Display>Member Diagrams</i>	GTSDisplayMemberForces	<u>2.6.80</u>
Finite Element Results	 Elements	<i>GTS Display>Element Results</i>	GTSDisplayElementResults	<u>2.6.81</u>
Finite Element Results Selection	 Selection	<i>GTS Display>Element Results Selection</i>	GTSDisplayElementResultsSel	<u>2.6.82</u>
Member Code Check Results	 Code Check	<i>GTS Display>Member Code Check Results</i>	GTSColorCodeCheck	<u>2.6.83</u>
Displacements Datasheets	 Displacements	<i>GTS Display>Results Datasheets> Displacements</i>	GTSDataSheetJointDisp	<u>2.6.84</u>
Member Forces Datasheets	 Member Forces	<i>GTS Display>Results Datasheets> Displacements</i>	GTSDataSheetMemberForces	<u>2.6.84</u>
Section Forces Datasheets	 Section Forces	<i>GTS Display>Results Datasheets> Displacements</i>	GTSDataSheetSectionForces	<u>2.6.84</u>
Reactions Datasheets	 Reactions	<i>GTS Display>Results Datasheets> Displacements</i>	GTSDataSheetReactions	<u>2.6.84</u>
Stresses Datasheets	 Stresses	<i>GTS Display>Results Datasheets> Displacements</i>	GTSDataSheetStresses	<u>2.6.84</u>
Code Check Datasheets	 Code Check	<i>GTS Display>Results Datasheets> Displacements</i>	GTSDataSheetCodeCheck	<u>2.6.84</u>
Report Builder		-	GTSReportBuilder	<u>2.6.85</u>
Clear Results	 Clear	<i>GTS Display>Clear Results Layer</i>	GTSDisplayResultsClear	<u>2.6.86</u>
Current Version	-	<i>GTS Display>Version</i>	GTSVersion	<u>2.6.87</u>