GT STRUDL® Version 42
User Guide

CAD Modeler
Getting Started Guide

Release Date: November 2023

HEXAGON

Notice

This CAD Modeler Getting Started Guide is applicable to GT STRUDL Version 42 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

Copyright © 2023 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- Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States.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Unpublished - rights reserved under the copyright laws of the United States.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, ProtoViewCtl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Incorporates portions of 2D DCM, 3D DCM, and HLM by Siemens Product Lifecycle Management Software III (GB) Ltd. All rights reserved. Gigasoft is a registered trademark, and ProEssentials a trademark of Gigasoft, Inc. VideoSoft and VXFlexGrid are either registered trademarks or trademarks of ComponentOne LLC 1991-2017, All rights reserved. Oracle, JD Edwards, PeopleSoft, and Retek are registered trademarks of Oracle Corporation and/or its affiliates. Tribon is a trademark of AVEVA Group plc. Alma and act/cut are trademarks of the Alma company. Other brands and product names are trademarks of their respective owners.

Table of Contents

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.

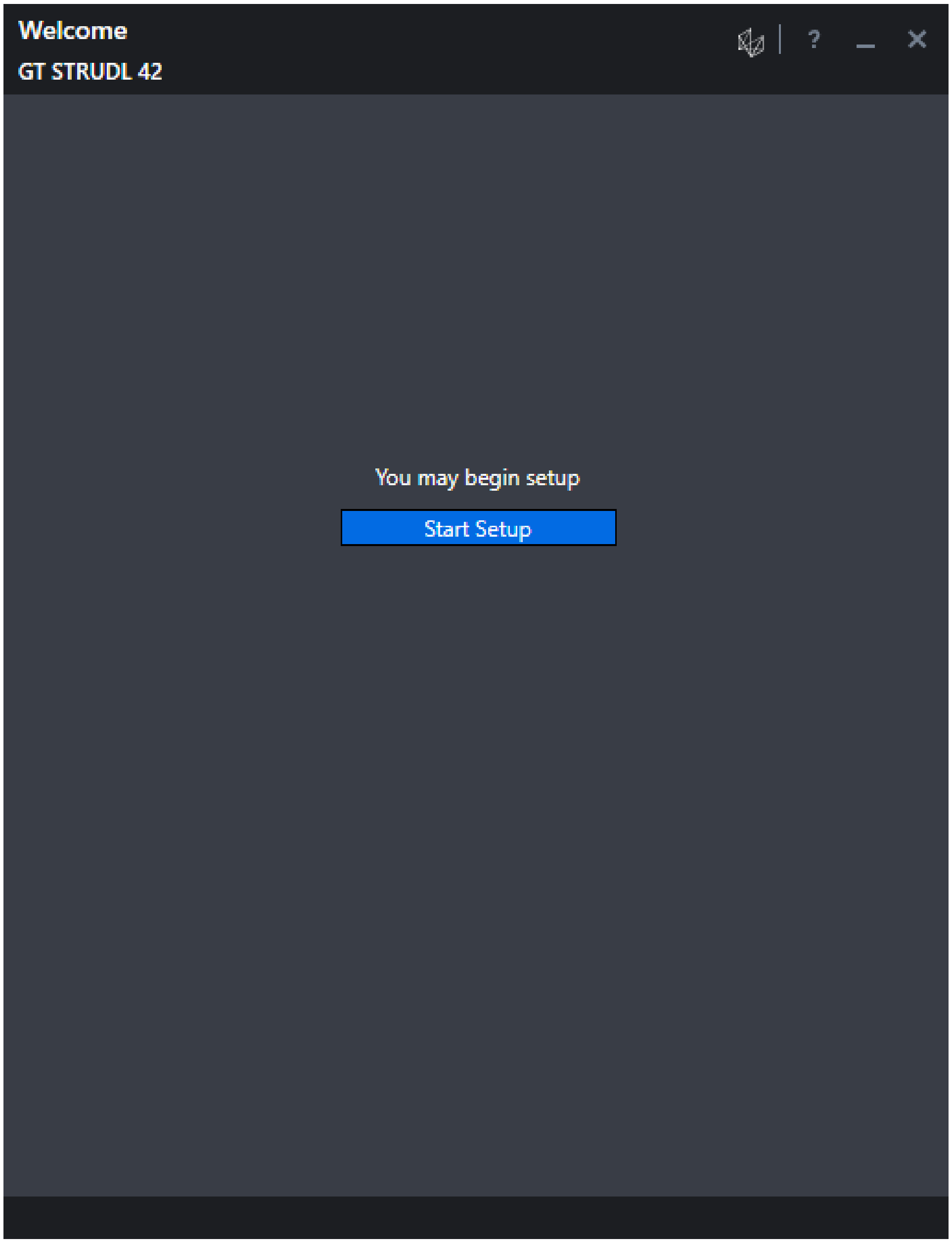
1.1. Introduction

This document contains information about:

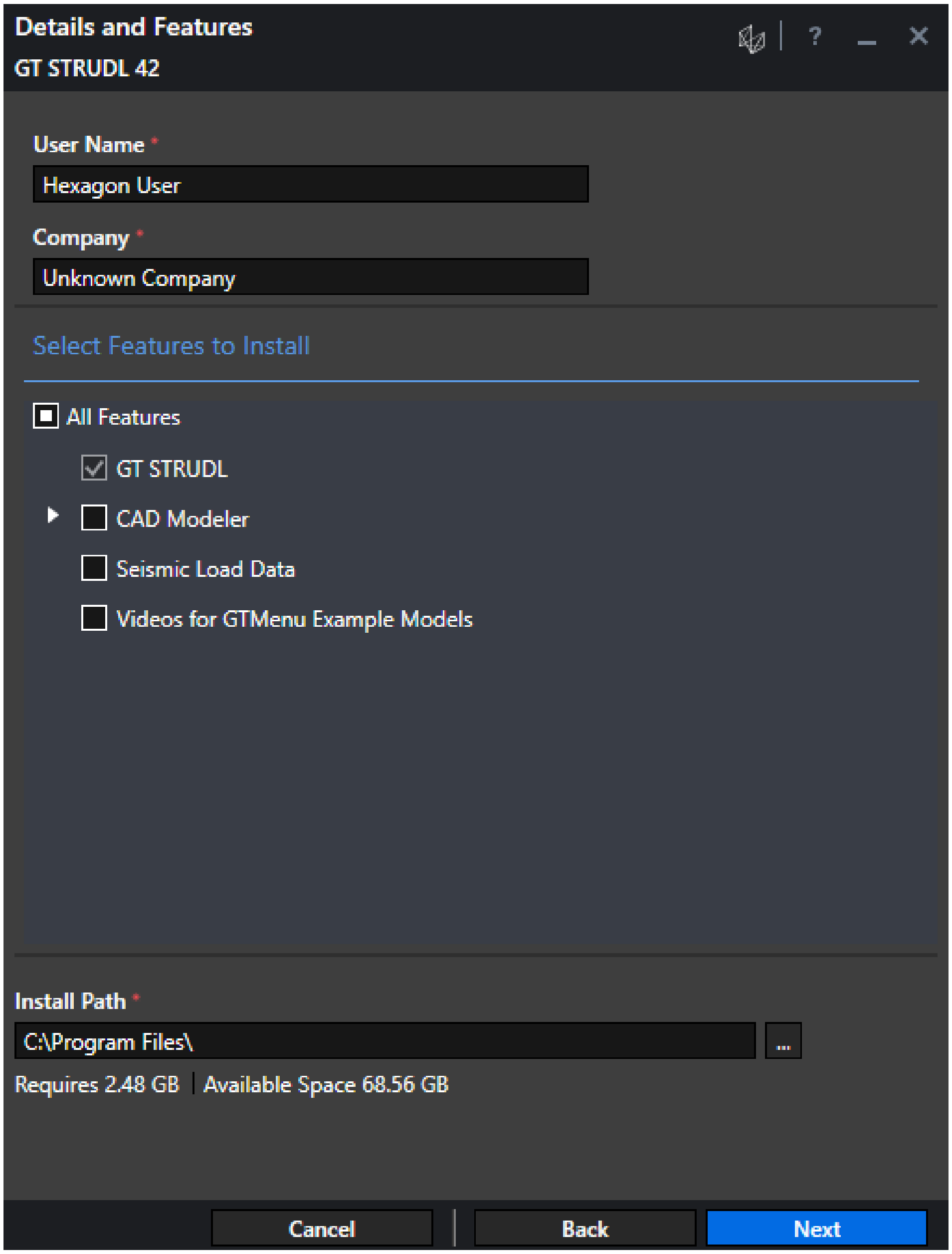
- Installing CAD Modeler
- Configuring AutoCAD/BricsCAD and running CAD Modeler for first time
- List of CAD Modeler Functions with detailed explanation for each one
- Tutorial examples
- Summary of CAD Modeler commands

1.2. Installing CAD Modeler under Windows 11

CAD Modeler is installed during GT STRUDL v42 installation. Start by pressing “Start Setup” button.

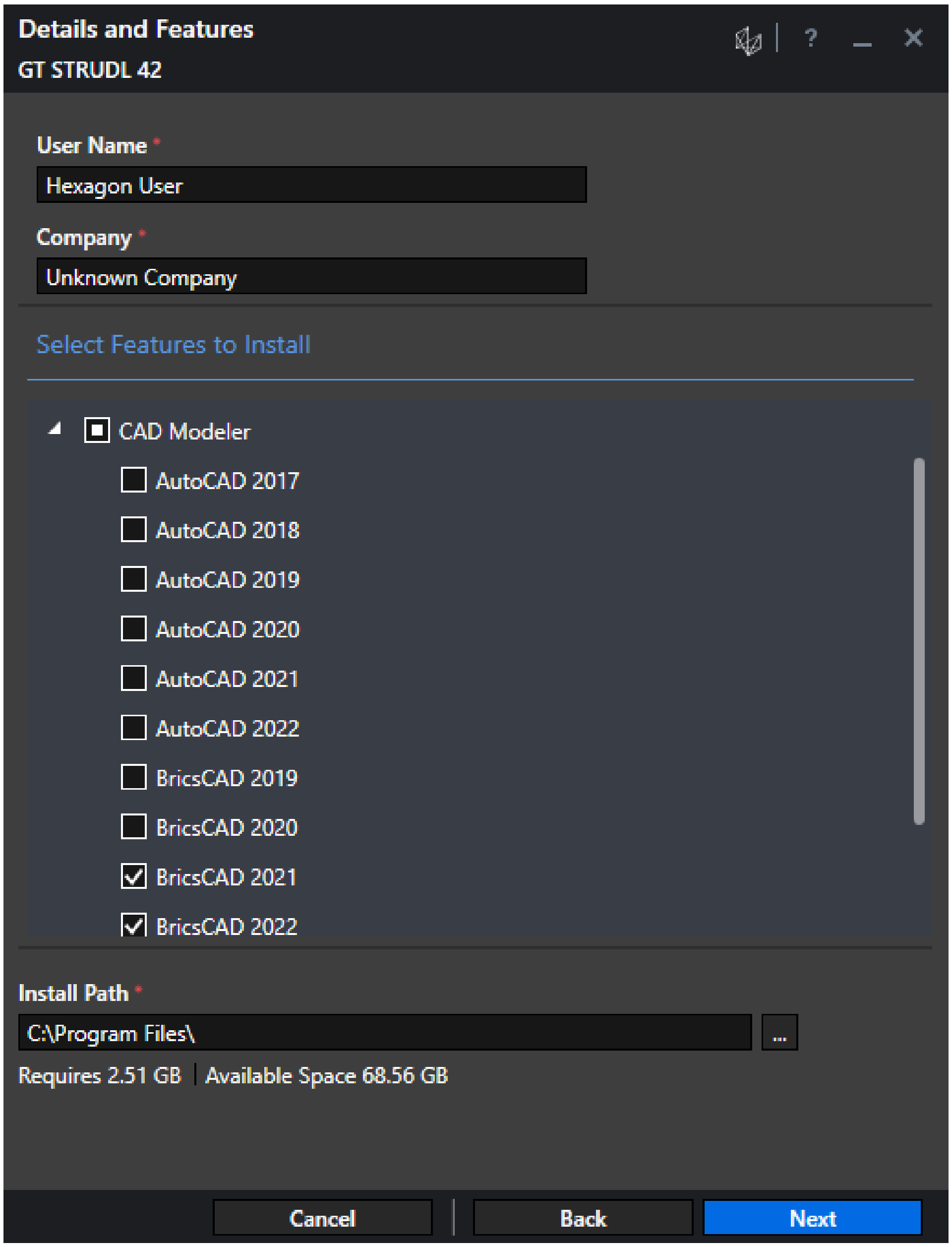


In order to install CAD Modeler, expand the "CAD Modeler" on the form shown below during the GT STRUDL main installation procedure. Then, available versions of CAD Modeler will be displayed.

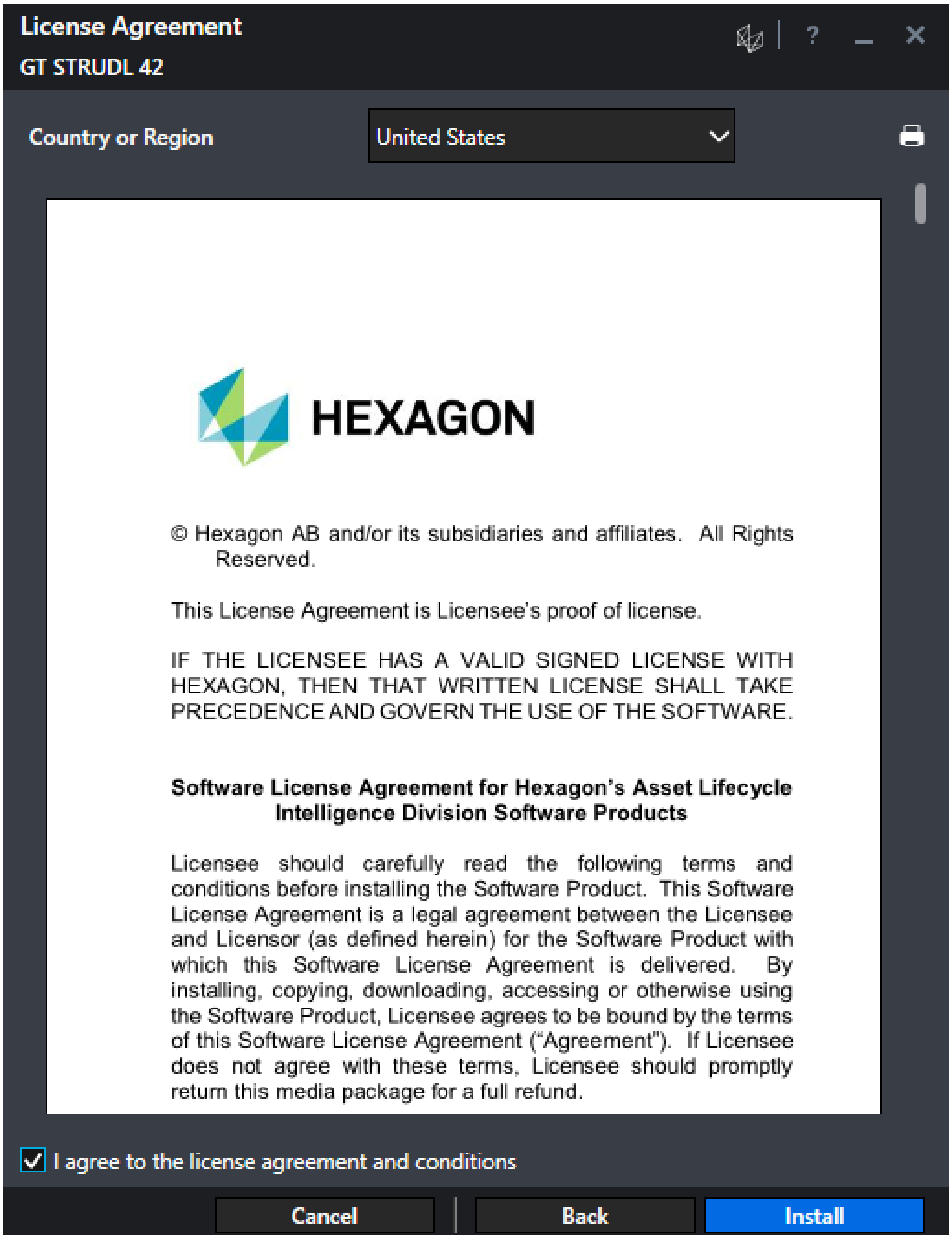


Select AutoCAD or BricsCAD versions to install the CAD Modeler and press “Next” button.

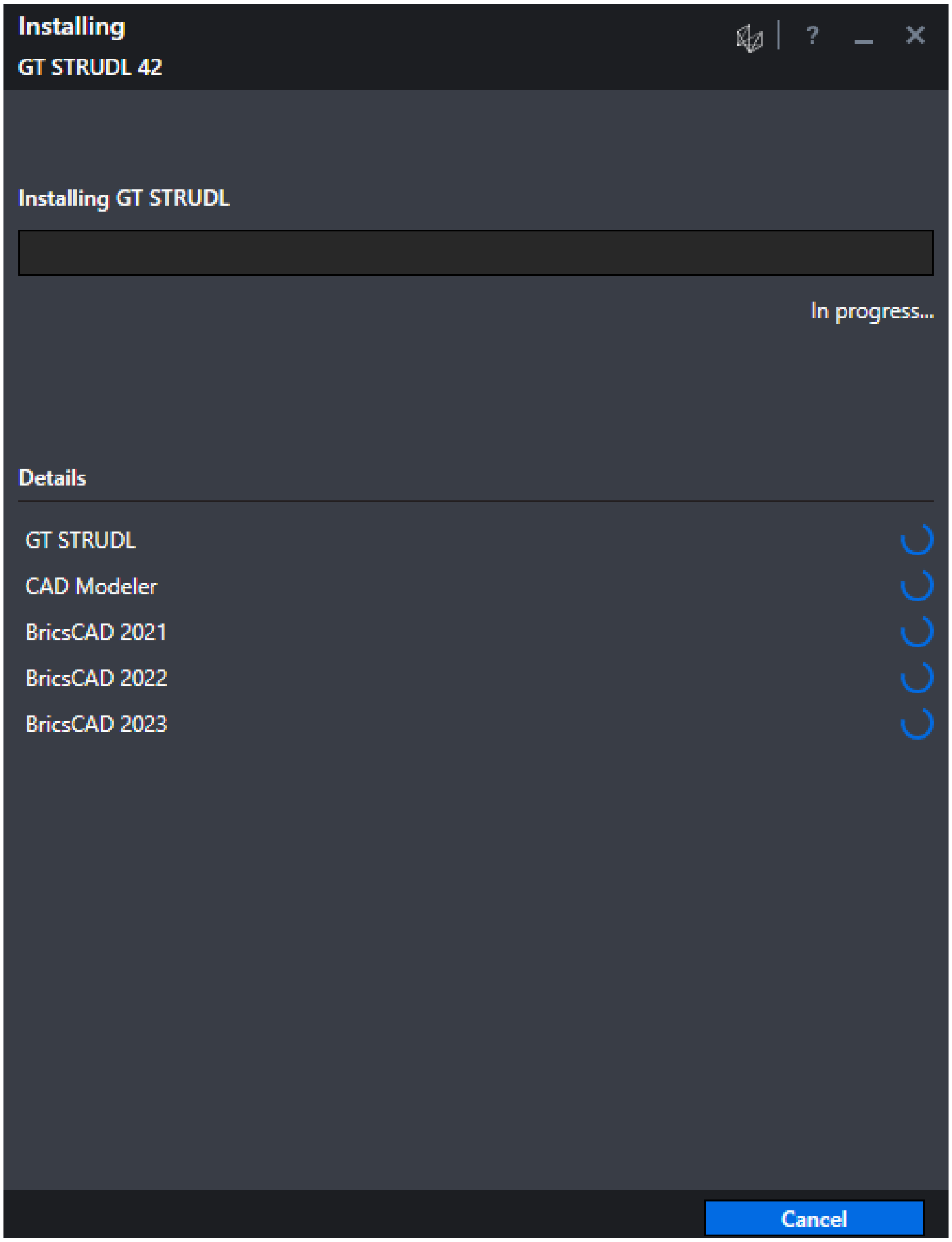
An AutoCAD version (2017-2022) or a BricsCAD version (Platinum or Pro 19.x, 20.x, 21.x, 22.x, or 23.x) has to be installed in the computer prior to CAD Modeler installation.



Check the license and agreement box and press the button "Install".

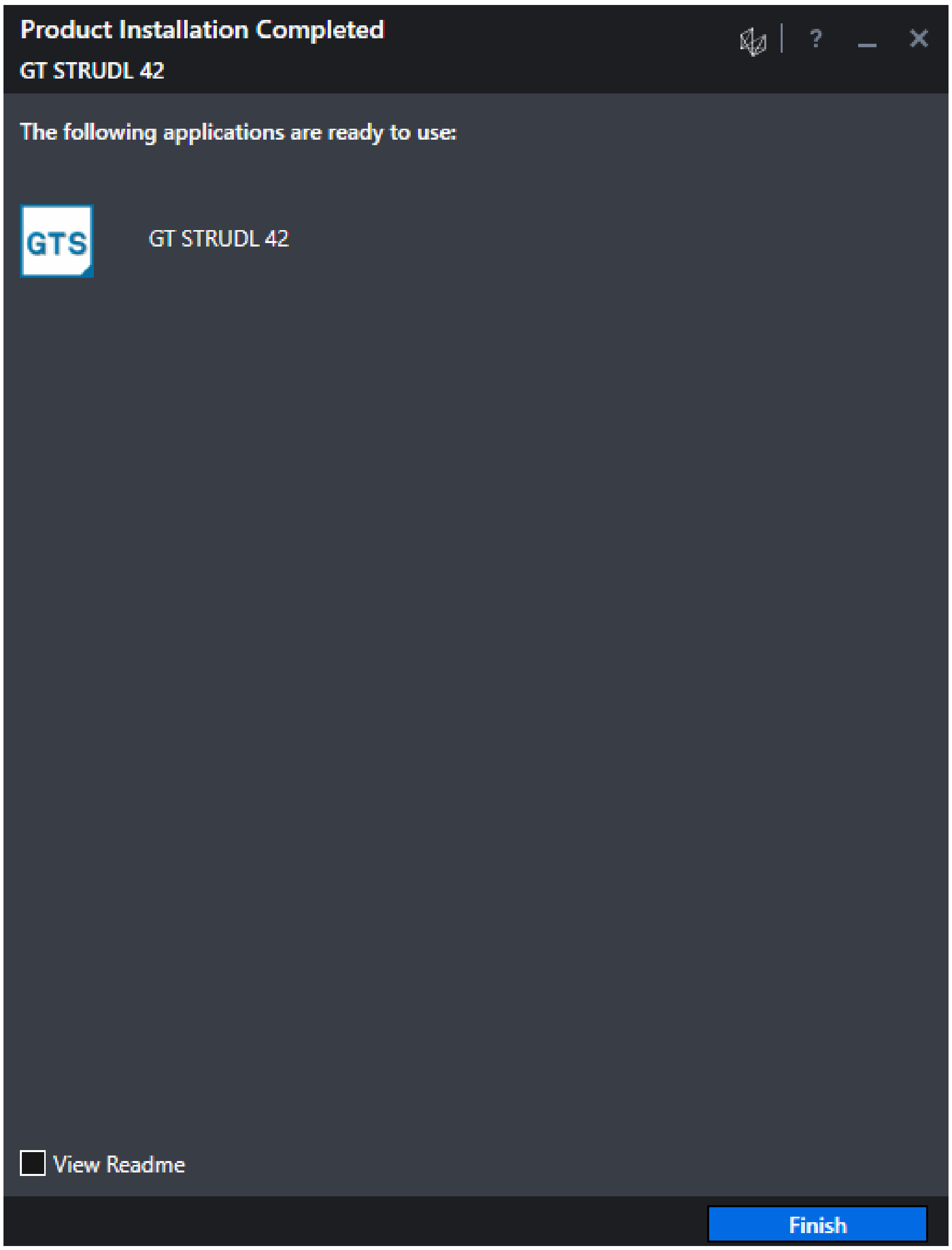


The next screen summarizes your selection.



By pressing “Finish”, the installation is complete.

CAD Modeler is installed in the same installation directory with GT STRUDL, under the sub-directory “CADModeler”. For example,“C:\Program Files\GTSTRUDL\42\CADModeler” is a typical CADModeler installation directory.



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.

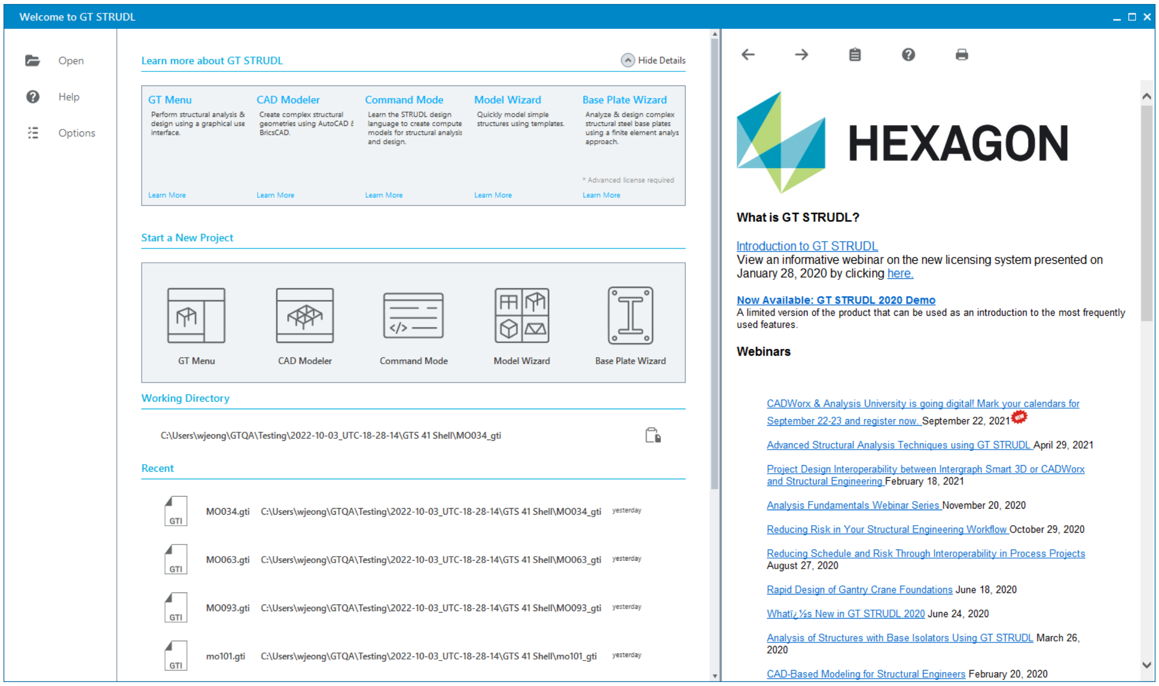
All GT STRUDL structural data, which are created using CAD Modeler, are stored in two files: the .dwg file which contains the AutoCAD/BricsCAD information, and the .db file which contains the structural data. The two files have the same filename (only the extension is different) and they are linked together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

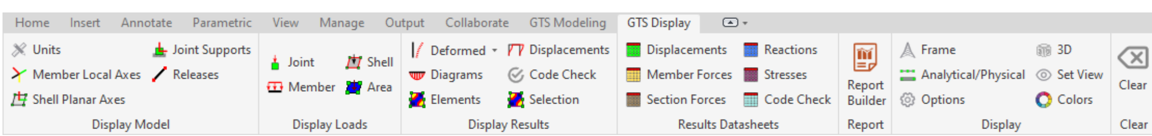
2.2. Running CAD Modeler

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.

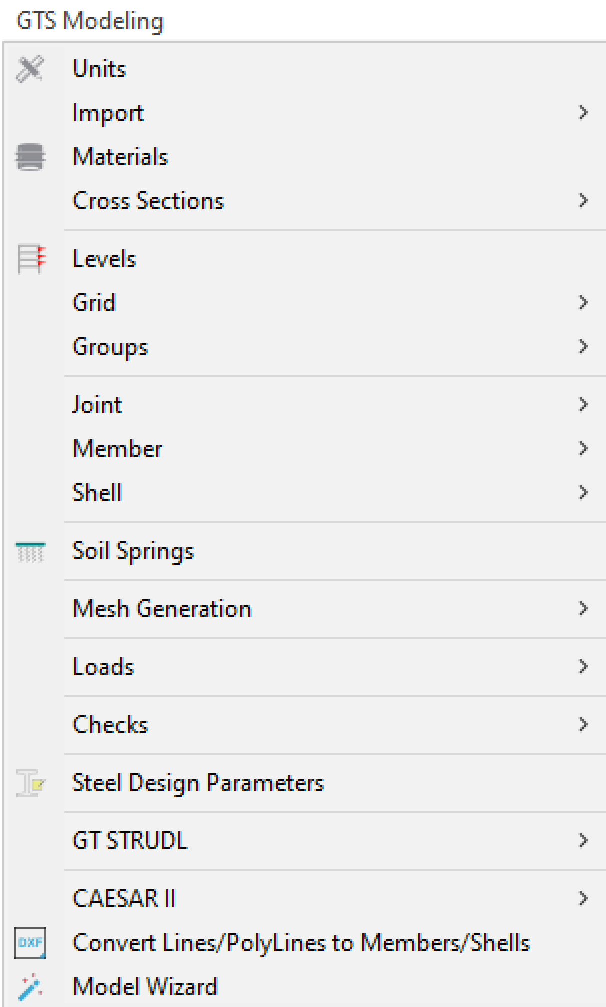


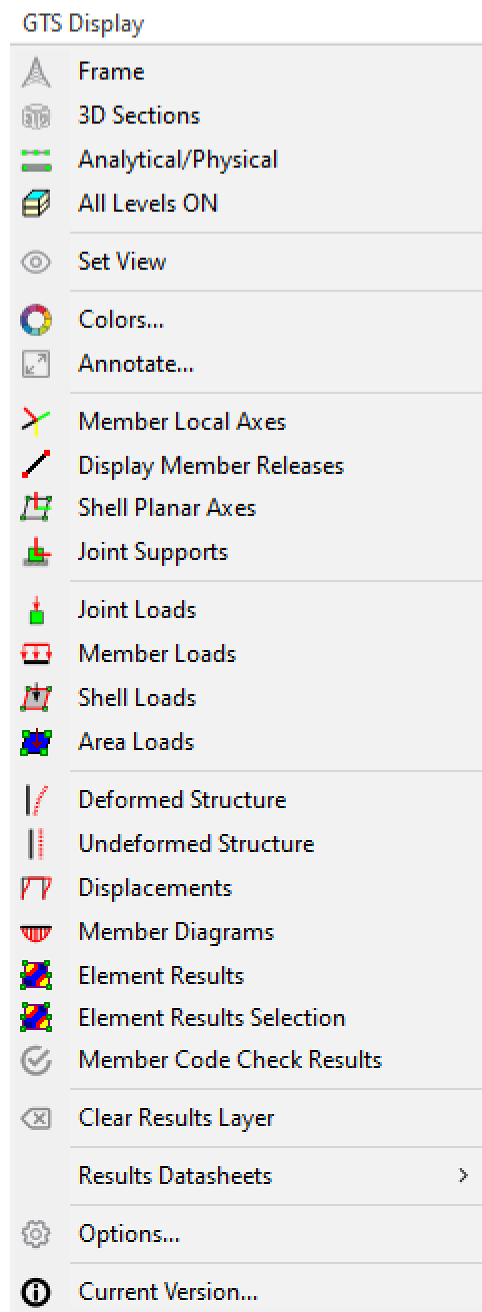
2.3. Menu Bar and Ribbon Area

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

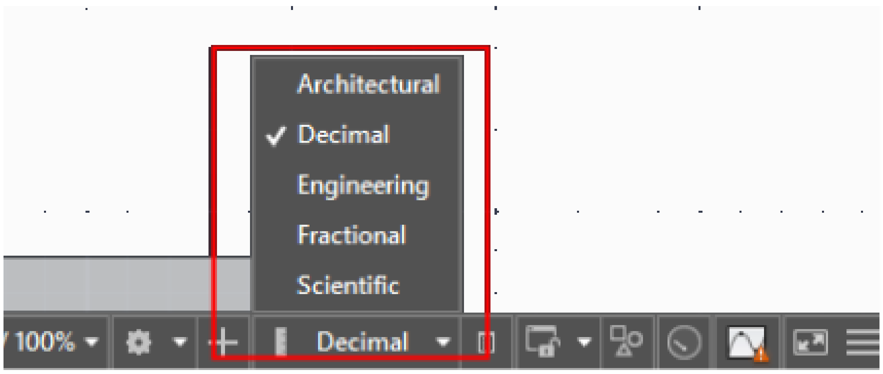
You can use AutoCAD or BricsCAD commands, such as Move, Copy, Rotate, Mirror and Delete to generate your model faster. The assumptions made in the use of each command are:

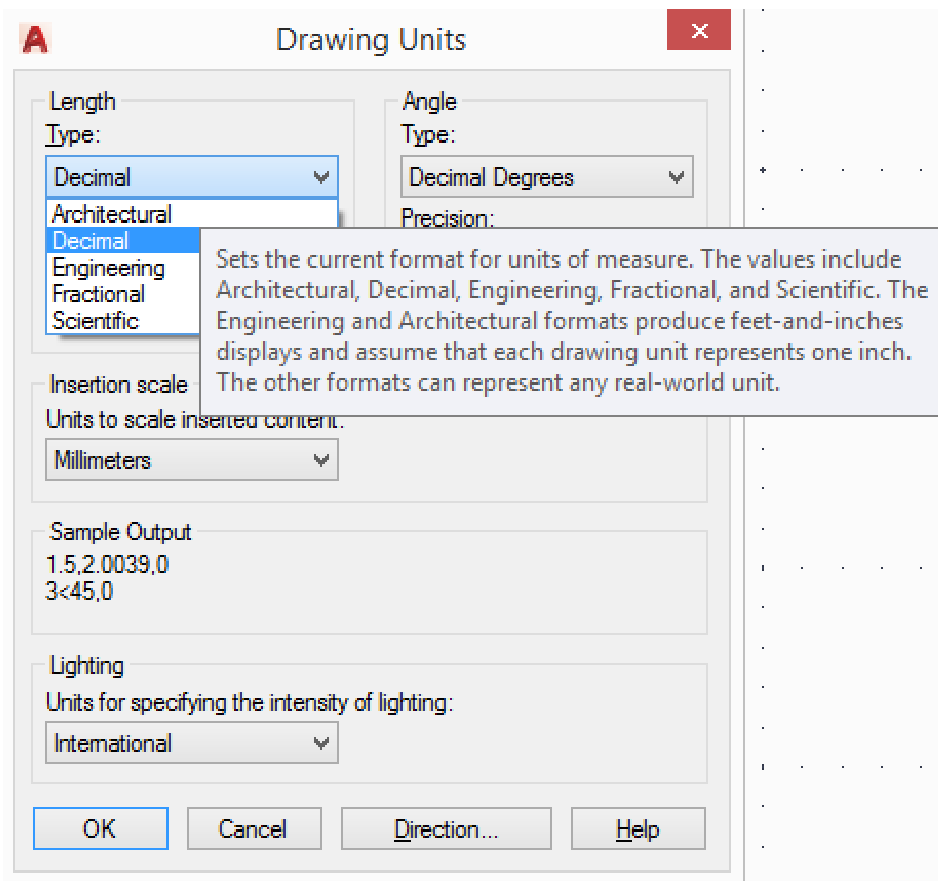
- Move: By moving a joint, the members and finite elements connected to the joint “follow” this movement
- Copy: Joint, Member and Element Loads and Supports are not copied
- Mirror: Joint, Member and Element Loads and Supports are not copied or mirrored. The Beta Angle of members is not mirrored. Element 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.





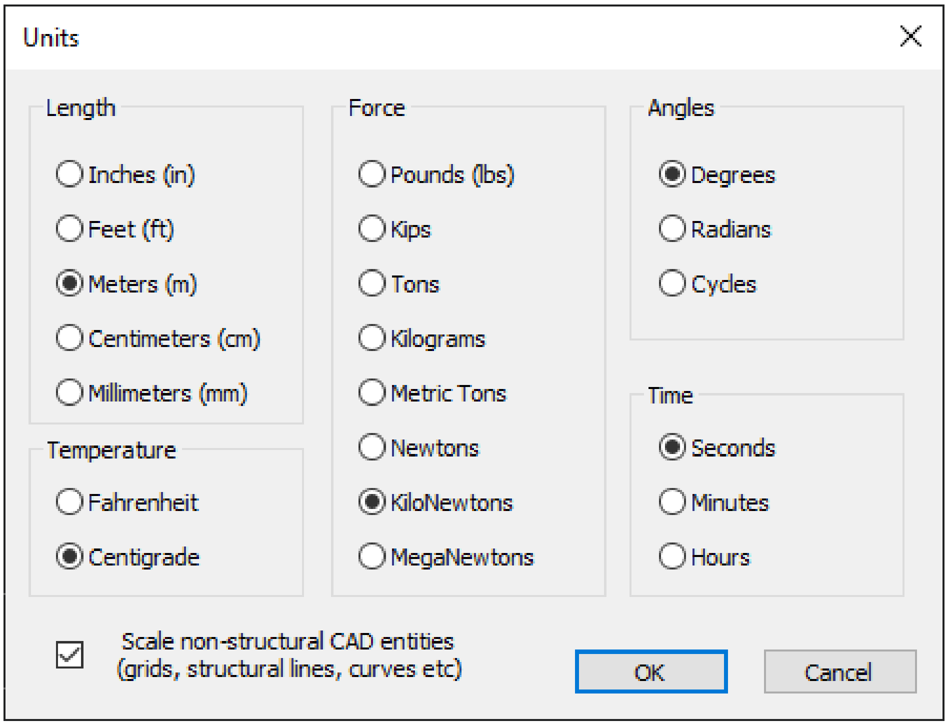
As written in AutoCADs' documentation "*The **Engineering** and **Architectural** formats produce feet-and-inches displays and assume that each drawing unit **represents one inch**. The other formats can represent any real-world unit*". Therefore, if you select **Engineering** or **Architectural** formats 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 

or from the menu "*GTS Modeling>Units*" or by typing `GTSUnits` at the command prompt.

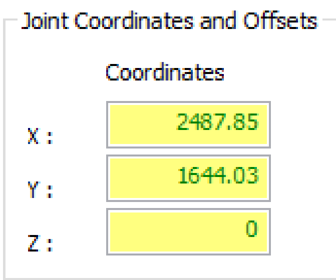


You can change the Units any time during working in CAD Modeler. Moreover, you can choose if the non-structural AutoCAD/BricsCAD entities, such as grids, structural lines, curves, polylines, etc will be scaled together with the structure whenever you change the length units.

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:

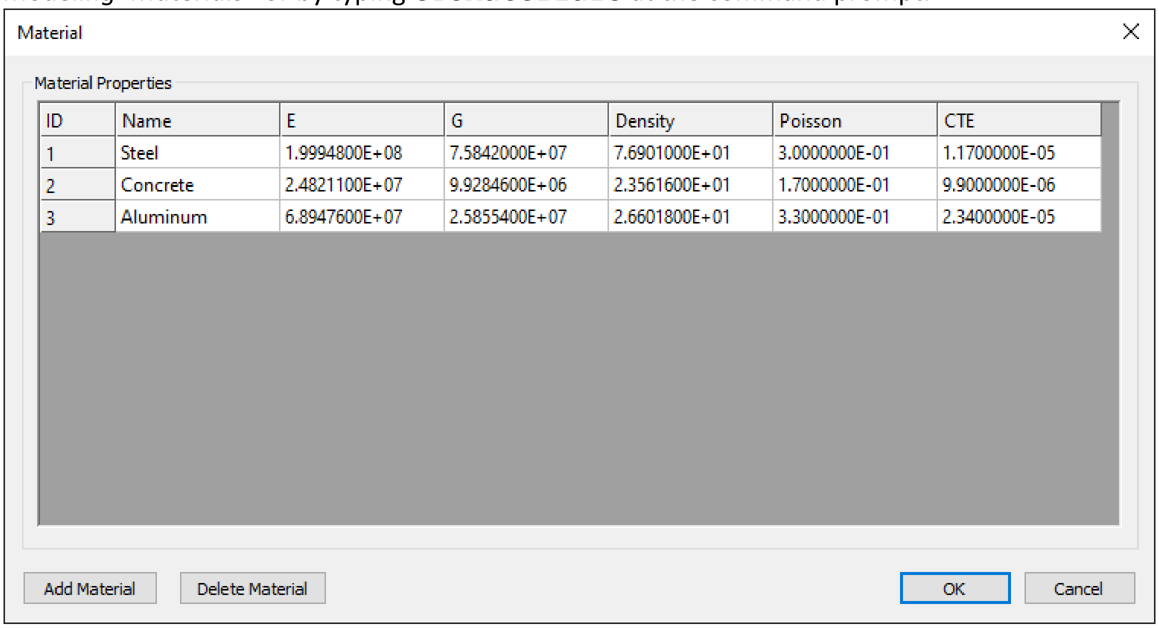
Note: Some Edit Boxes appear in yellow background and green fonts, like the one at the picture below. You can use mixed units in the yellow edit boxes. For more information about Mixed Units and the valid syntax, please read GT STRUDL GT Menu Guide.



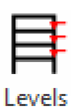
2.6.2. Materials

The following form is used to modify existing material properties or create new materials. You can select this command either from the ribbon command 

or from the menu "*GTS Modeling>Materials*" or by typing `GTSMaterials` at the command prompt.



2.6.3. Levels

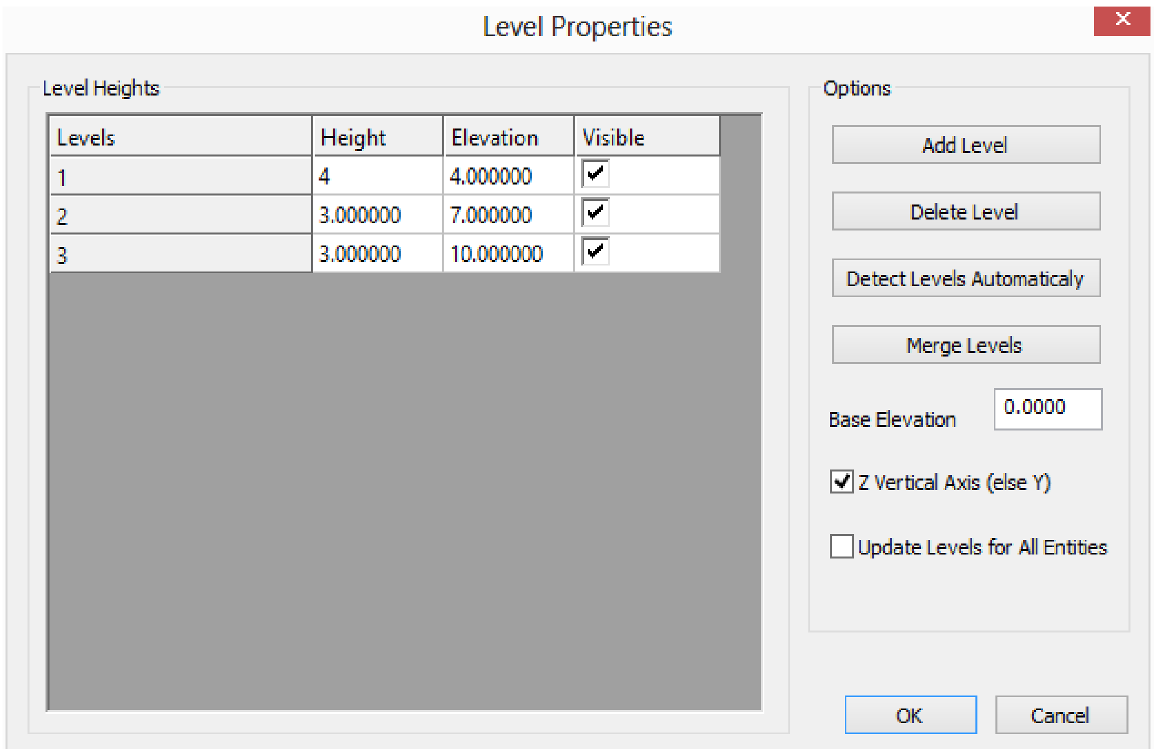
It is optional to define Levels (stories) in your structure. However, it is recommended that you do so when modeling industrial or other building-like structures as this will save you time during the creation of your model. You can access the level properties dialog from the ribbon icon 

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.



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.

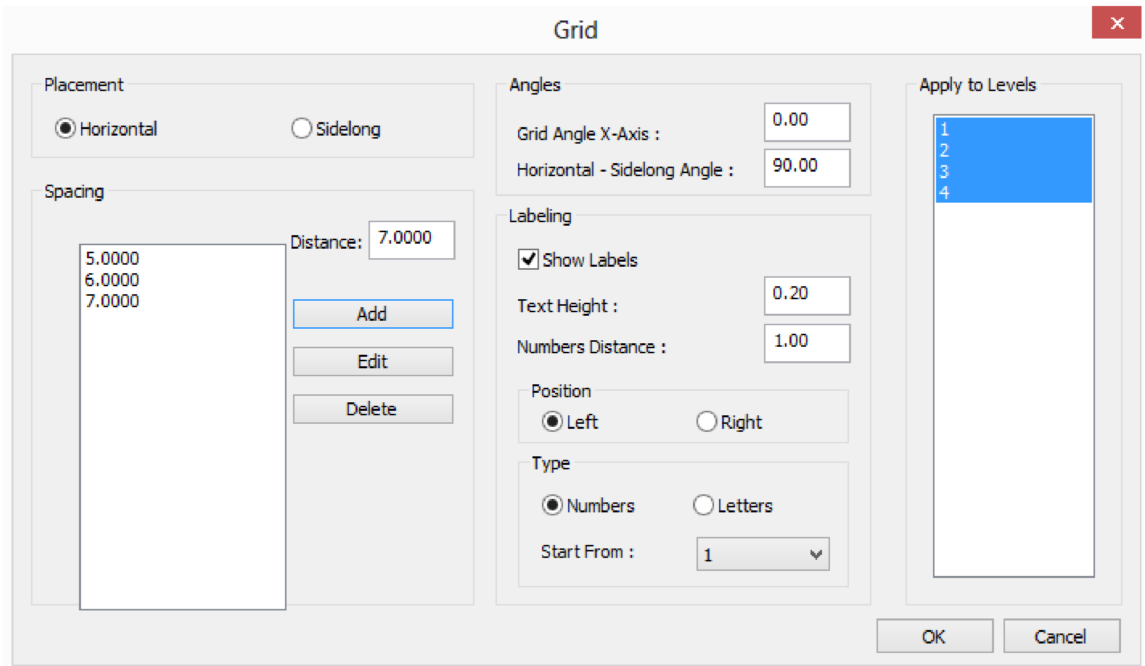
Moreover, you can define a grid system and/or generate vertical members (Columns) with a single click. These commands will be explained below.

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

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 

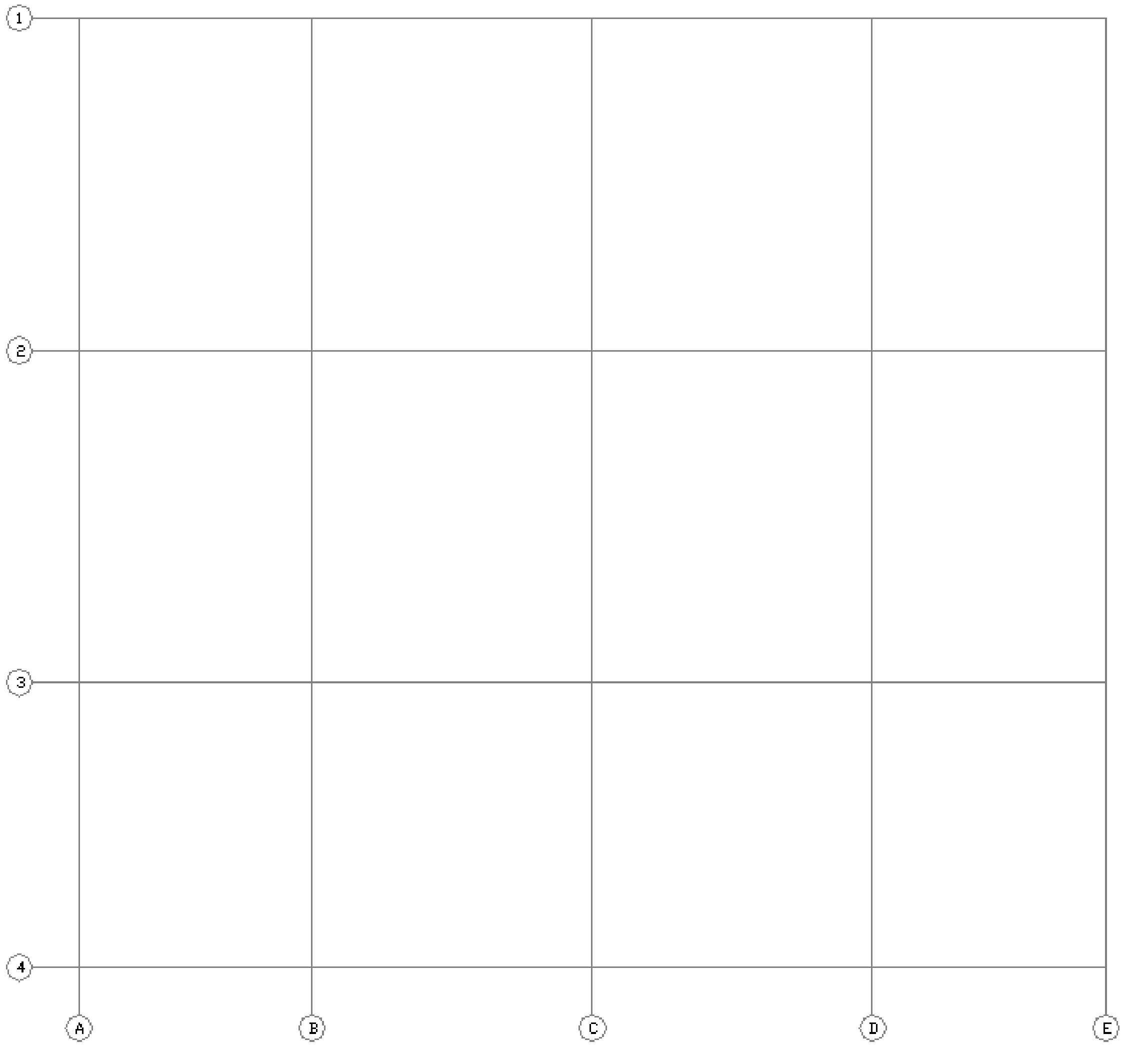
, or from the menu “*GTS Modeling>Grid>Create*” or by typing `GTSGrid` at the command prompt.



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 S*tarting 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).

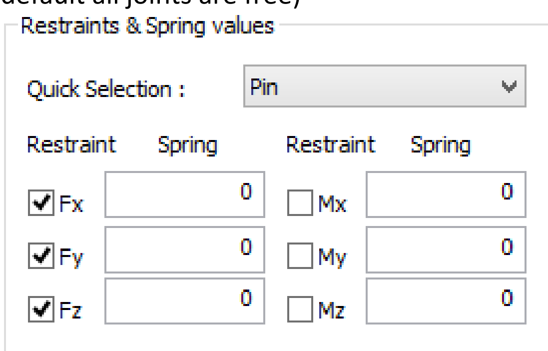
If you have already defined Levels at the structure, you can generate individual joints at the current lever from the ribbon command At Level or from the menu "*GTS Modeling>Joint>Generate Joint at Level*" or by typing `GTSJointLevel` at the command prompt. You then have to enter only X and Y coordinate (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).

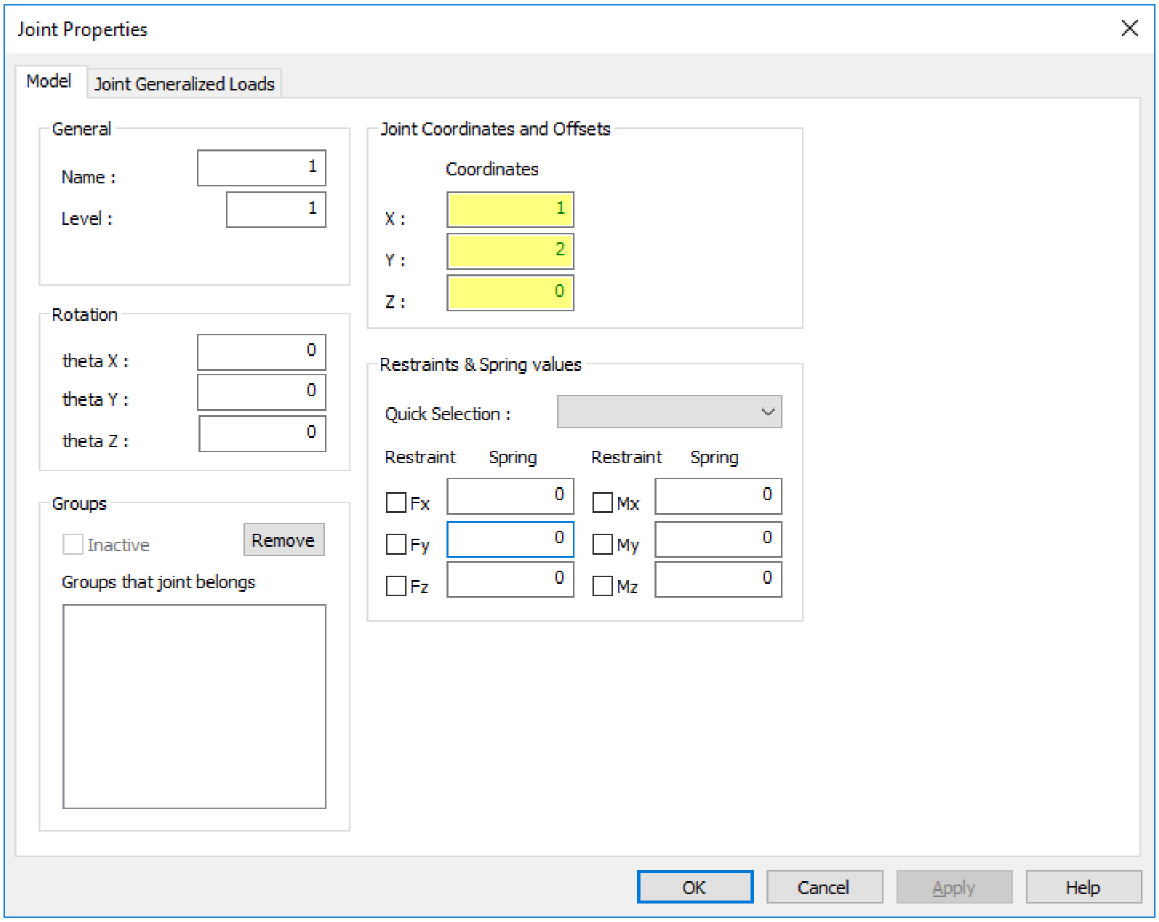
2.6.7. Joint Supports

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)



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.



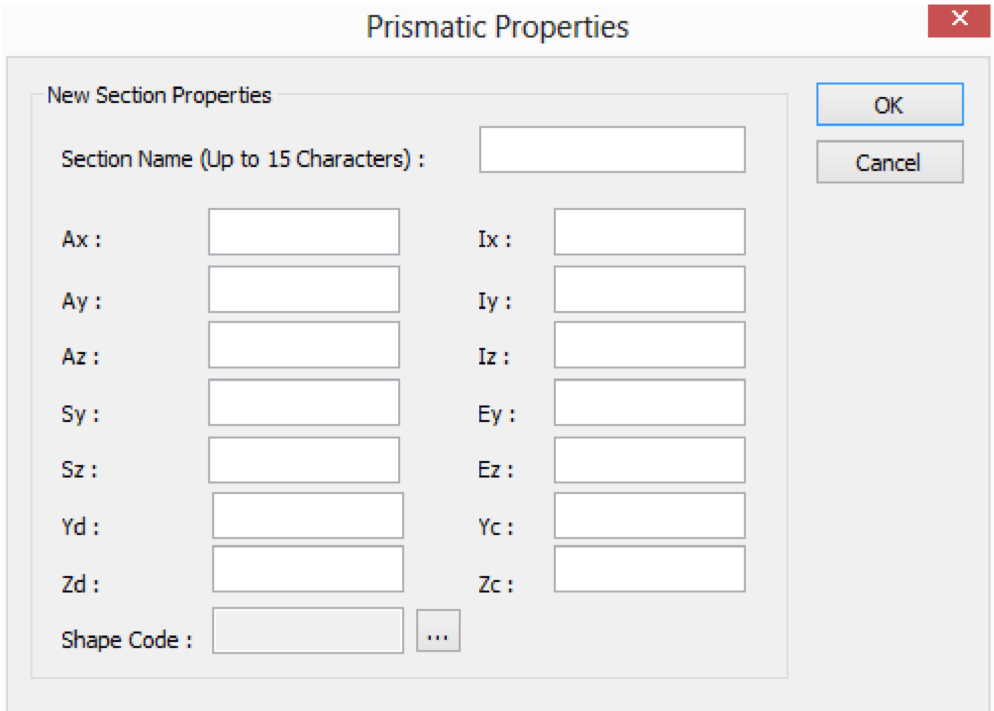
The "*Joint Properties*" form appears, and at the "*Model*" tab you can enter the *Name* of the Joint (up to 8 characters) the Level that the joint belongs (optional), the theta *rotation* angles for rotated support joints, the *Groups* that the joints belongs to, the *coordinates* of the joint in the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

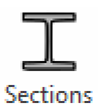
If you select more than one joint, then "*Multiple Selection*" appears at the top of the Joint Properties form, and all data entered in the form will be applied to all selected joints.

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 `GTSPrismatic` at the command prompt.
In the dialog shown below, you enter the cross-section properties in the current unit system.

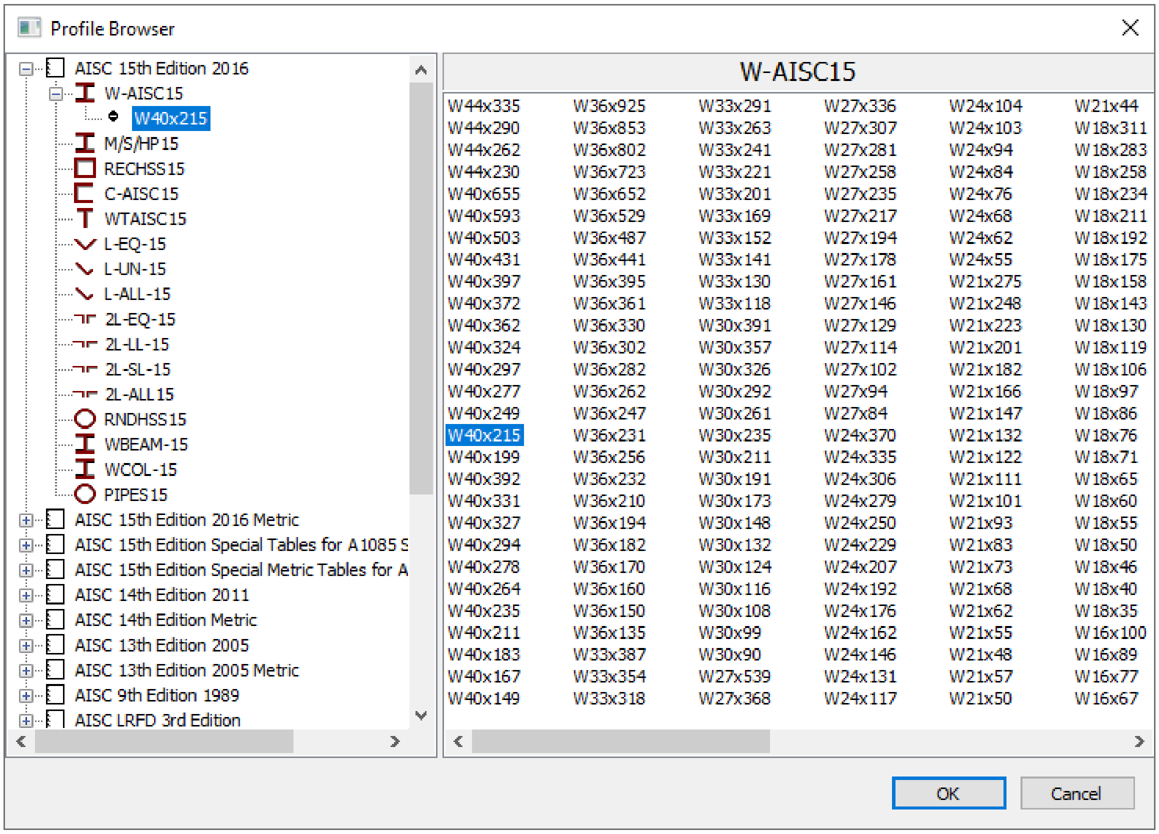


You can access the existing cross section library (GT STRUDL tables) from the ribbon command 

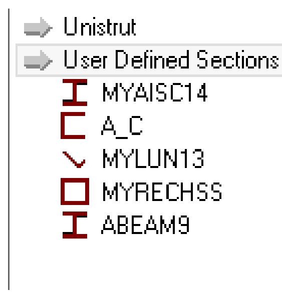
, or from the Menu "*GTS Modeling>Cross Sections>Table*" or by typing `GTSParams` at the command prompt.

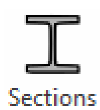
Using the following form, where all GT STRUDL built-in TABLES are available, you select the cross sections for your project by double clicking on them, at the right part of the screen. The list of selected cross-section profiles appears at the left part of the screen and the selected profiles have a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

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 `GTSOpenDS` 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”.



Next time you select the command 

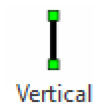
you will be able to see the user defined sections at the bottom of the list.

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

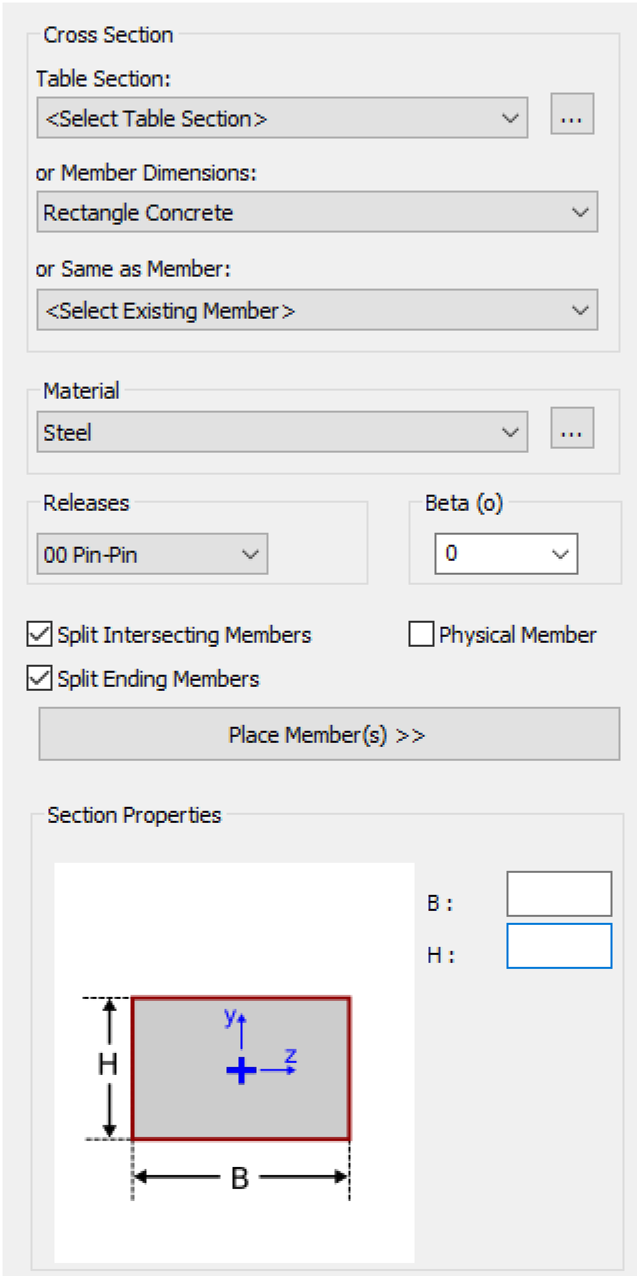
You can generate individual members from the ribbon command 

or from the menu "*GTS Modeling>Members>Generate Beam Members*" or by typing `GTSBeam` 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 lever from the ribbon command 

or from the menu "*GTS Modeling>Member>Generate Vertical Member*" or by typing `GTSColumn` 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).

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 

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 

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 spit 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 

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 

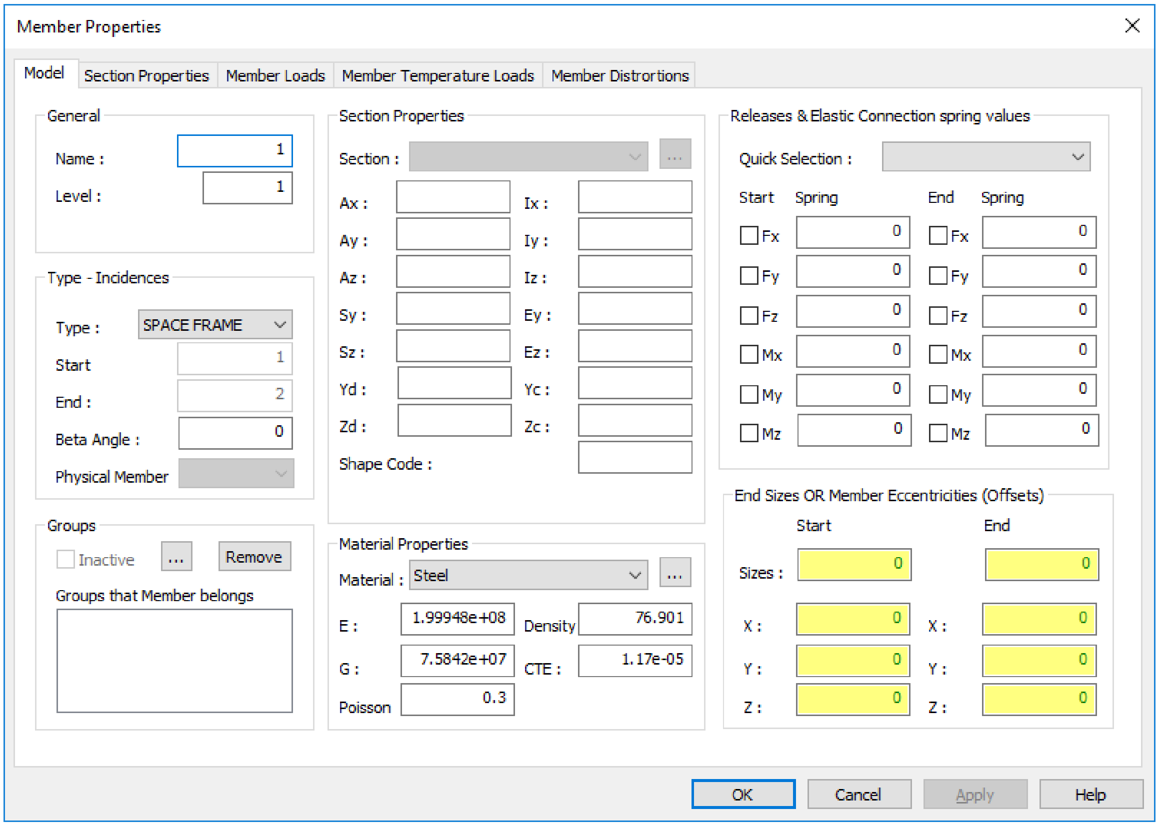
or from the menu "*GTS Modeling>Member>Merge Members*" or by typing `GTSMergeMembers` at the command prompt and select two members. The two members must have a common joint (middle). After merging the middle joint is NOT deleted and you have to delete it manually. This joint can be removed manually,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

2.6.15. Member Properties

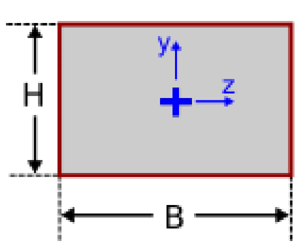
You can change the properties of a member from the ribbon command 

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.

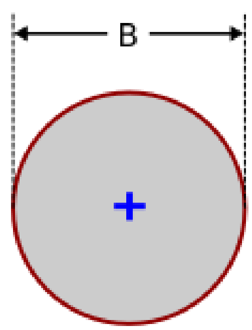
The "*Member Properties*" form appears, and at the "*Model*" tab you can enter the *Name* of the Member (up to 8 characters), the *Level* that the member belongs to (optional), the *Type* of the Member (Space Frame or Space Truss), *Starting* and *Ending* Joints, *Beta Angle*, the *Groups* that the member 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*.



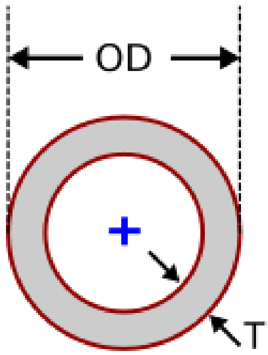
Using the “*Section Properties*” tab, you can define typical concrete shapes or other shapes that appear in the next image. Shapes having the identifier “Concrete” should be used for thick sections.



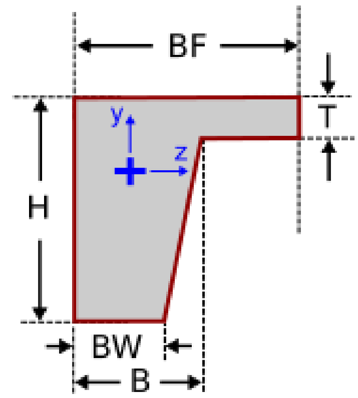
Rectangle (Concrete)



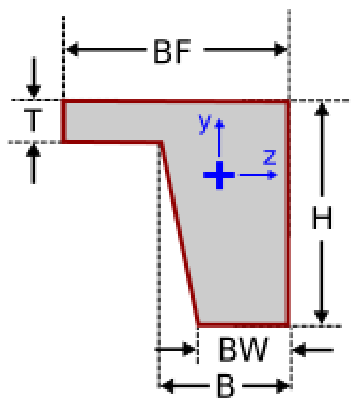
Circle (Concrete)



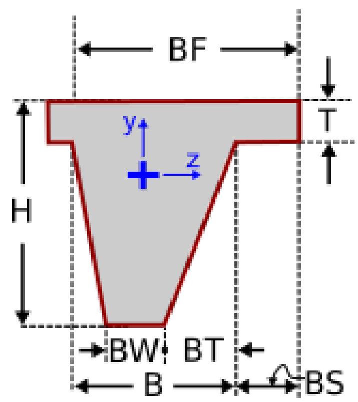
Pipe



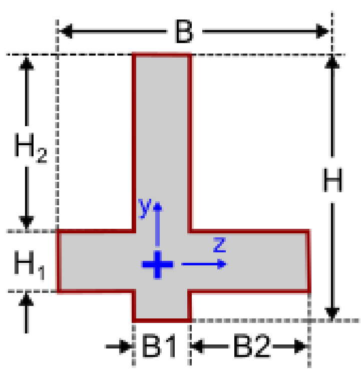
Right L (Concrete)



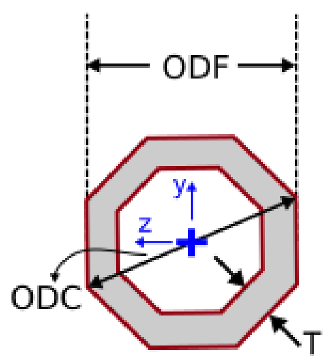
Left L (Concrete)



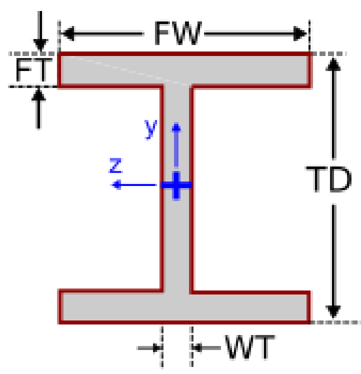
T Shape (Concrete)



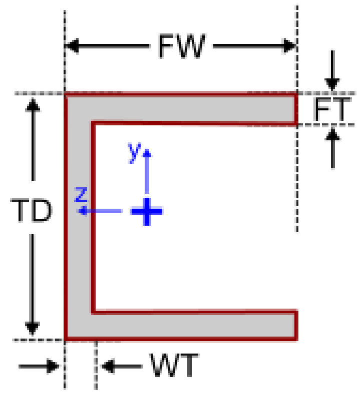
Cross (Concrete)



Polygon

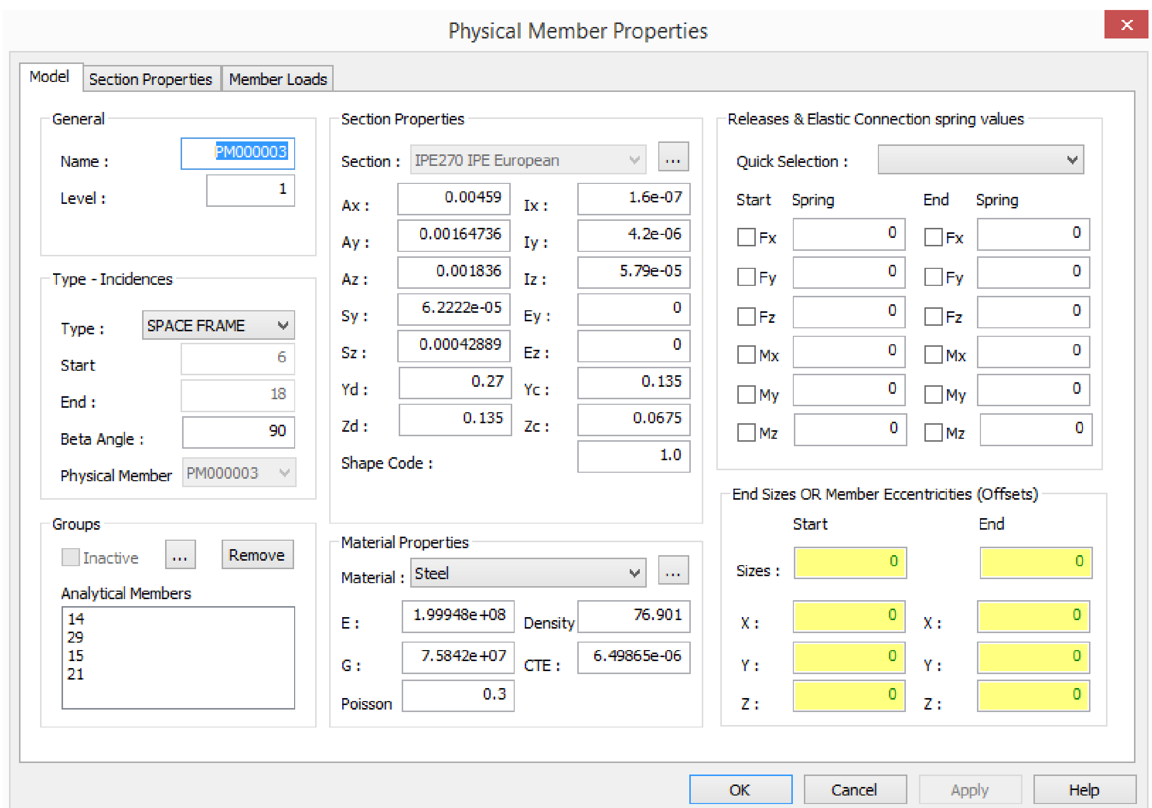


I Shape



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.



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 

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:

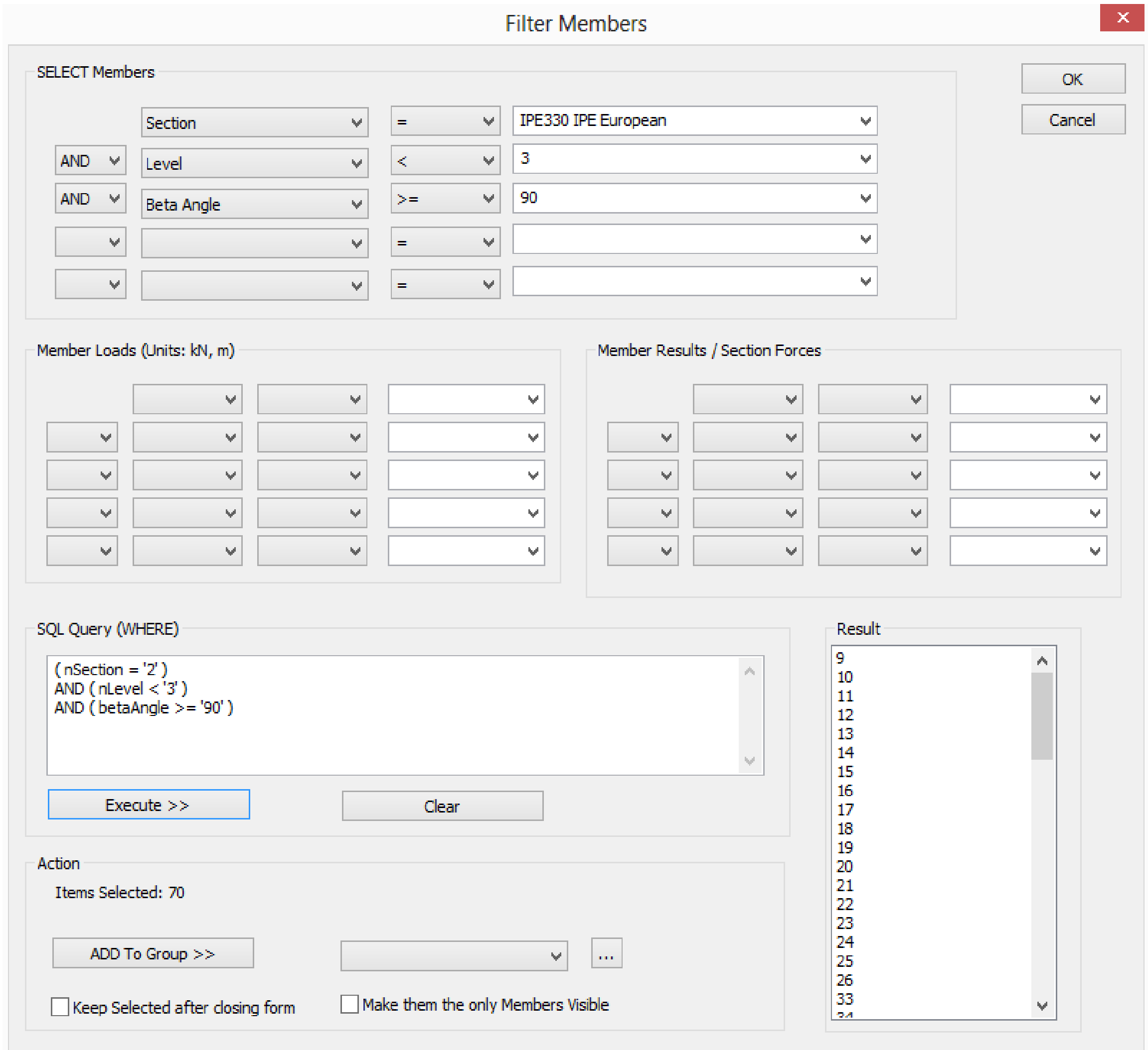
- **Their Properties**, that can be: Name, Level, Section, Material, Beta Angle, Group, Release Statues, Kf values, Eccentricities and End Sizes
- **Their Loading Data**, that can be: Load Case, Load Type, Load Direction, Load Values and Location.
- **Their Analysis Results**, that can be: Load Case, Member Forces Fx – Fy – Fz – Mx – My – Mz for both ends and section forces Fx - Fy – Fz - Mx – My – Mz.

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*"



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 

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 

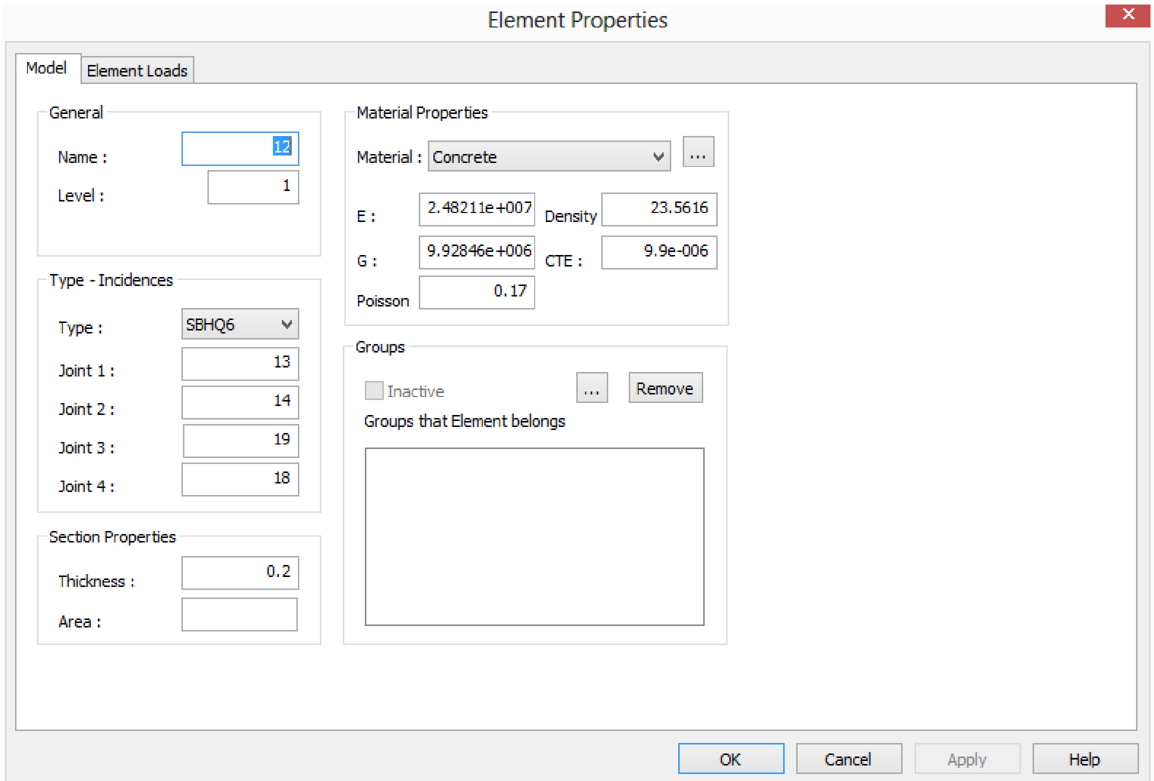
or the menu "*GTS Modeling>Shell>Find*" or by typing `GTSFEID` at the command prompt and enter the name of the element. If the element name exists, the shell element will be selected (by clicking on "change" you can modify it without making a new selection).

2.6.20. Shell Properties

You can change the properties of a shell finite element from the icon 

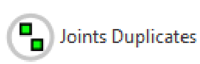
or the menu "*GTS Modeling>Shell>Change*" or by typing `GTSShellChange` at the command prompt and select the shell or the shells to be edited or by double-clicking on an existing shell element.

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.

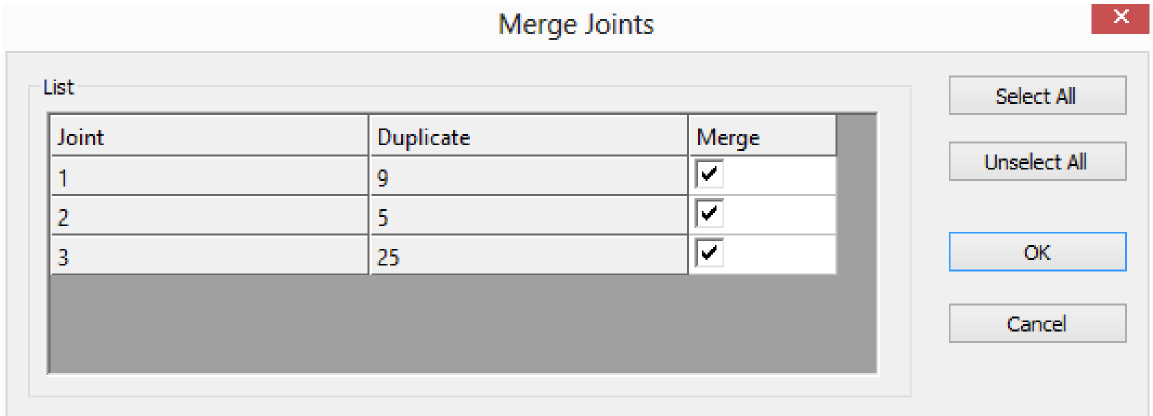


If you select more than one element, then "*Multiple Selection*" appears at the top of the Element Properties form, and all data entered in the form will be applied to all selected elements.

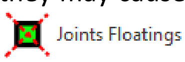
2.6.21. Joints Duplicates

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 

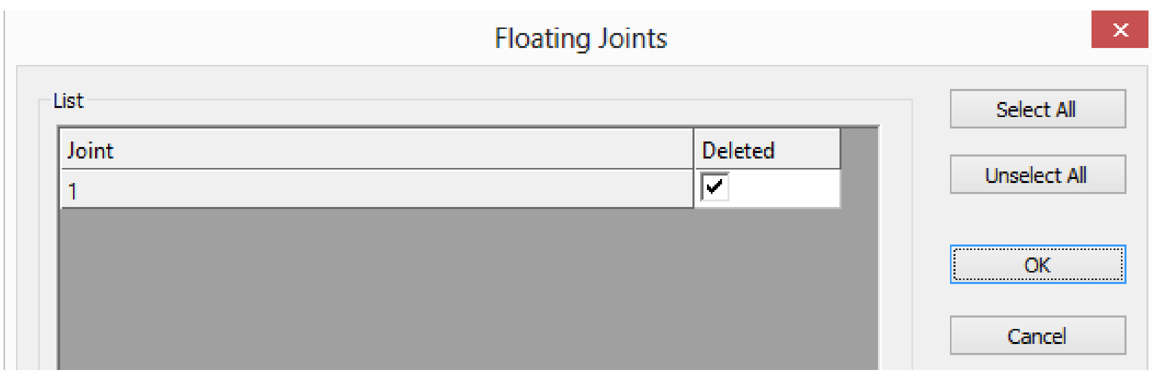
in the "Find/Change/Check" panel or select "*GTS Modeling>Checks>Joint Duplicates*" from the menu or type `GTSCheckDuplicateJoints` at the command prompt. You then must enter the desired merge accuracy (*Enter Merge Tolerance <0.001000>*). If duplicate joints exist in the structure, a new dialog appears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



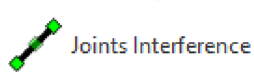
2.6.22. Joints Floatings

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 

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.



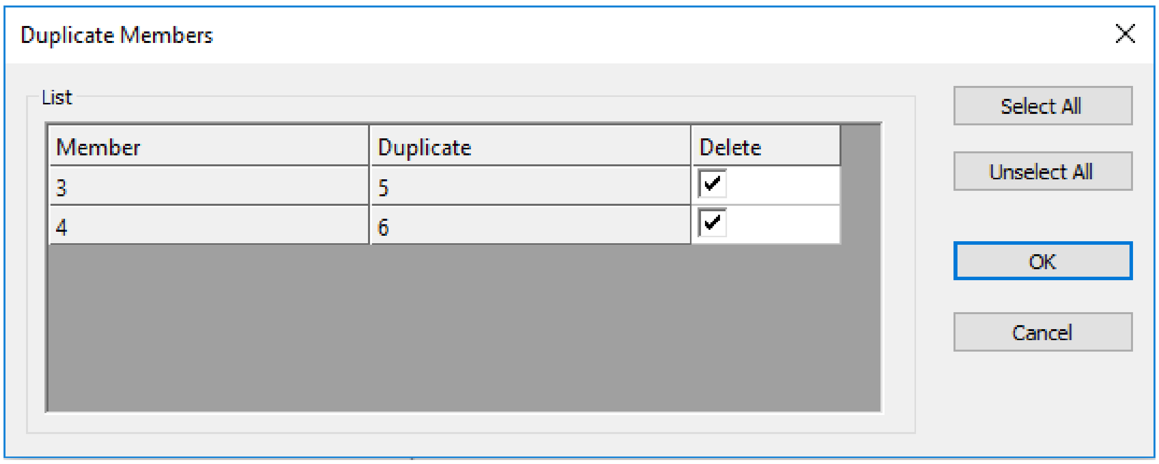
2.6.23. Joints Interference

In order to check in order to search for joints close to a member, not connected to it, you can use the ribbon command 

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 command Physical Members 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)
```

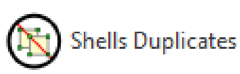
#5.Check that all members form a straight line. . If not, there is an error message:

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

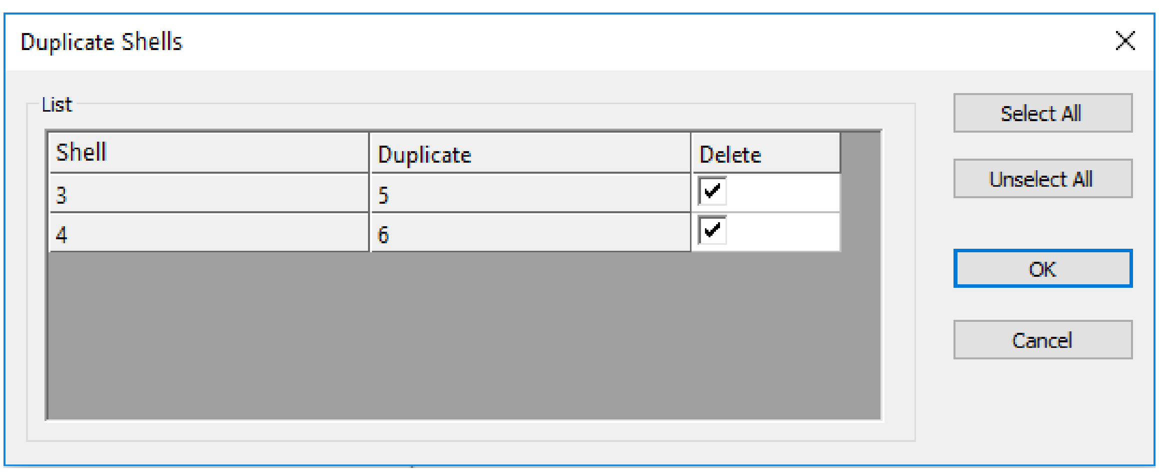
If all 5 checks are successful an informative message appears:

```
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 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).



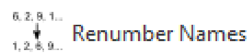
2.6.28. Names Duplicates

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 

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
```

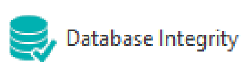
2.6.29. Renumber Names

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 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 mmust 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 the ribbon command 

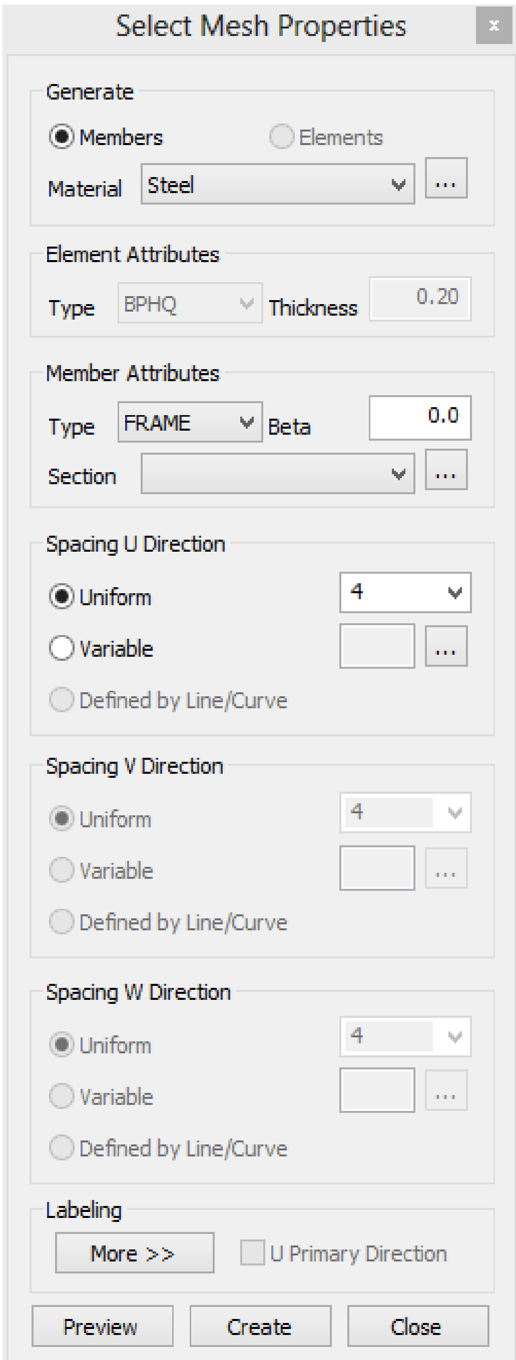
in the "Find/Change/Check" panel or select "*GTS Modeling>Checks>Database Integrity*" from the Menu or type `GTSCheckDatabase` at the command prompt. There may be database records, without a dwg entity, or the opposite, dwg entities without a database record. This may happen in very extreme cases (for 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 

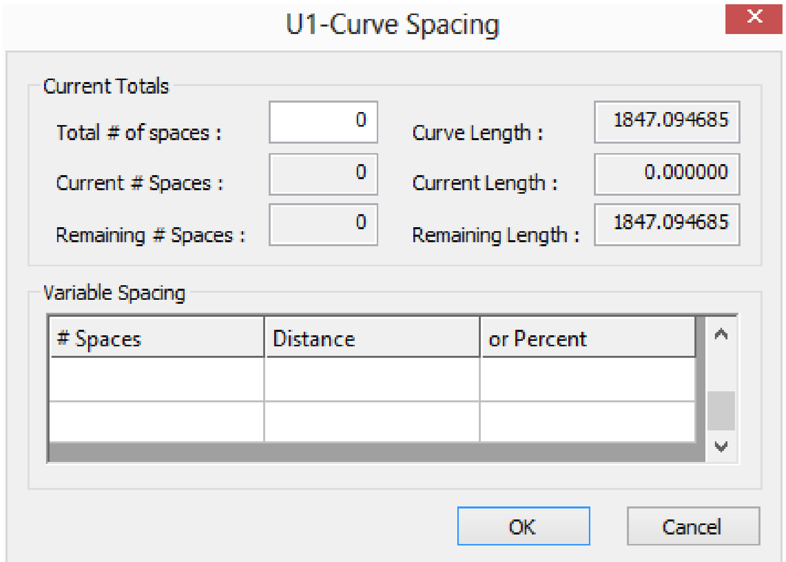
or from the menu "*GTS Modeling>Mesh Generation>1D Along Line or Curve or Circle*" or by typing GTSMesh1D at the command prompt.



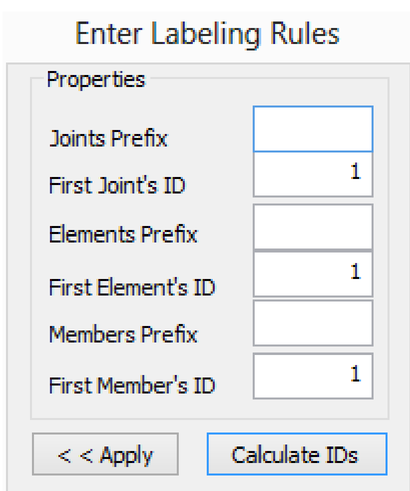
After selecting the AutoCAD/BricsCAD linear entity the Mesh Properties dialog appears, where you can define:

- 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 shownbelow:.



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).



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 

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.

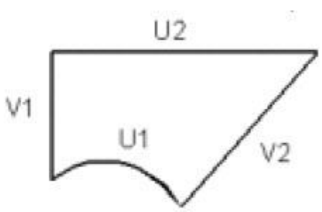
The dialog has the same options as in the 1D mesh command and in addition you can also define:

- 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 <u>Members OR Finite Elements</u> between four selected AutoCAD/BricsCAD linear entities, that can be Lines or Arcs, from the ribbon command 

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:



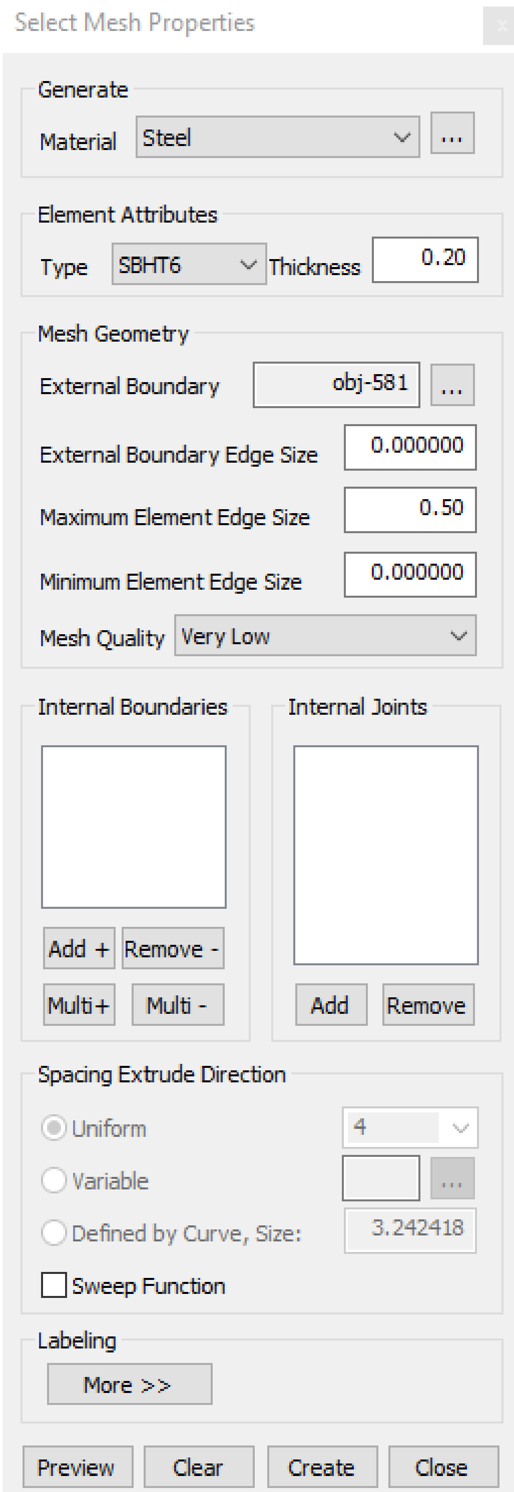
The dialog has the same options as in the 2D mesh between two curves command.

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 

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:



- 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.

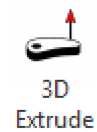
NOTE: Internal boundaries may intersect the external mesh curves, but it is recommended that you manually spit the external curve at this point

- 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.

-

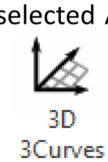
2.6.35. Meshing by extruding a polyline

You can create Finite Elements by extruding an AutoCAD/BricsCAD closed curve, that can be a Polyline or a Circle, from the ribbon command 

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 <u>Members OR Finite Elements</u> between three selected AutoCAD/BricsCAD linear entities, that can be Lines or Arcs, from the ribbon command 

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 

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.

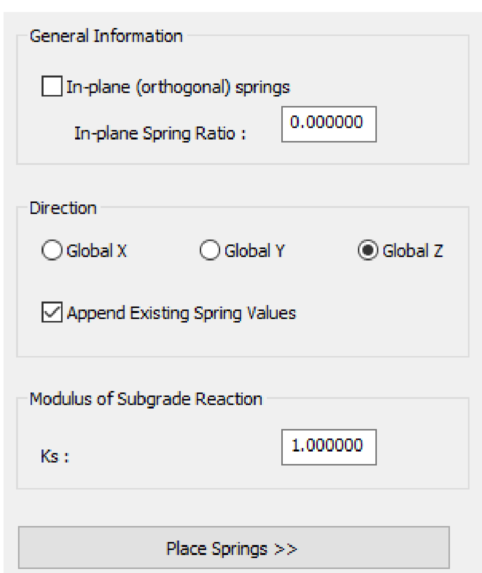
2.6.38. Soil Springs

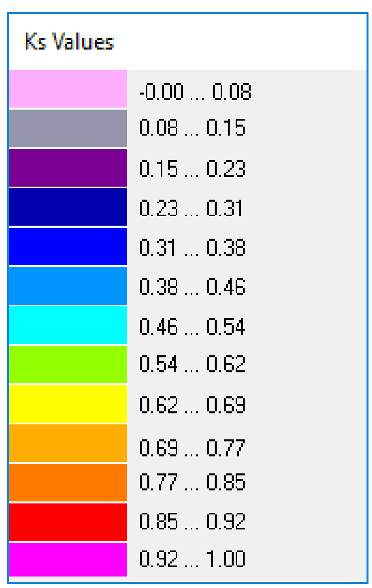
You can create soil springs on individual/multiple shell element(s) from the icon 

or by typing `GTSFoundationSprings` 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.





2.6.39. Export to CAESAR II

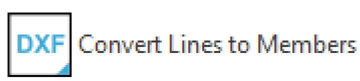
The current drawing can be exported into CAESAR II Modeler (str file) from the menu “*GTS Modeling>CAESAR II> Export STR*” or by typing `GTSExportSTR` at the command prompt. You select the members to be exported and an STR File is generated. Immediately after the log file appears on the screen. A typical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

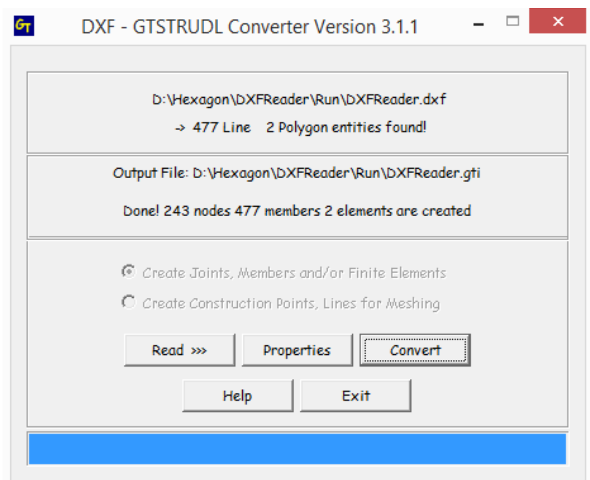
If the cross sections used in CAD Modeler (and GT STRUDL) are not available in CAESAR’s section library you get a warning like this:

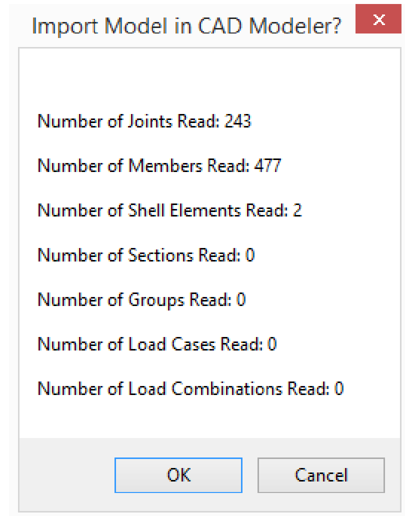
```
WARNING: Section L1x1x1/4 is not available in CII, please use another
one or edit
F:\\HexagonPPM\CaesarII\PlantStructure\\PStructure_0708_01.str file
manually
```

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 

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.

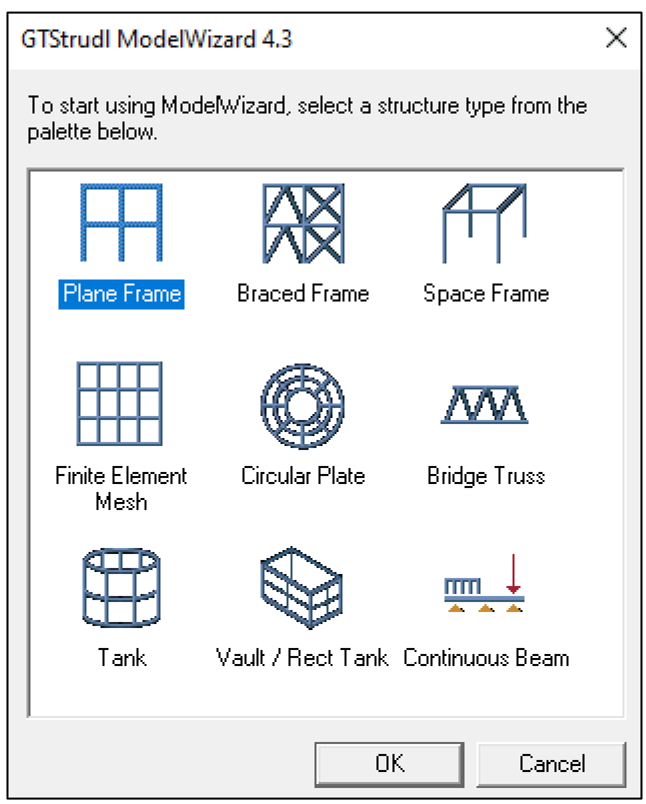




2.6.41. Model Wizard

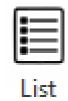
You can now use Model Wizard to create and import the model to CAD Modeler. This option is very useful if you want to create a typical tank and further edit it in CAD Modeler, or append it into an existing model. You can start the Model Wizard from the ribbon command

or by typing `GTSModelWizard` at the command prompt.



2.6.42. Groups

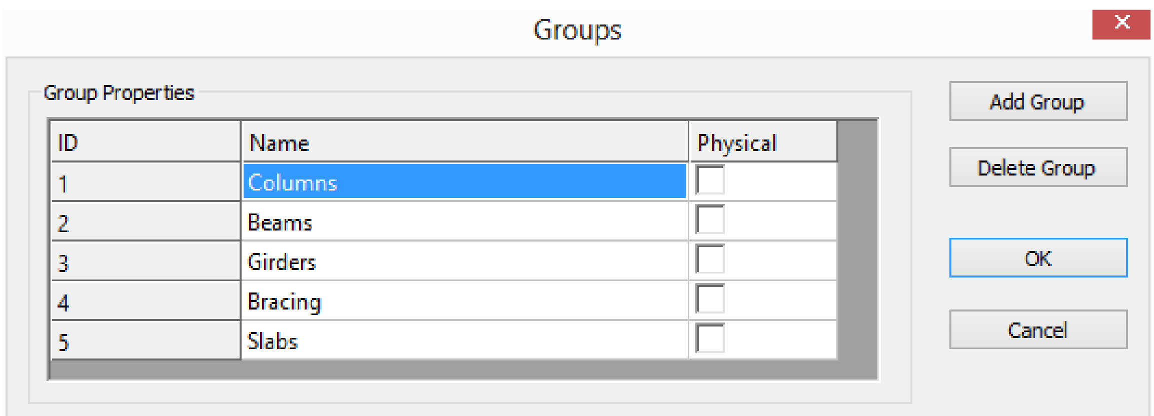
It is optional to define Groups in your model, but it is strongly advised to do so, since it will be easier to control the display and modeling of parts of your structure. Each Group is defined as a set of joints, members and finite elements. Each structural entity can belong in more than one group. 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 

or from the menu "*GTS Modeling>Groups>Manage*" or by typing `GTSGroups` at the command prompt.

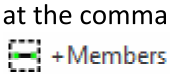
Using the Groups Form you can:

- 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).

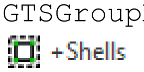


After defining a group you can add joints, members and shell elements to it using the commands:

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



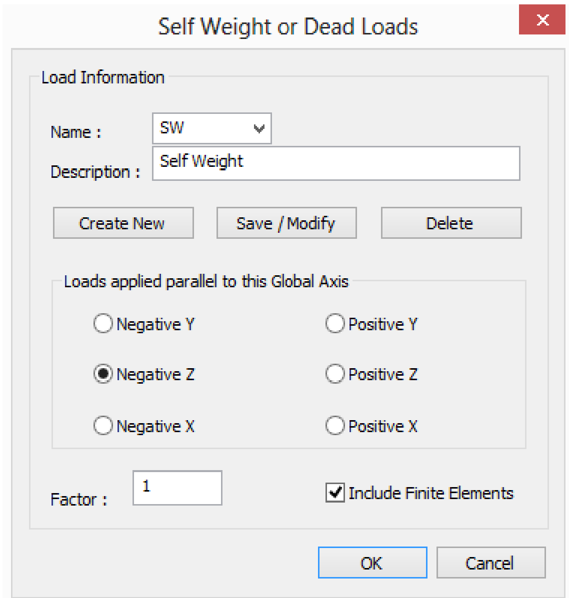
ribbon icon, or "*GTS Modeling>Groups>Add Members*" or by typing `GTSGroupMembers` at the command prompt



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.

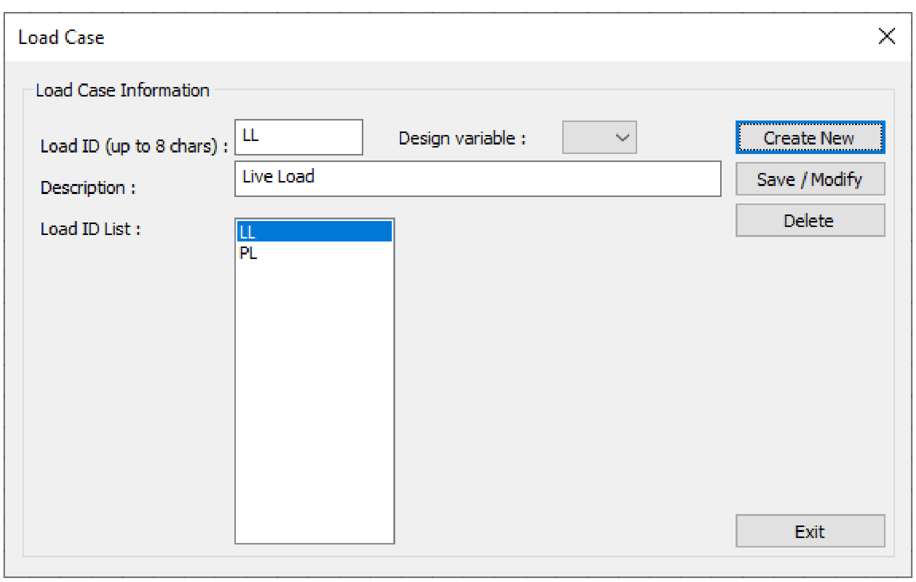


The "*Self-Weight*" form appears where you can define:

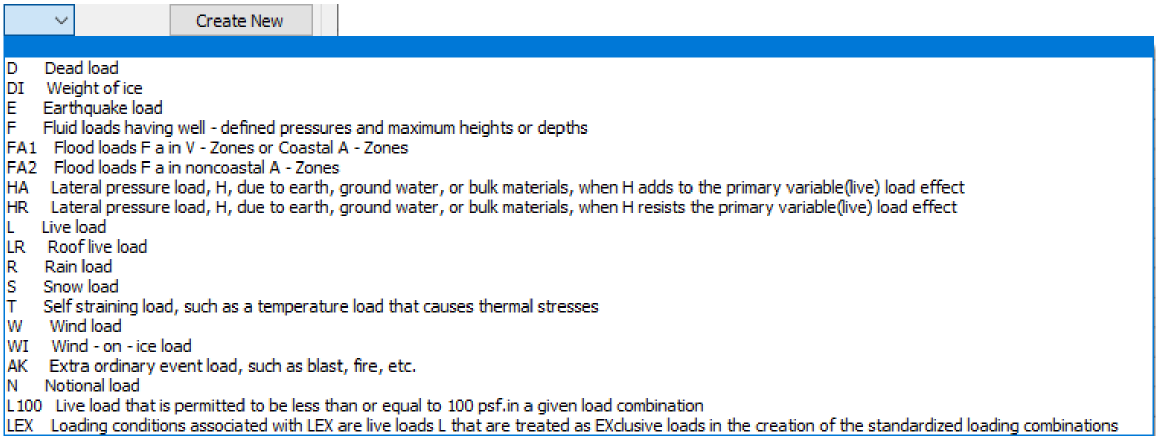
- the global direction of the self-weight
- the load factor (default = 1.0) for the self-weight
- if the self-weight of finite elements will be taken into account or not

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.



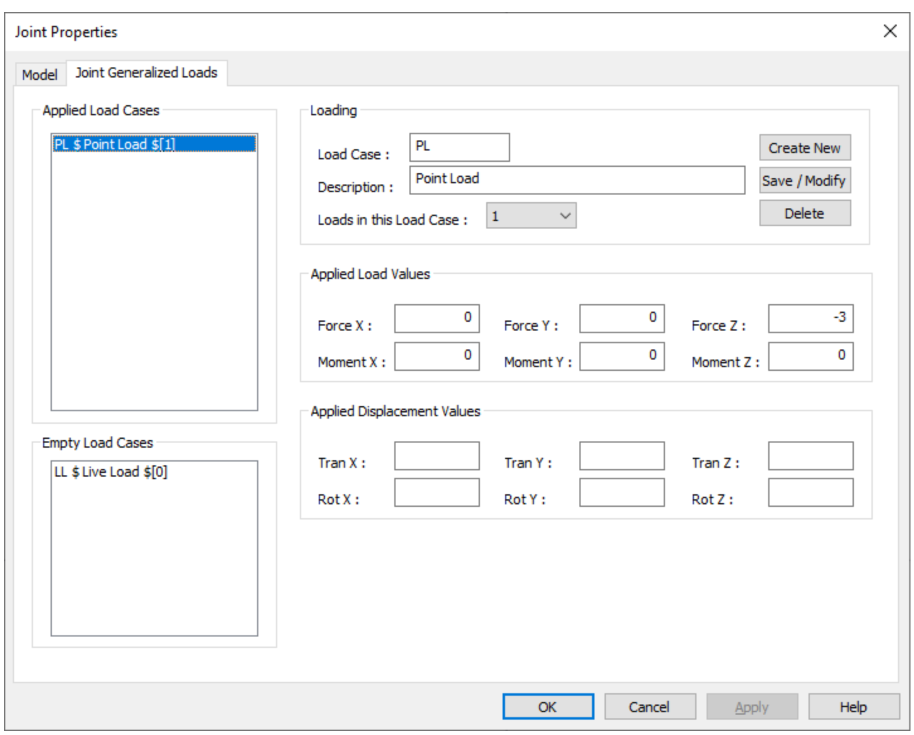
Design variable associates the Load Case with the Design Load Variable. This information can be used later on, when creating the standardized load combinations.



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.

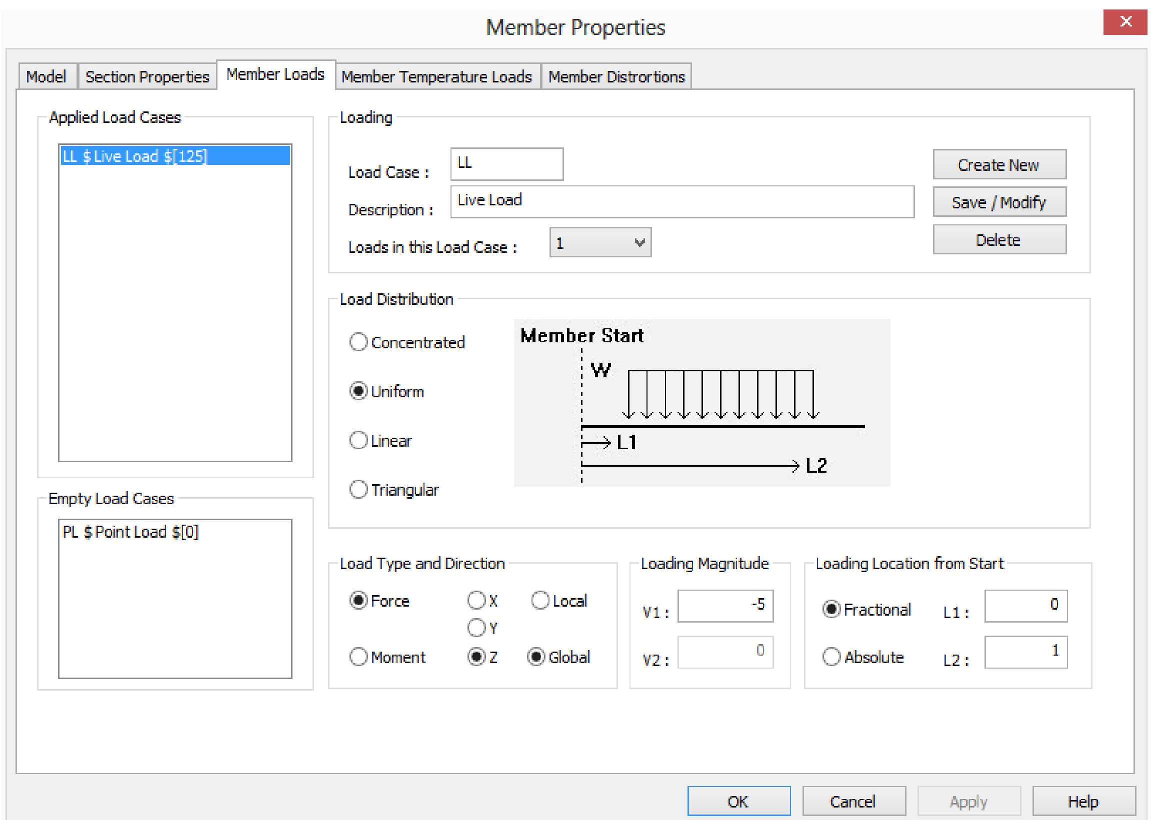
At the "*Joint Generalized Loads*" tab you can apply joint loads or displacements. On the left part of the form, you can see a list of all available load cases: Load cases having loads already applied to the specific joint appear at the top list box. Load cases that do not have any loads applied to the 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.



2.6.46. Member Loads

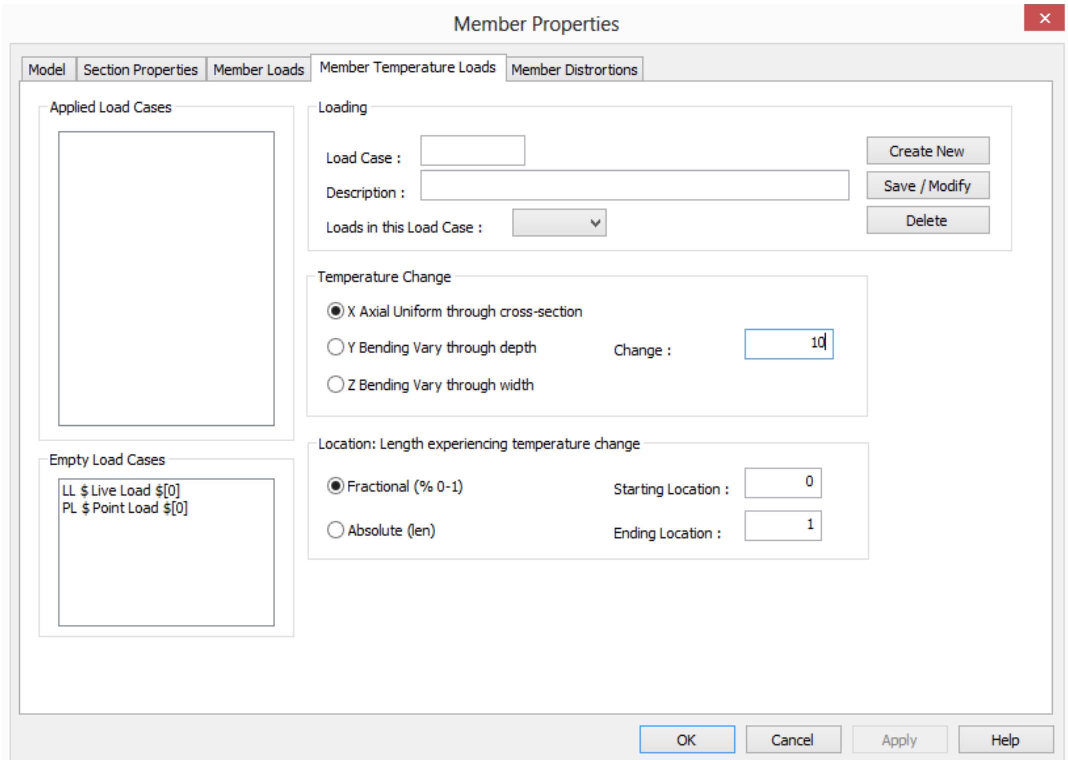
A Member Load can be entered from the ribbon command Member or from the menu "*GTS Modeling>Loads>Member Load*" or by typing `GTSBeamLoad` at the command prompt. You then have to select the member or the members that the load will be applied to.

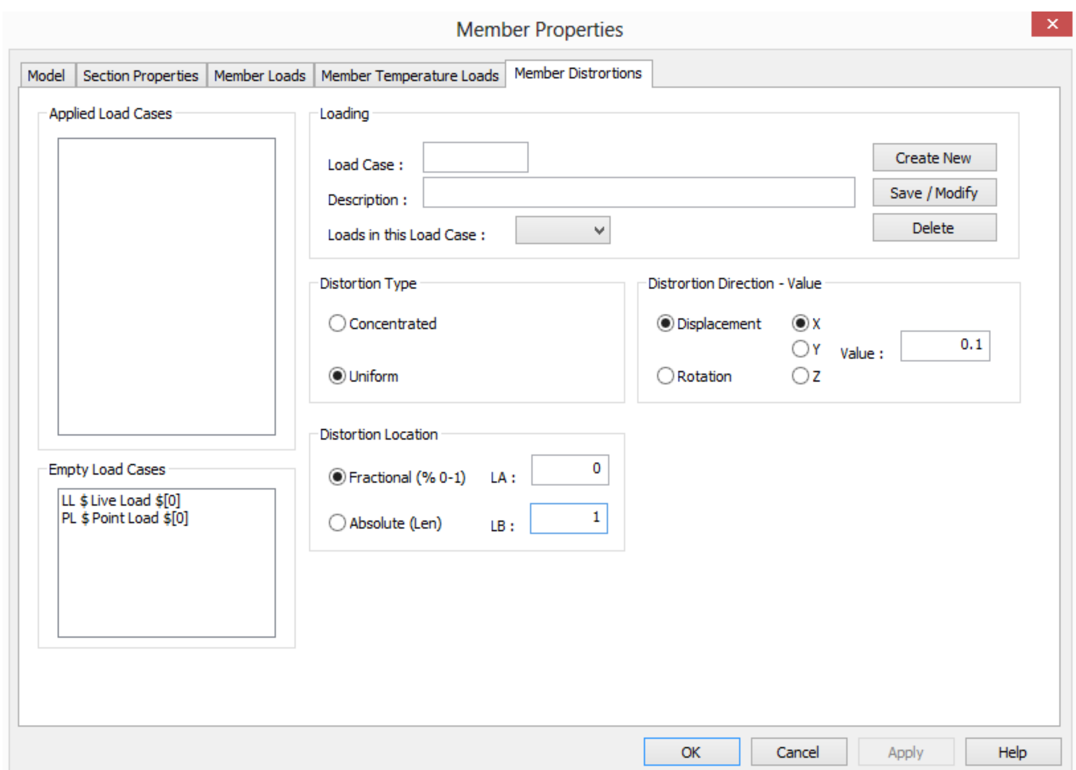
At the "*Member Loads*" tab or the Member Properties form as shown below, you can apply member loads. In the left part of the form you can see a list of all available load cases: Load cases having loads already applied to the specific member appear in the top list box. Load cases that do not have any 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.



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 of uniform distortion of the member in any direction along the member as shown on the next page.

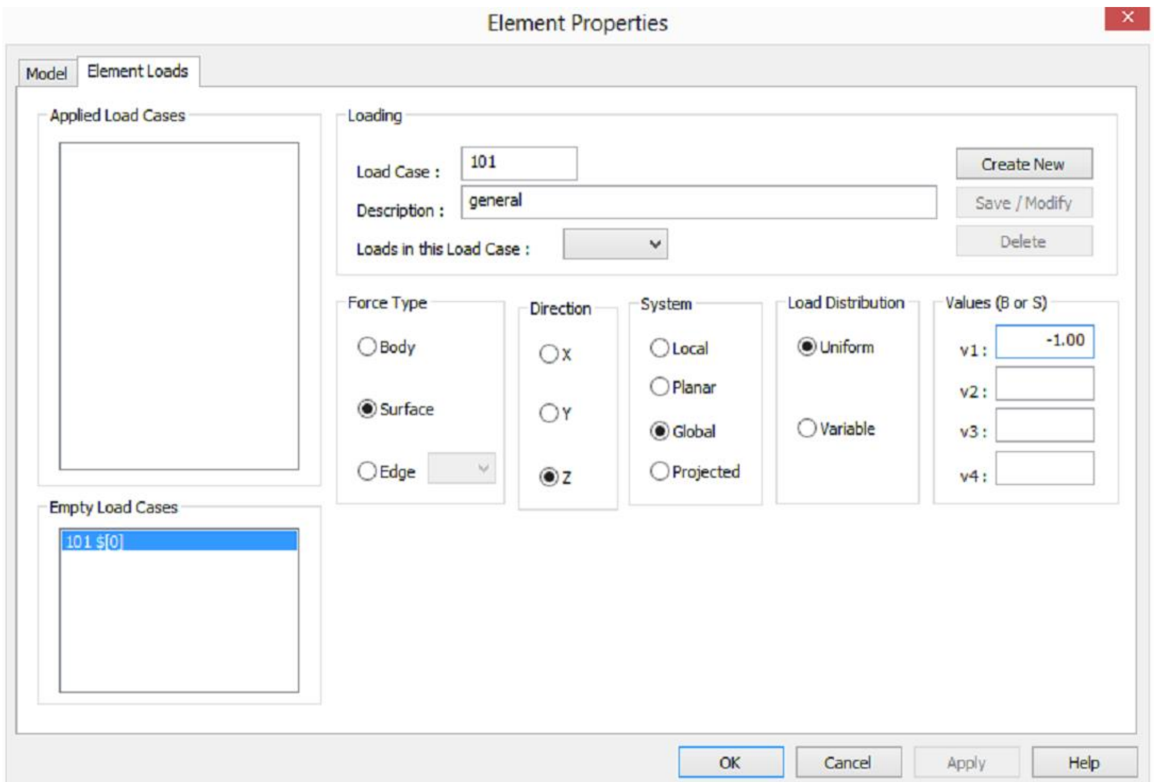

Member Properties
Model
Section Properties
Member Loads
Member Temperature Loads
Member Distrortions
Applied Load Cases
Loading
Load Case :
Description :
Loads in this Load Case :
Create New
Save / Modify
Delete
Temperature Change
X Axial Uniform through cross-section
Y Bending Vary through depth
Z Bending Vary through width
Change :
10
Location: Length experiencing temperature change
Fractional (% 0-1)
Absolute (len)
Starting Location :
0
Ending Location :
1
Empty Load Cases
LL $ Live Load $[0]
PL $ Point Load $[0]
OK
Cancel
Apply
Help


Member Properties
Model
Section Properties
Member Loads
Member Temperature Loads
Member Distrortions
Applied Load Cases
Loading
Load Case :
Description :
Loads in this Load Case :
Create New
Save / Modify
Delete
Distortion Type
Concentrated
Uniform
Distrortion Direction - Value
Displacement
Rotation
X
Y
Z
Value :
0.1
Distortion Location
Fractional (% 0-1)
Absolute (Len)
LA :
0
LB :
1
Empty Load Cases
LL $ Live Load $[0]
PL $ Point Load $[0]
OK
Cancel
Apply
Help

2.6.47. Shell Loads

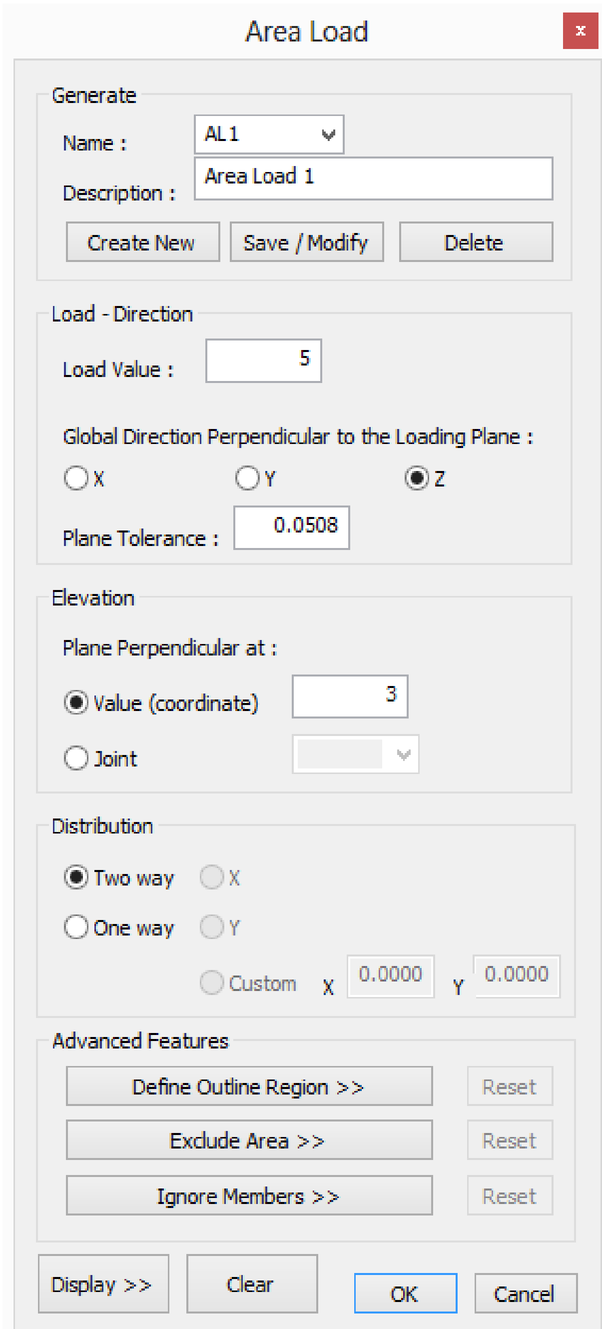
A Shell Load can be entered from the ribbon command Shell or from the menu "*GTS Modeling>Loads>Shell Load*" or by typing `GTSShellLoad` at the command prompt. You then have to select the shell or the shells that the load will be applied to.

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.



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.



Using the Area Load form you can define:

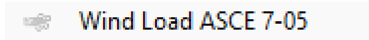
- The Name of the Load (up to 8 characters)
- The description of the load
- The Loading Value in current units (force/length^2). A positive value is applied in the negative global DIRECTION.
- The Global direction of the loading plane (X, Y or Z) and the tolerance
- The position (Elevation) of the plane, 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:

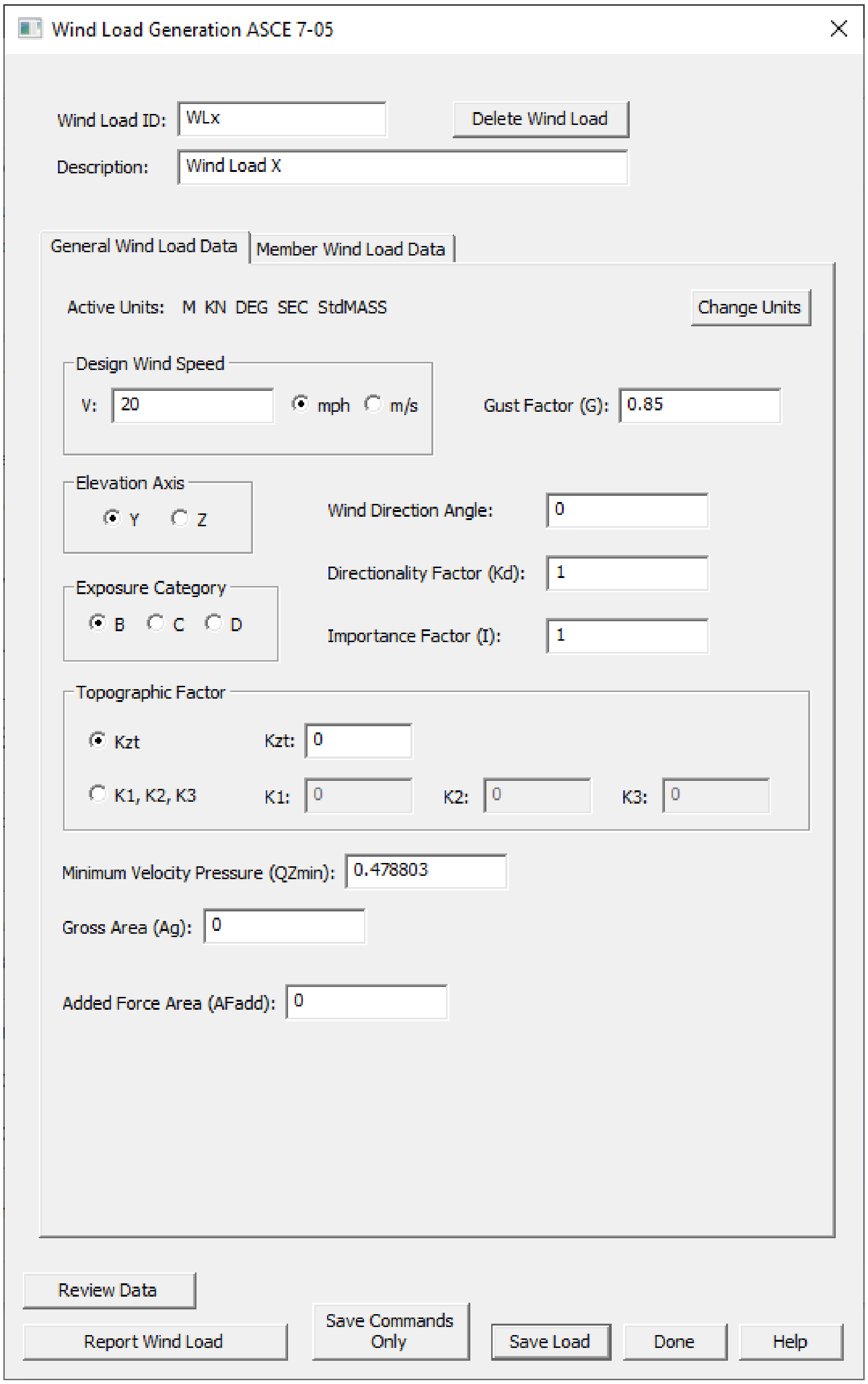
- Outline Region: Select the members that form the closed perimeter of the loaded area. If an Outline Region is defined then the area load is applied ONLY to the selected area and NOT to the whole plane having the elevation defined above.
- Exclude Area: Internal openings or islands not being loaded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

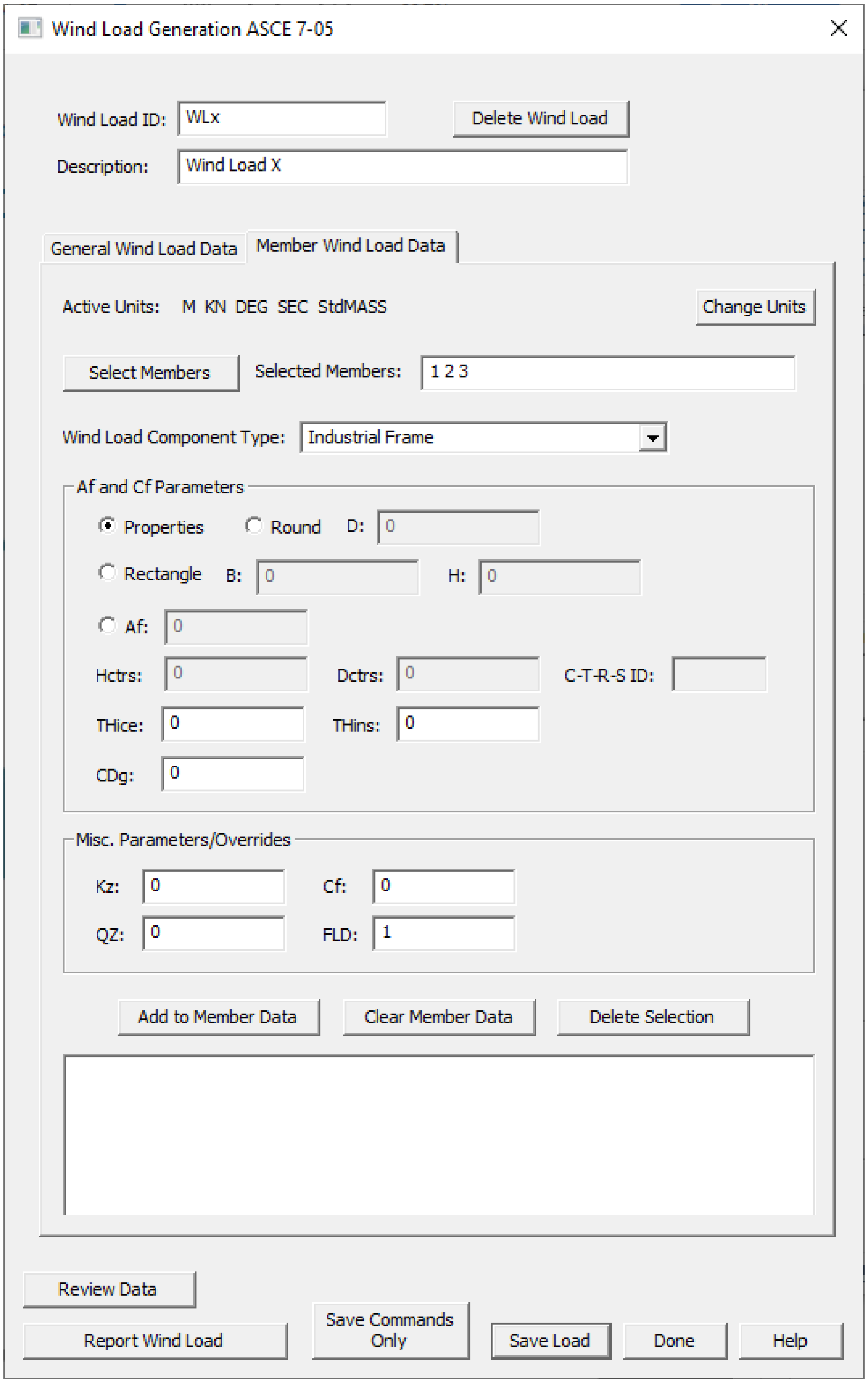
2.6.49. Wind Load ASCE 705

You can define Wind Loads using a similar form with GT Shell. A Wind Load can be entered from the ribbon command 

or from the menu *"GTS Modeling>Loads>Wind Load ASCE 705"* or by typing `GTSWindLoadsASCE705` at the command prompt.



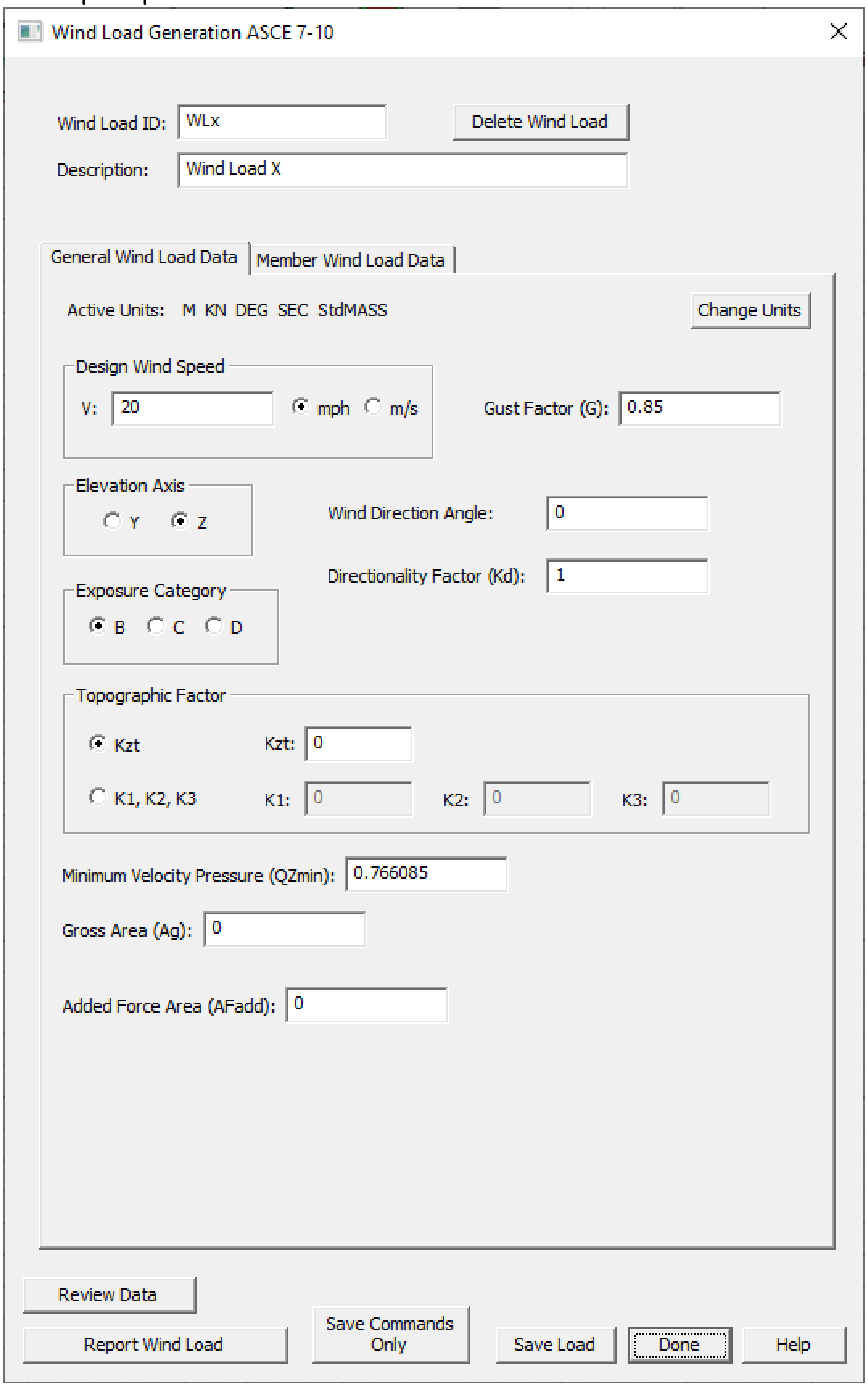
In addition, you can select members (to apply loads) interactively with mouse picks in the CAD environment.

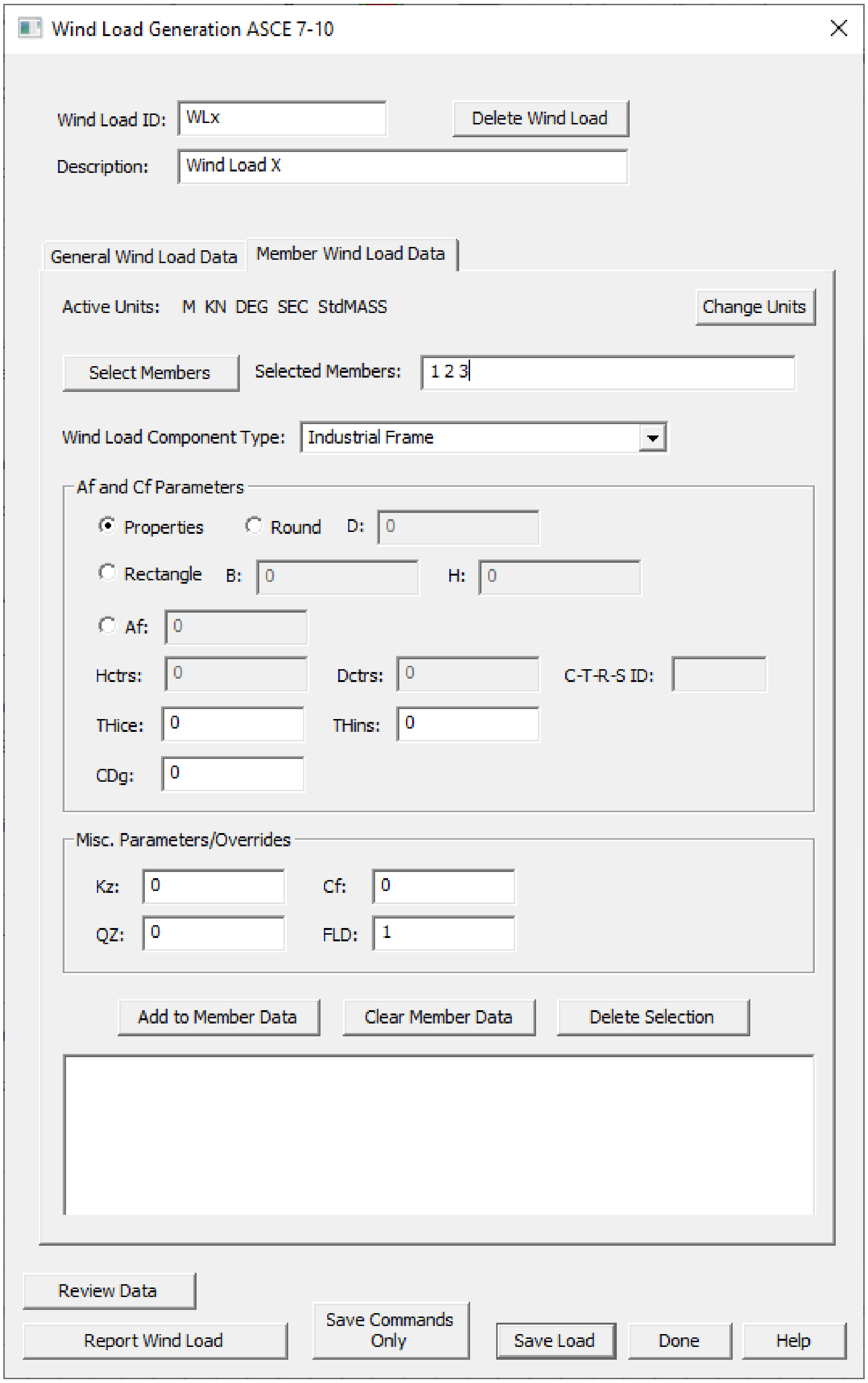


2.6.50. Wind Load ASCE 710

A Wind Load can be entered from the ribbon command 

or from the menu "*GTS Modeling>Loads>Wind Load ASCE 710*" or by typing `GTSWindLoadsASCE710` at the command prompt.



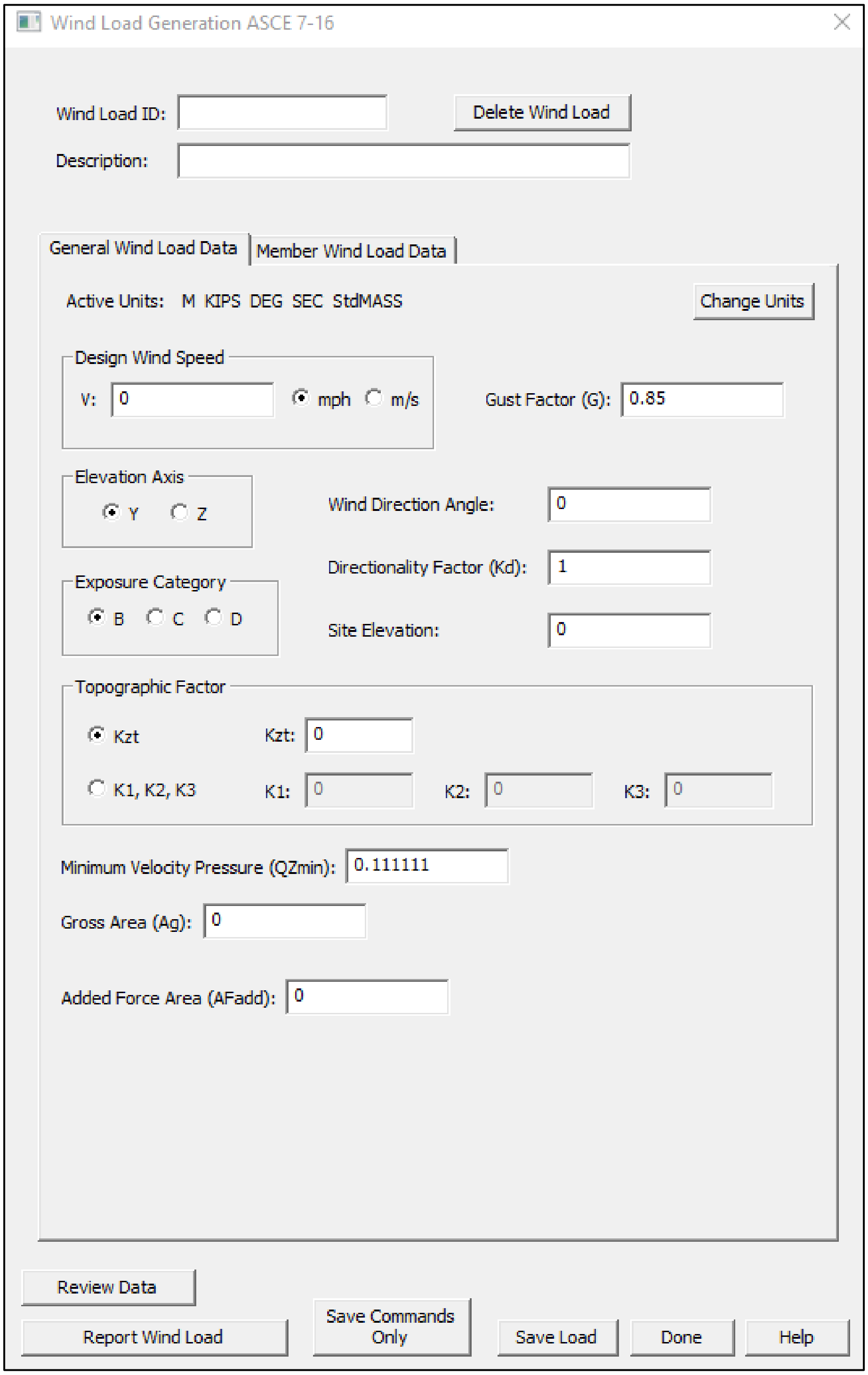


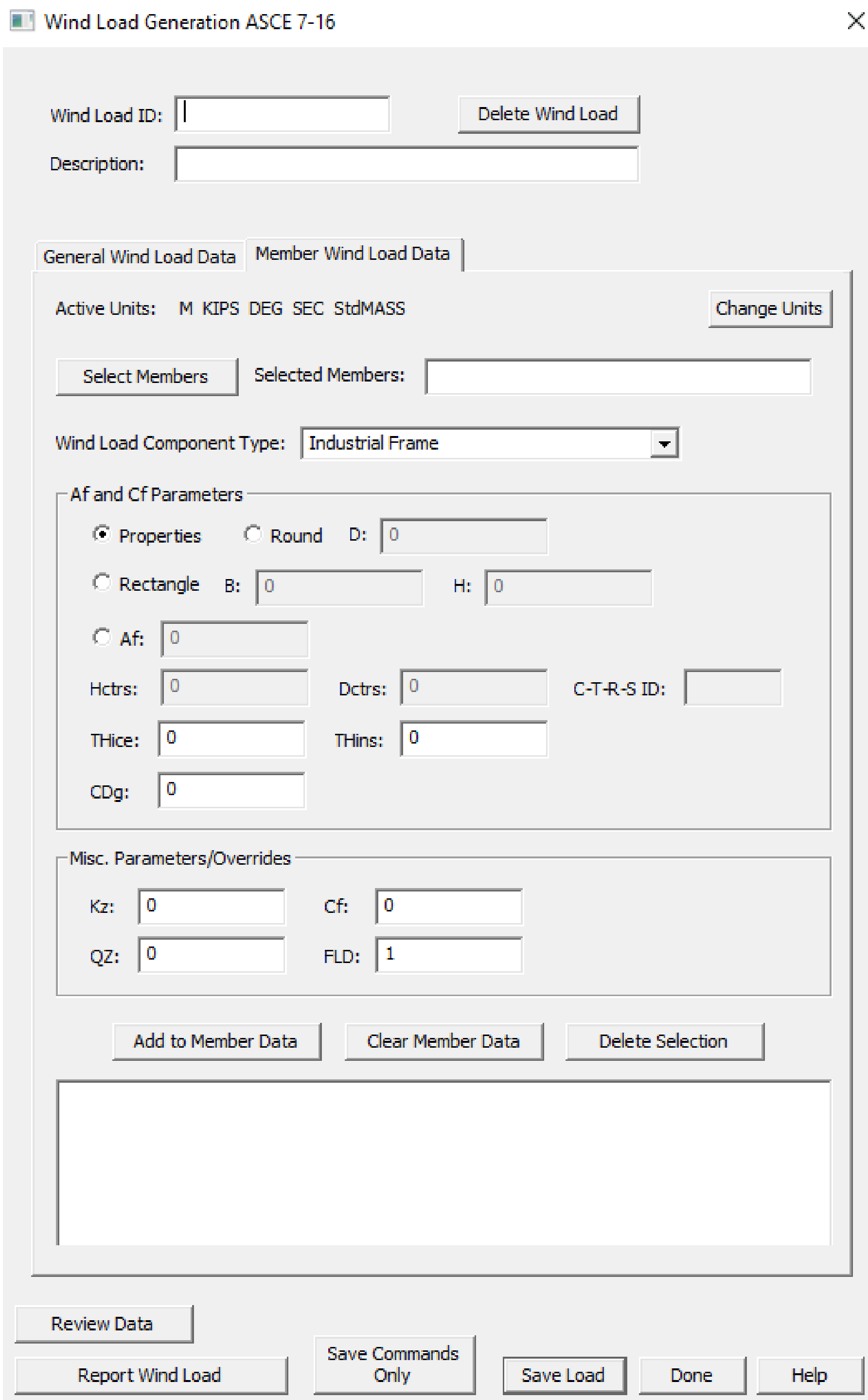
Wind Load Generation ASCE 7-10
Wind Load ID:
WLx
Delete Wind Load
Description:
Wind Load X
General Wind Load Data
Member Wind Load Data
Active Units: M KN DEG SEC StdMASS
Change Units
Select Members
Selected Members:
1 2 3
Wind Load Component Type:
Industrial Frame
Af and Cf Parameters
Properties
Round
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 

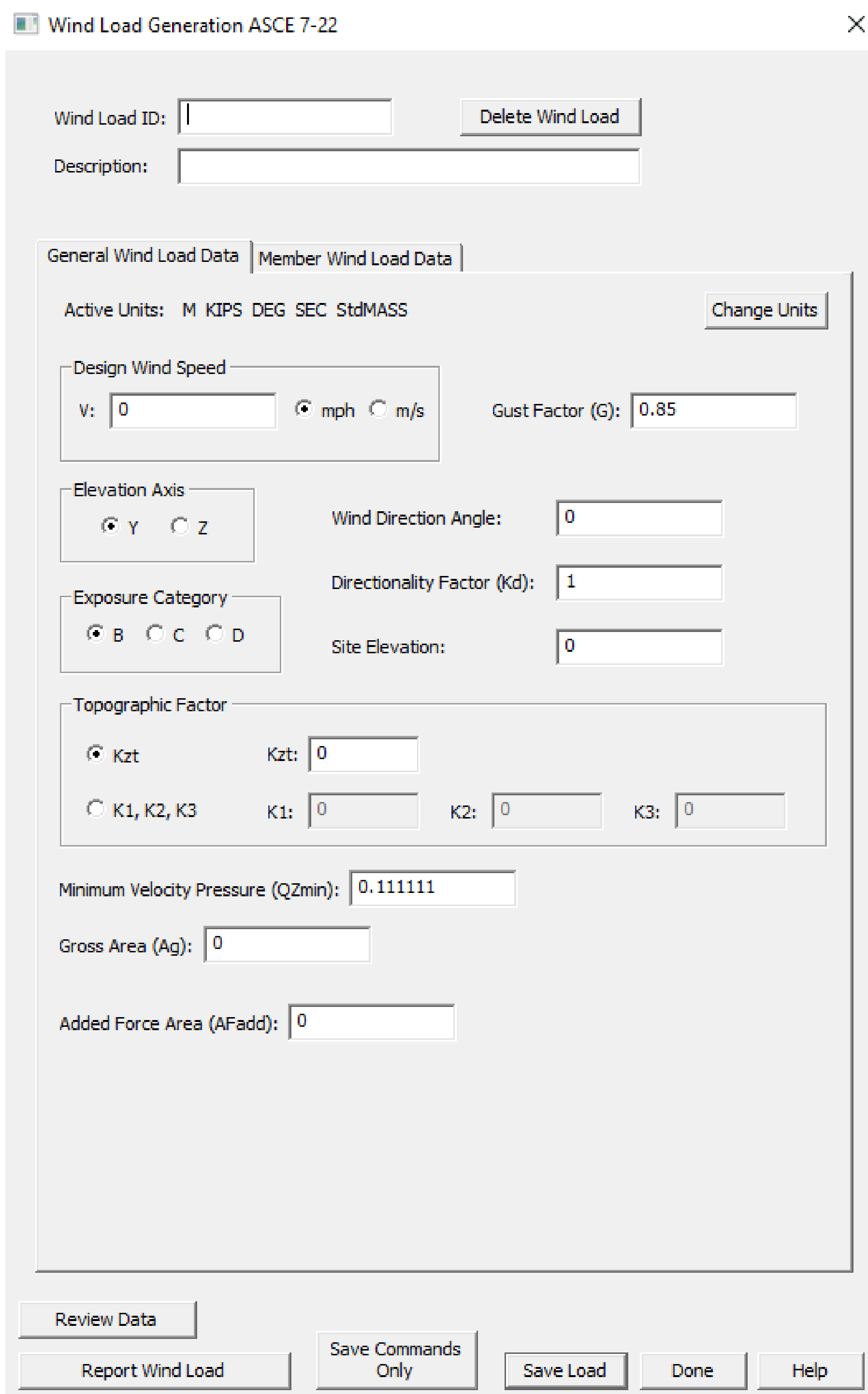
or from the menu "*GTS Modeling>Loads>Wind Load ASCE 716*" or by typing `GTSWindLoadsASCE716` at the command prompt.

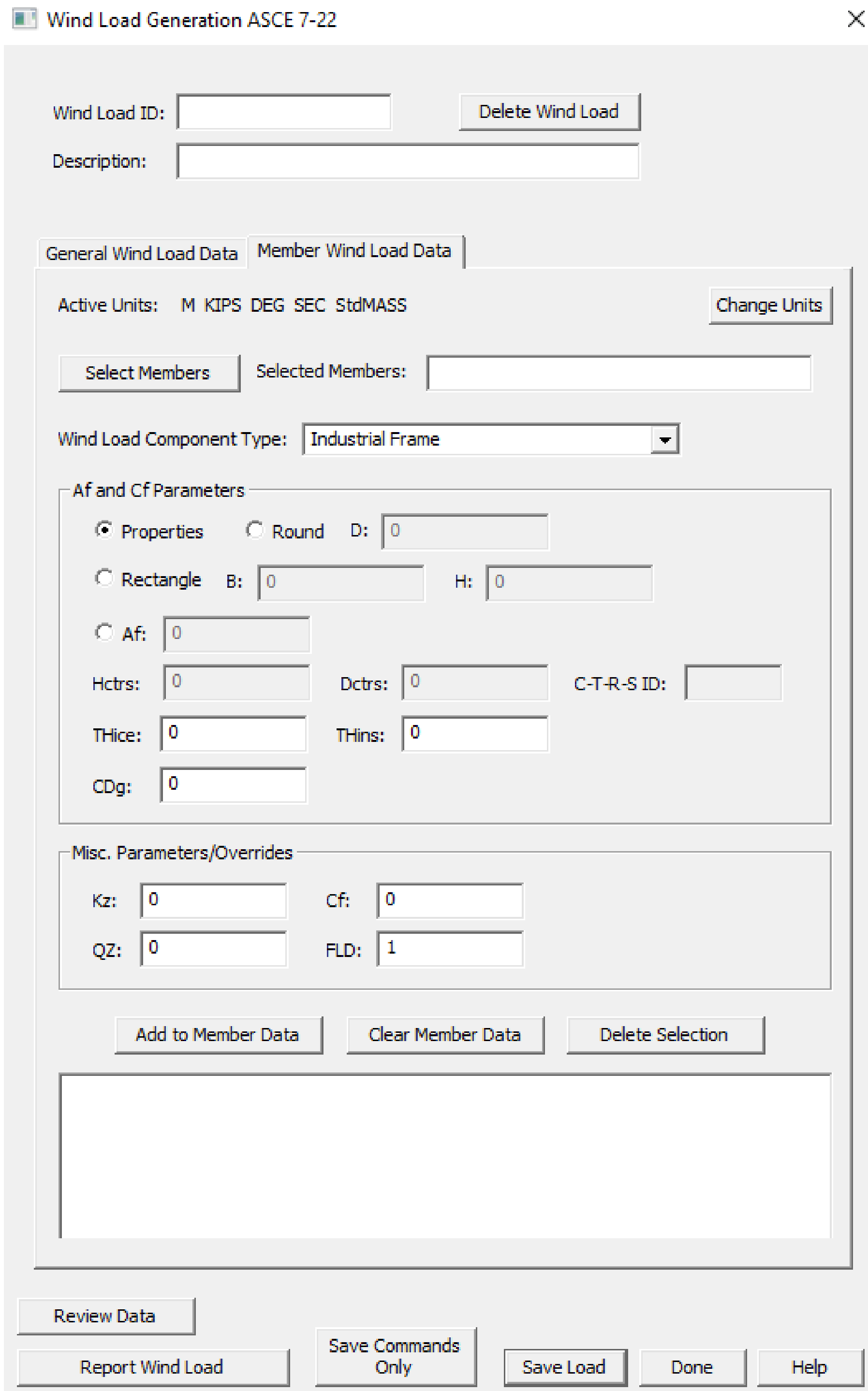



Wind Load Generation ASCE 7-16
Wind Load ID:
Delete Wind Load
Description:
General Wind Load Data
Member Wind Load Data
Active Units: M KIPS DEG SEC StdMASS
Change Units
Select Members
Selected Members:
Wind Load Component Type:
Industrial Frame
Af and Cf Parameters
Properties
Round
D:
Rectangle
B:
H:
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

2.6.52. 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.

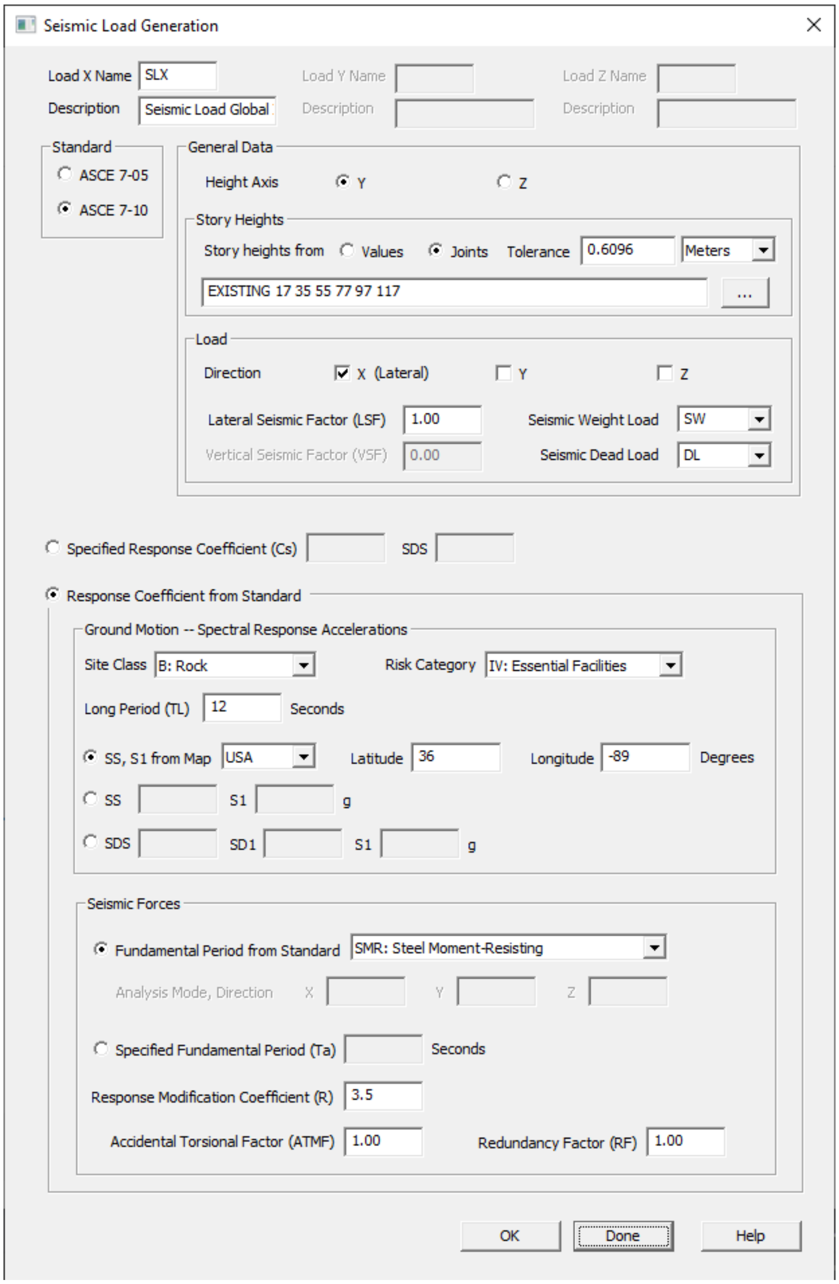



Wind Load Generation ASCE 7-22
Wind Load ID:
Delete Wind Load
Description:
General Wind Load Data
Member Wind Load Data
Active Units: M KIPS DEG SEC StdMASS
Change Units
Select Members
Selected Members:
Wind Load Component Type:
Industrial Frame
Af and Cf Parameters
Properties
Round
D:
0
Rectangle
B:
0
H:
0
Af:
0
Hctrs:
0
Dctrs:
0
C-T-R-S ID:
THice:
0
THins:
0
CDg:
0
Misc. Parameters/Overrides
Kz:
0
Cf:
0
QZ:
0
FLD:
1
Add to Member Data
Clear Member Data
Delete Selection
Review Data
Report Wind Load
Save Commands Only
Save Load
Done
Help

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 

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.

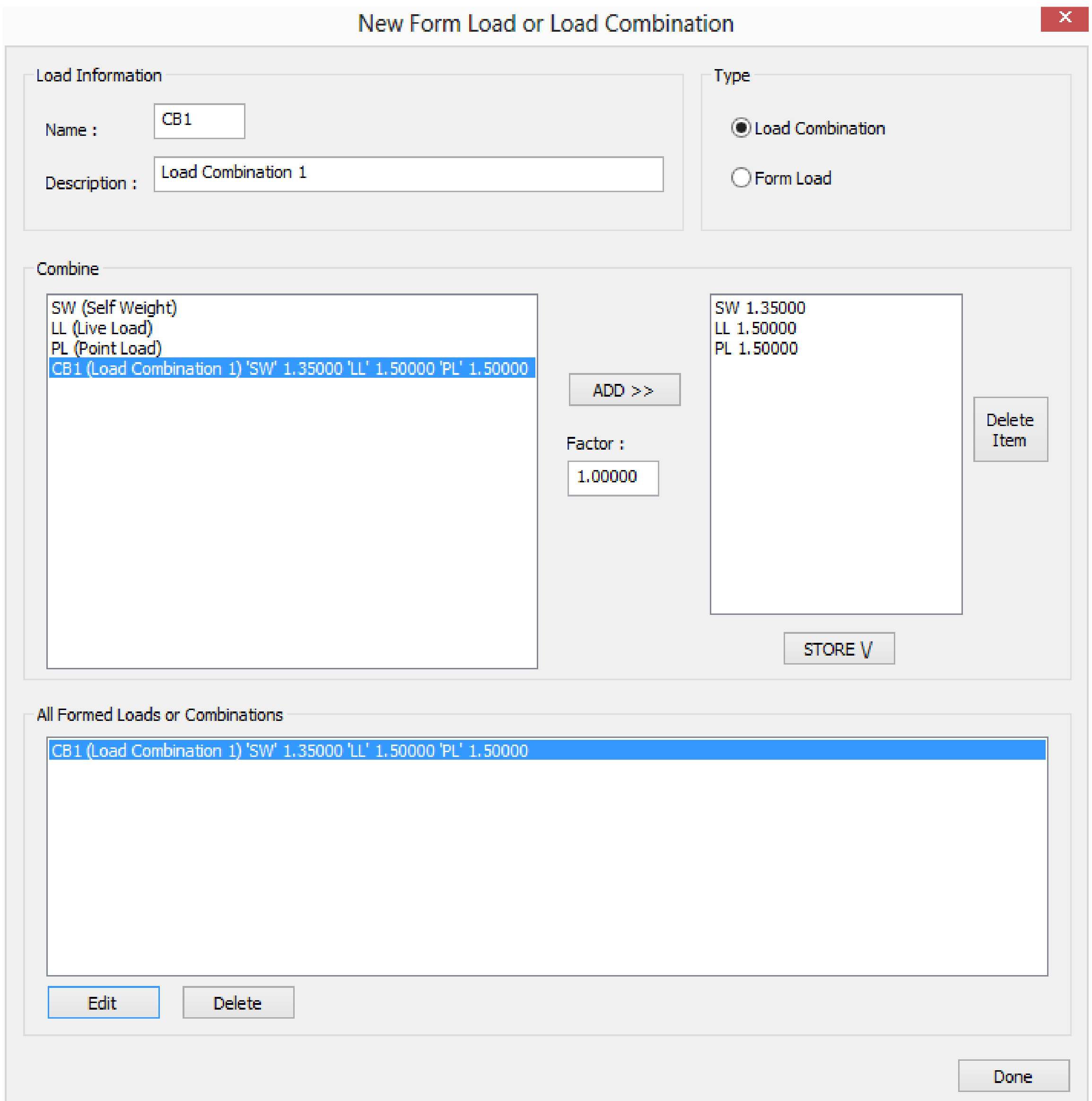


2.6.54. Load Combinations

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.

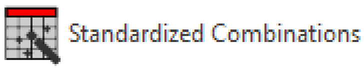
When all load cases are added, press the "*STORE*" button to store the load combination.

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.

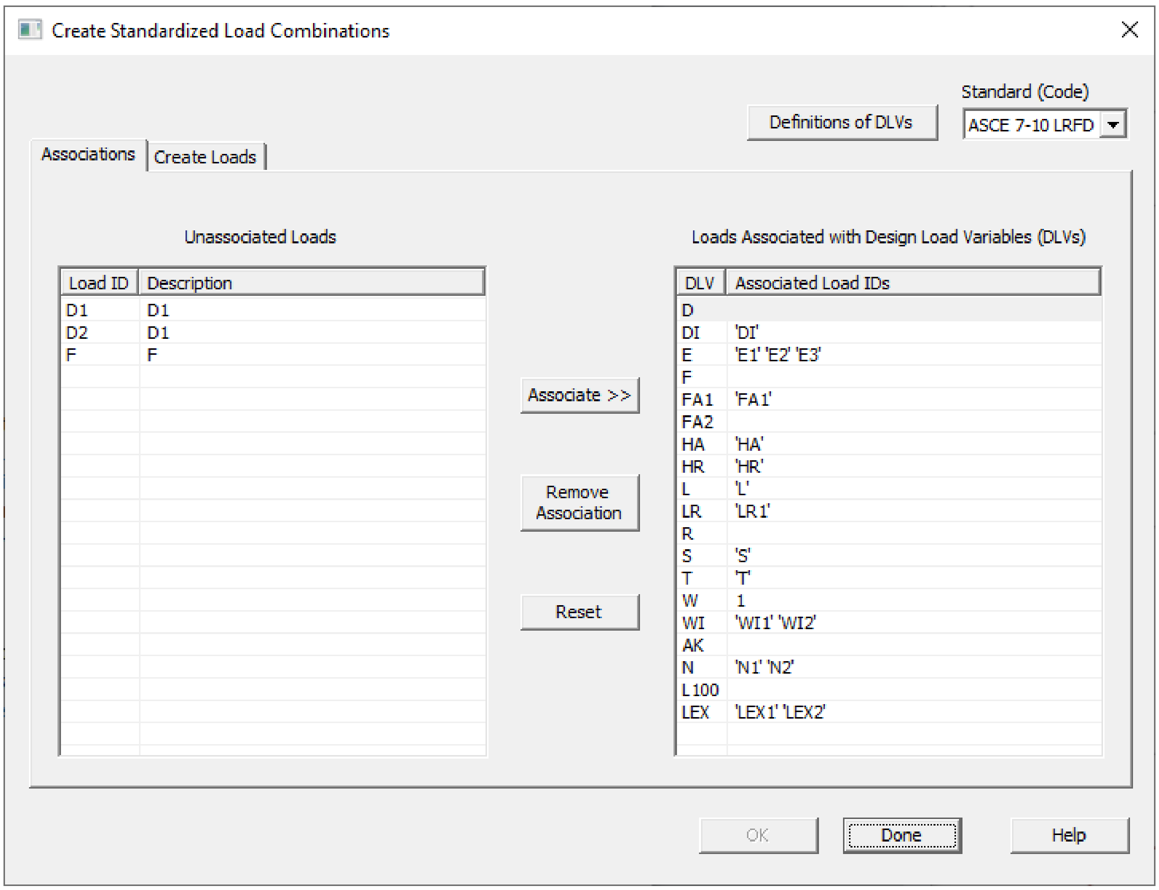


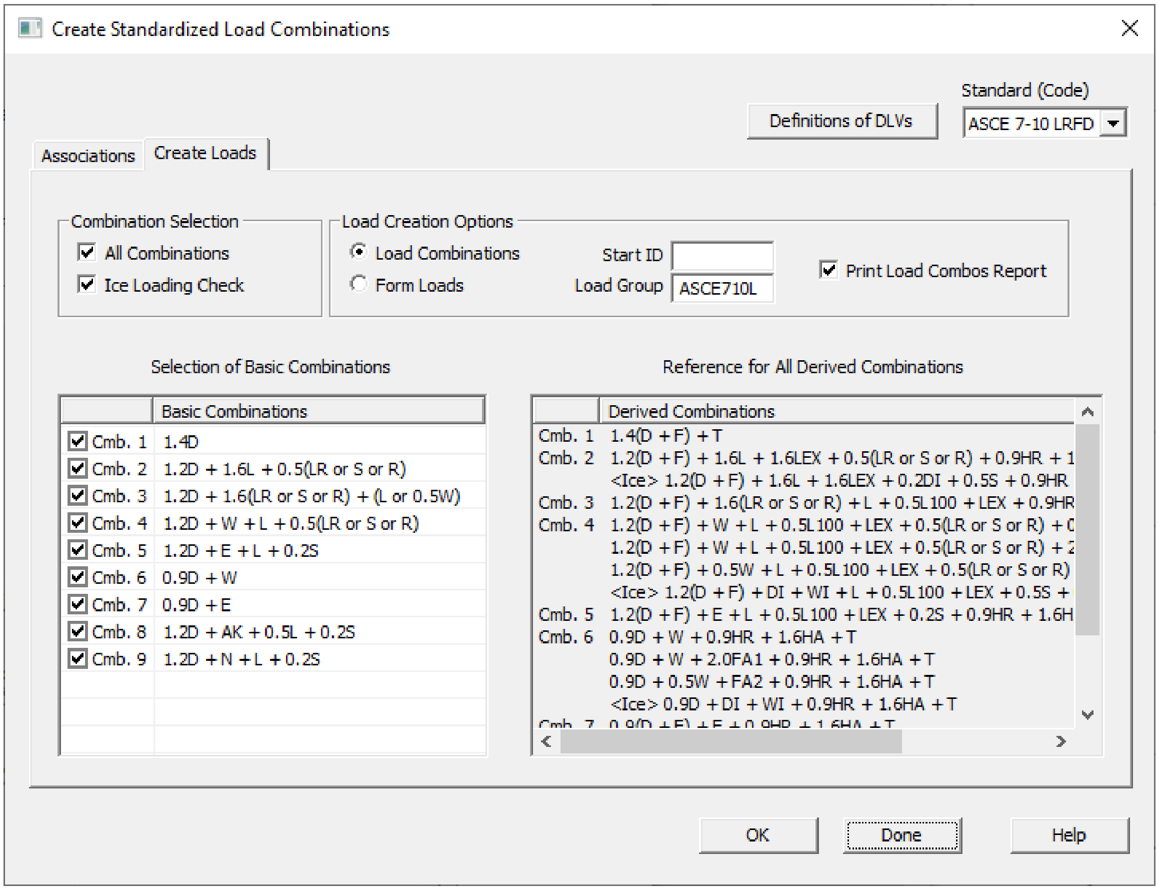
2.6.55. Standardized Combinations

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



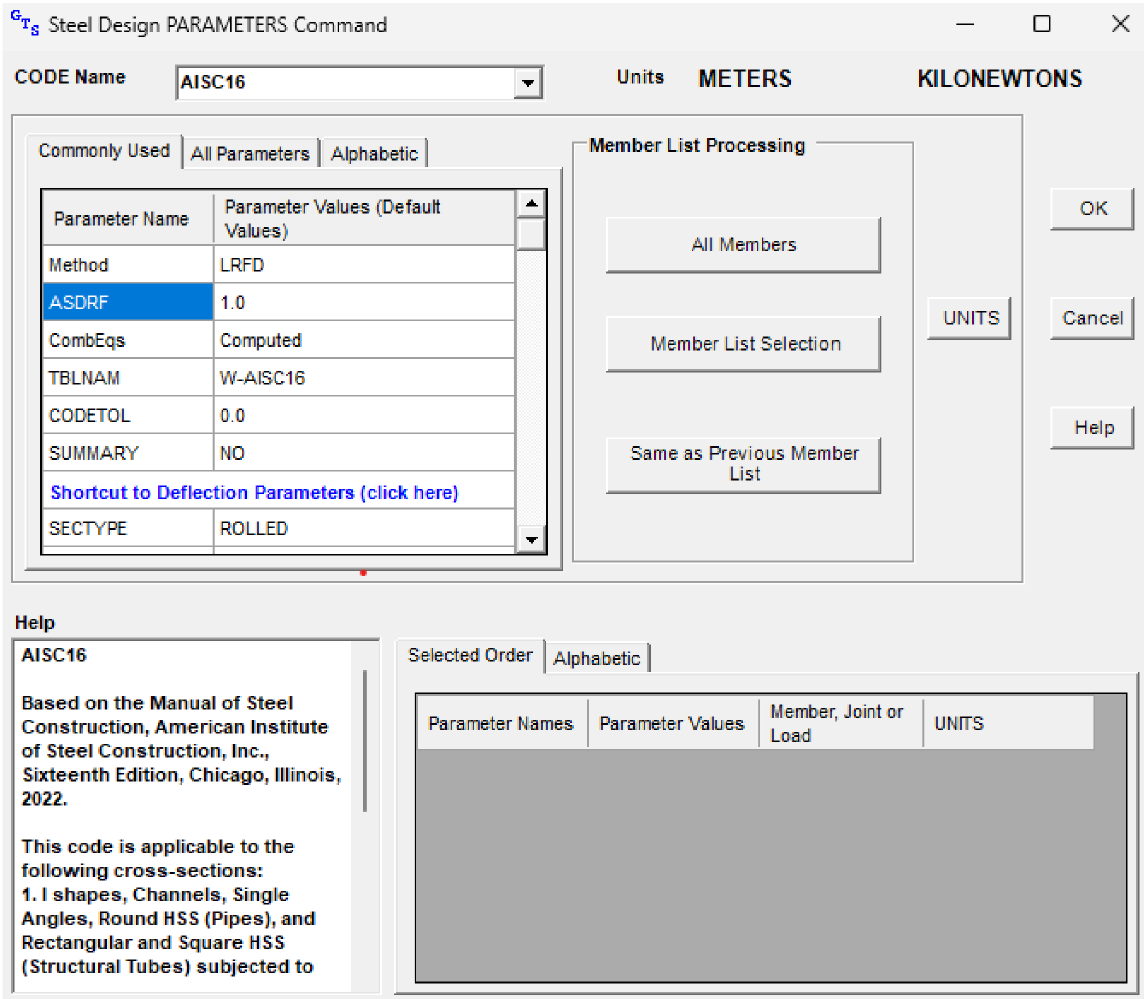
or from the menu *"GTS Modeling>Loads>Standardized Combinations"* or by typing `GTSLoadCombinationStandardized` at the command prompt.





2.6.56. Steel Design Parameters

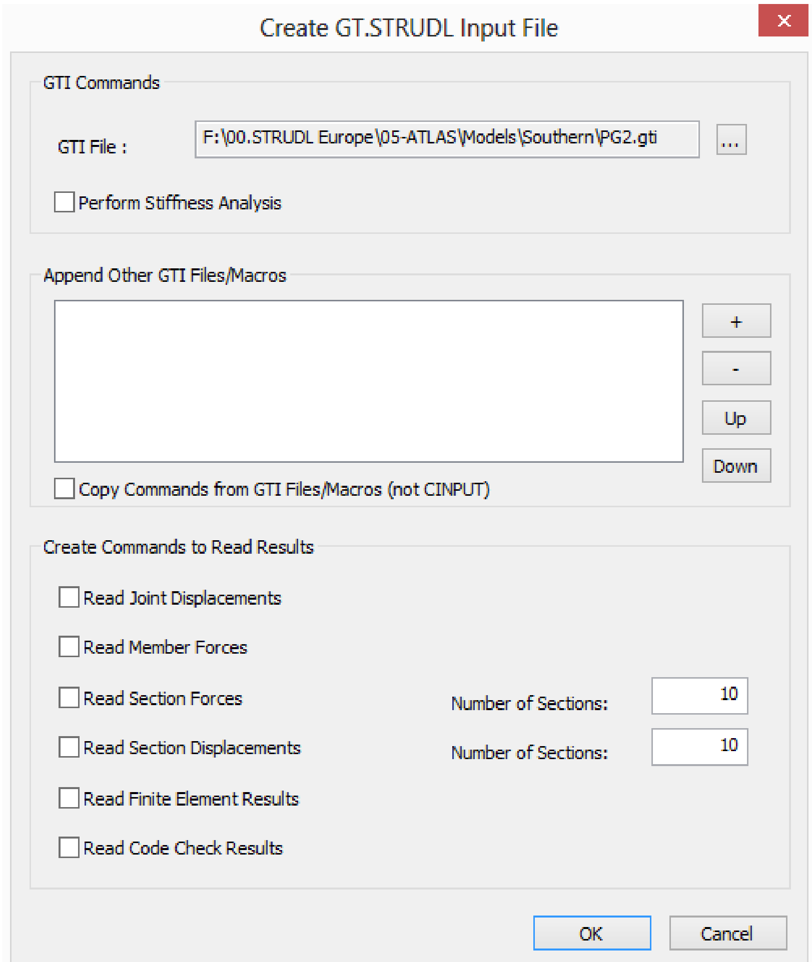
You can specify steel design parameters for AISC16, AISC15, AISC14, EC3, IS800, CSA-2014 and all other codes. Steel design parameters can be defined from the ribbon command Design or menu *"GTS Modeling>Steel Design Parameters"* or by typing `GTSSteelDesignParameters` at the command prompt. Parameters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



2.6.57. Create GTI

A GT STRUDL Text Input file can be generated from the ribbon command 

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.STRUDL GTI*" or by typing `GTSEditGTI` at the command prompt. The previously created GTI is opened for editing using the default text editor.

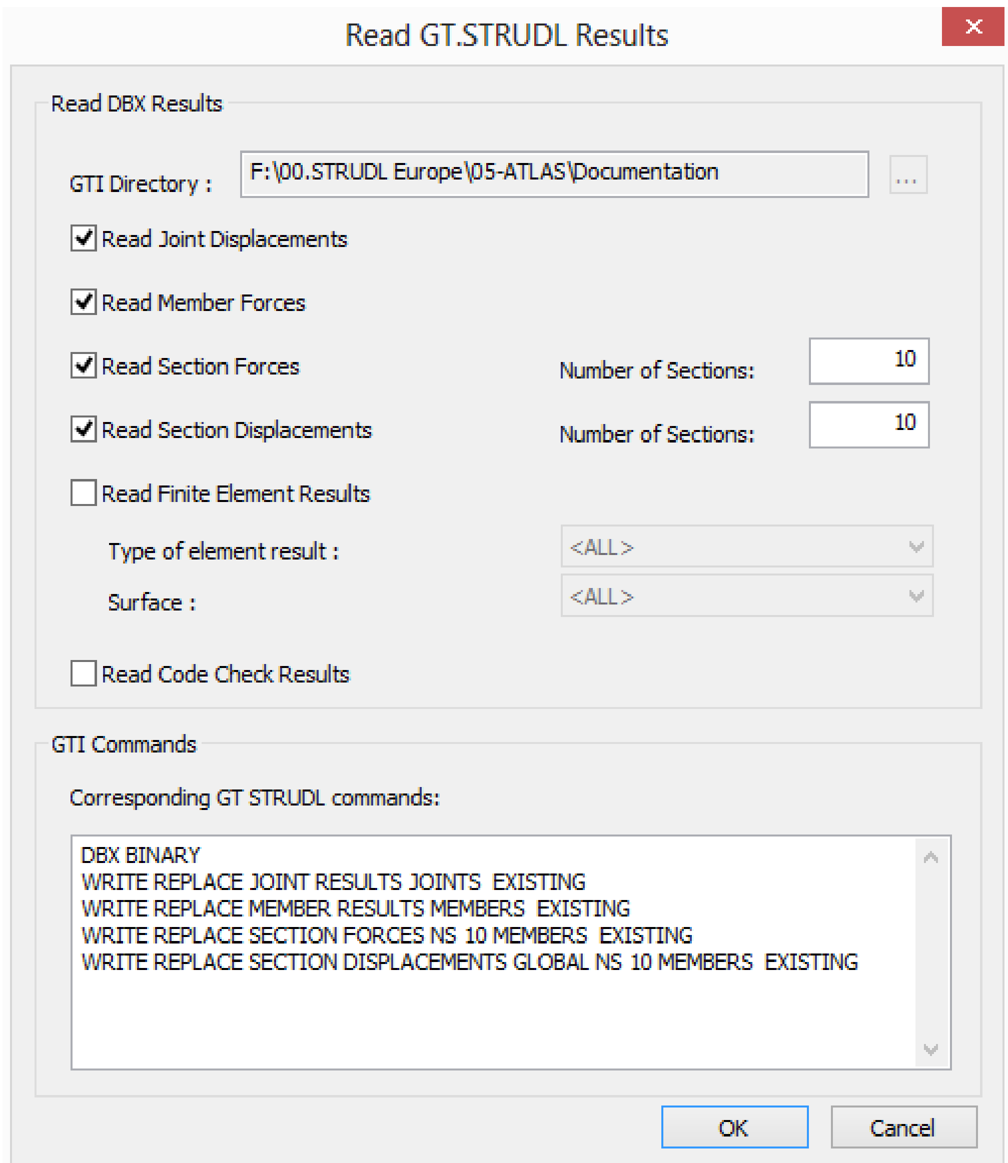
2.6.59. Execute GT STRUDL

GTSTRUDL can be launched, to process the previously created GTI, from the ribbon command GTS or from the menu "*GTS Modeling>Edit GT.STRUDL 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.



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

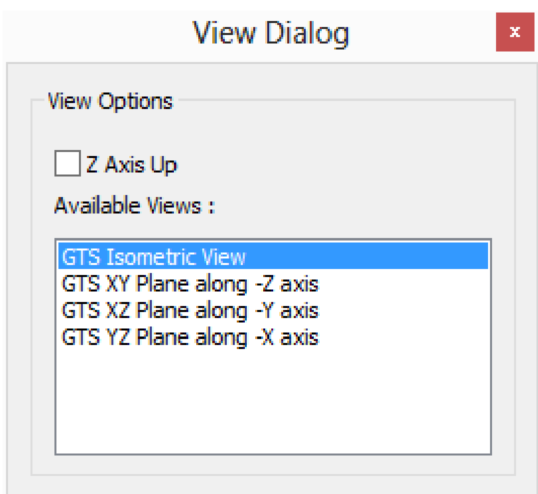
An existing GTI file can be imported it into CAD Modeler from the menu "*GTS Modeling>Import> GT.STRUDL GTI*" or by typing `GTSGTIRead` at the command prompt. Note that the GTI <u>should be generated by the command "File>Save>Text Input File…"</u> from GT STRUDL main menu.

2.6.62. Set Views

You can switch between different 2D or 3D views of the structure from the ribbon command 

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

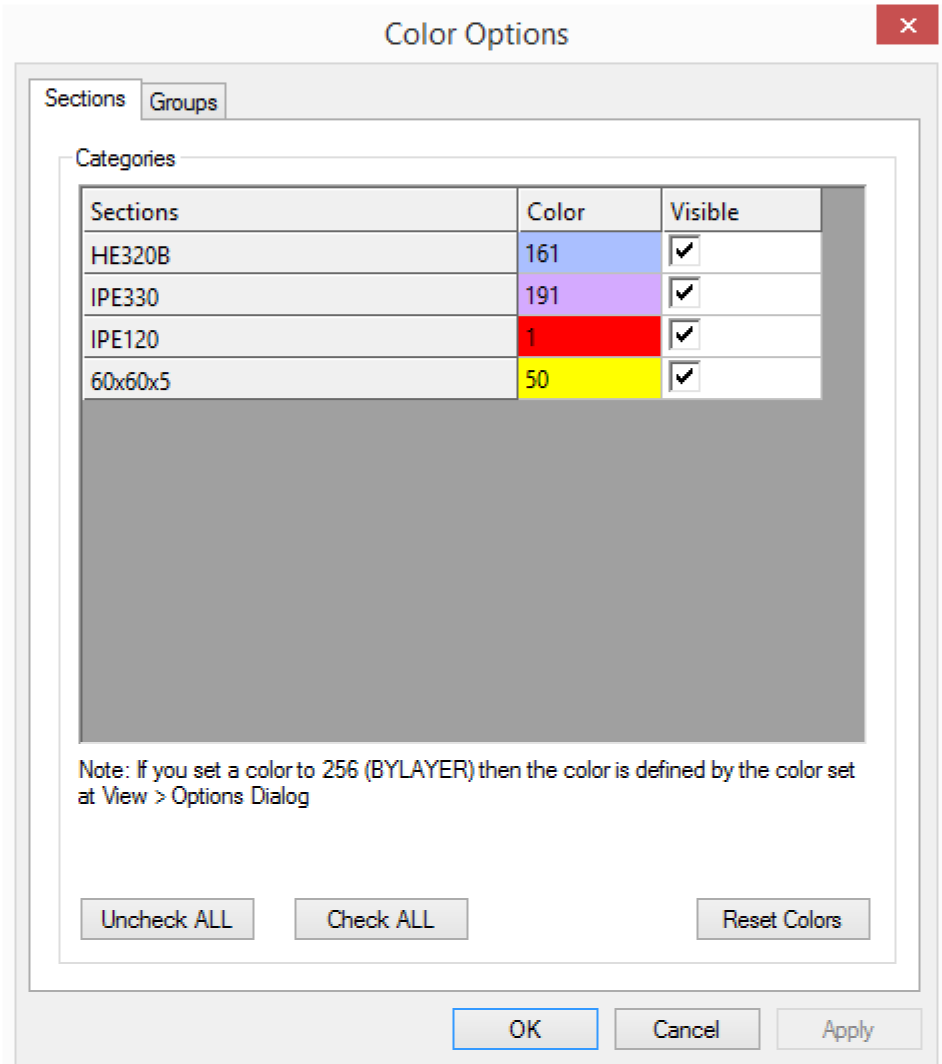
You can turn ON or OFF the physical member view of the structure. When Physical Member view is turned ON, all analytical members belonging to a physical member are hidden and the physical members are displayed instead, as single objects. You can use copy, edit and move commands on the physical member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

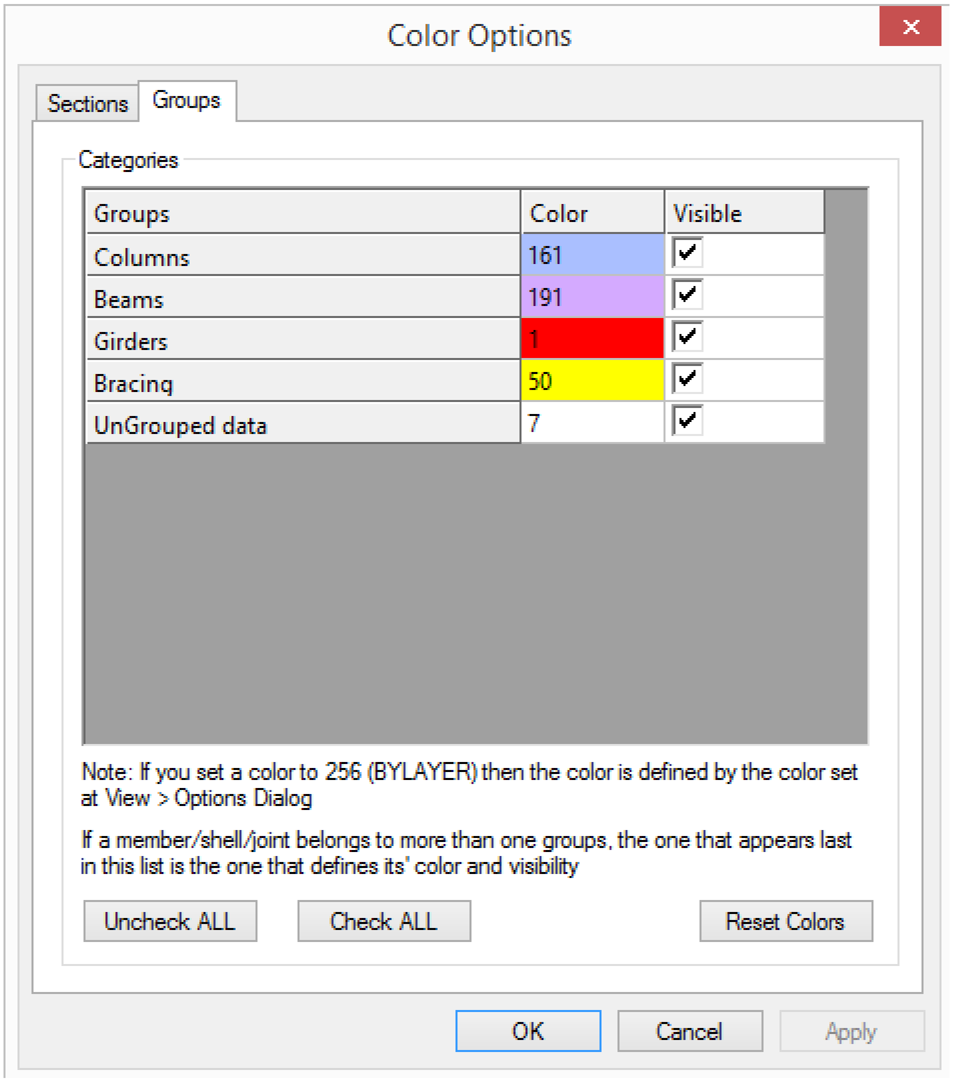
2.6.65. Colors and Visible Elements

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 Disaplay>Options Options, as explained in 2.6.66.



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.



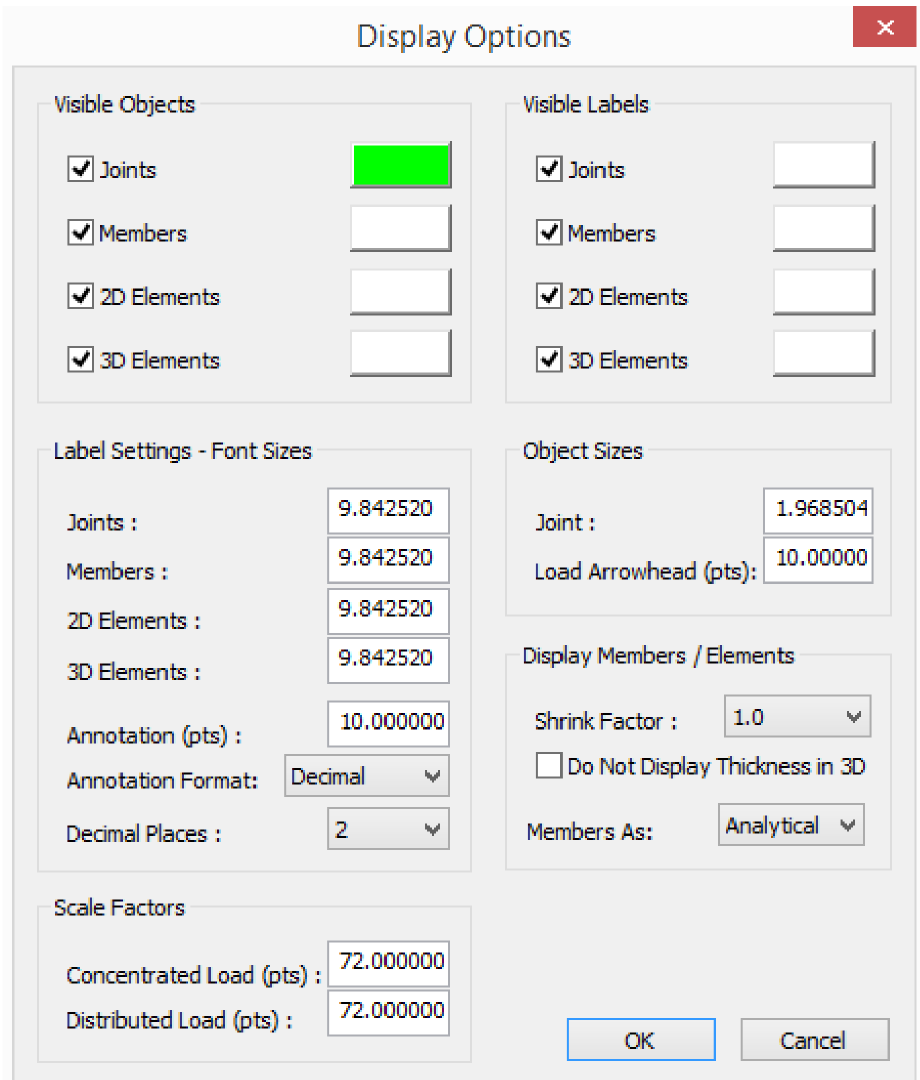
Note, that if the “Sections” tab is active when pressing “OK”, then the colors will be selected according to the “Sections” tab. If the “Groups” tab is active when pressing “OK”, then the colors will be selected according to the “Groups” tab.

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:

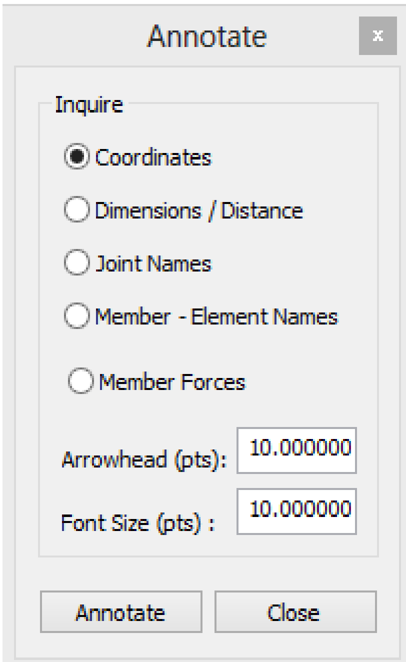
- set which objects will be visible or not
- set object colors
- set which labels will be visible or not
- set font sizes for labels. NOTE: Font sizes are defined in length units, except Annotation fonts that are entered in Points.

- 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)



2.6.67. Annotate

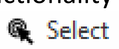
You can display information related to your model from the ribbon command Annotate or by typing `GTSAnnotate` at the command prompt. The “Annotate” form appears where you choose the type information needed, press the “Annotate” button and select the corresponding entities.



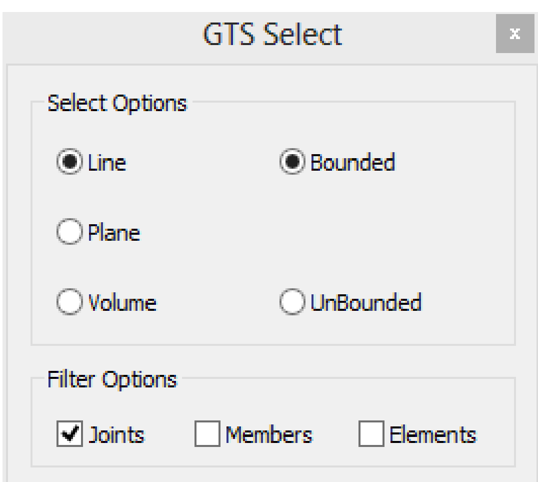
The available inquire options are:

- *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 

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

Volume or its extension

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 

(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 

(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 

(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

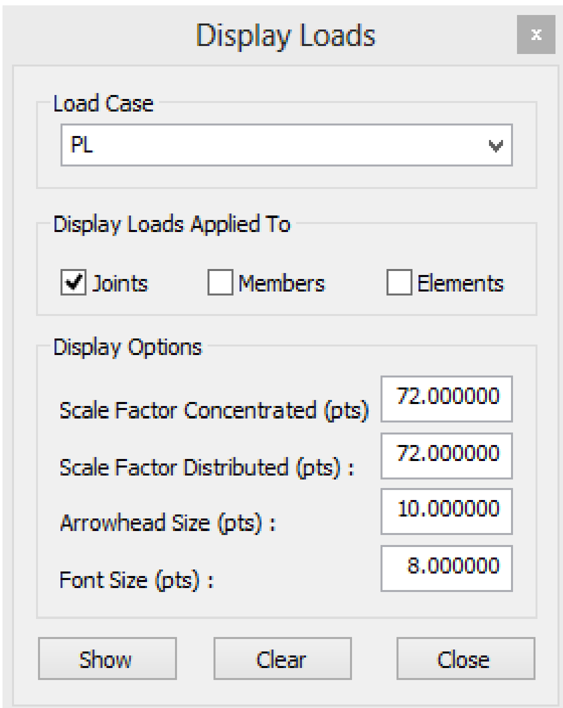
You can view the support status of each joint from the icon 

(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

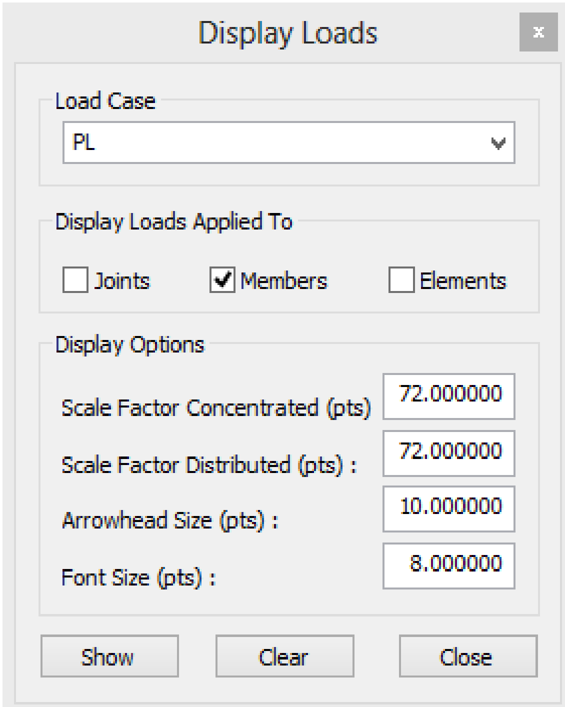
You can view the joint loads applied in the structure from the icon Joint (Ribbon GTS Display) or from the menu *"GTS Display>Joint Loads"* or by typing `GTSDisplayJointLoads` at the command prompt.



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.

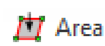
2.6.74. Display Member Loads

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.

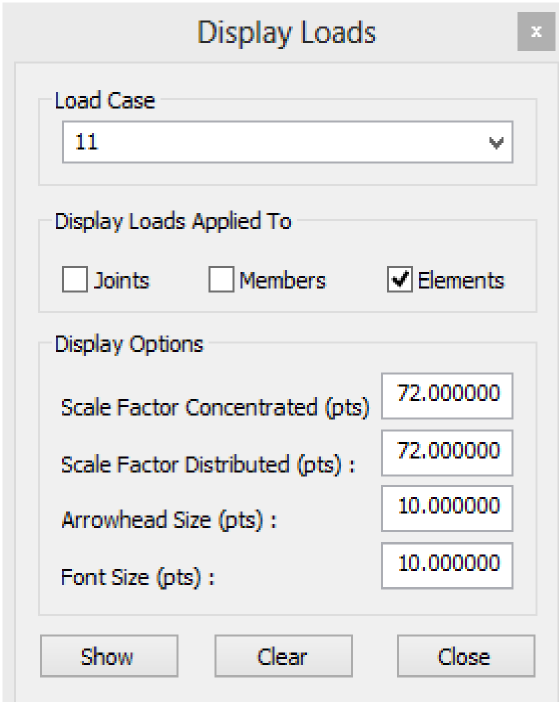


The "Display Loads" form appears where you can select the desired Load Case, the Scale factor for Concentrated or Distributed Member Loads, Arrowhead Size and the *Font Size*. The "Show" button displays the load arrow, and the "Clear" button erases them.

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.



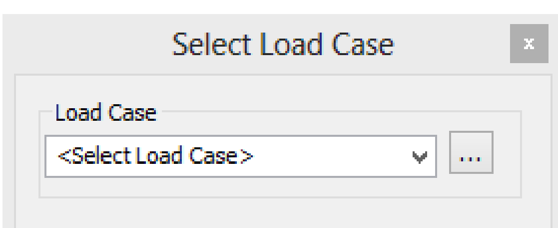
The “Display Loads” form appears where you can select the desired Load Case, the Scale factor for Concentrated or Distributed Member Loads, Arrowhead Size and the *Font Size*. The “Show” button displays the load arrow, and the “Clear” button erases them.

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 Lever, Lower Level icons.

2.6.78. Annotate Joint Displacements

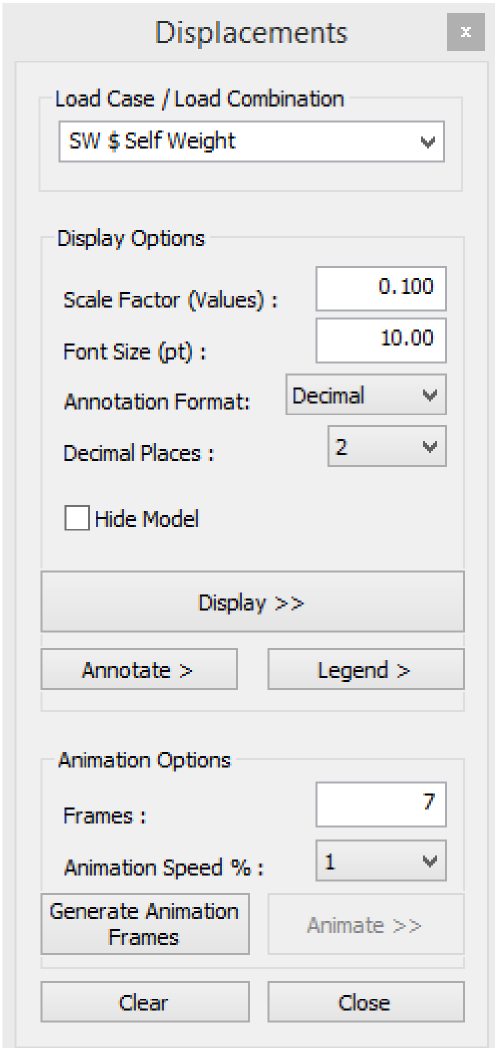
At the deformed state, you can annotate joint displacements of the structure from the icon 

(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 

(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:

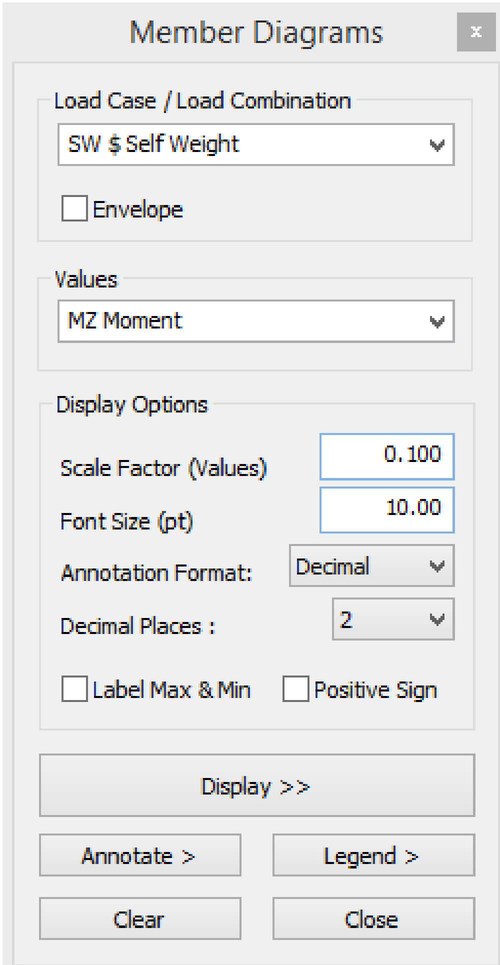
- The desired Load Case or Combination
- The Scale factor
- The *Font Size* (in pts) for Annotations and the Annotation Format.
- Choose to display the model or Hide it, so that the deformed shape is clearer.
- The "*Display >>*" button displays the deformed shape for the visible members and elements. 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 fist 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 

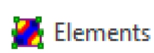
(Ribbon GTS Display) or from the menu "*GTS Display>Member Diagrams*" or by typing `GTSDisplayMemberForces` at the command prompt.



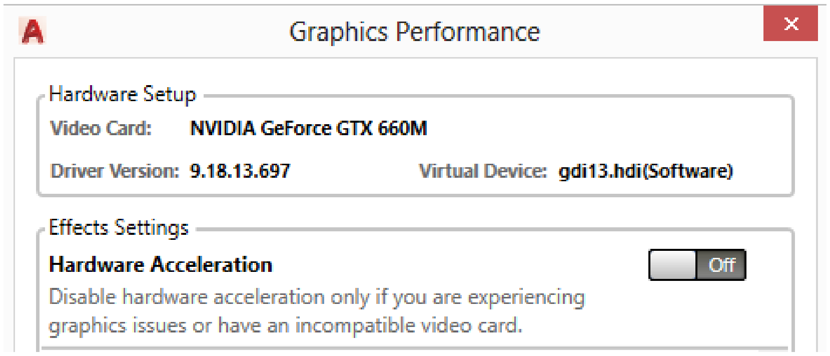
The "Member Diagrams" form appears where you can select:

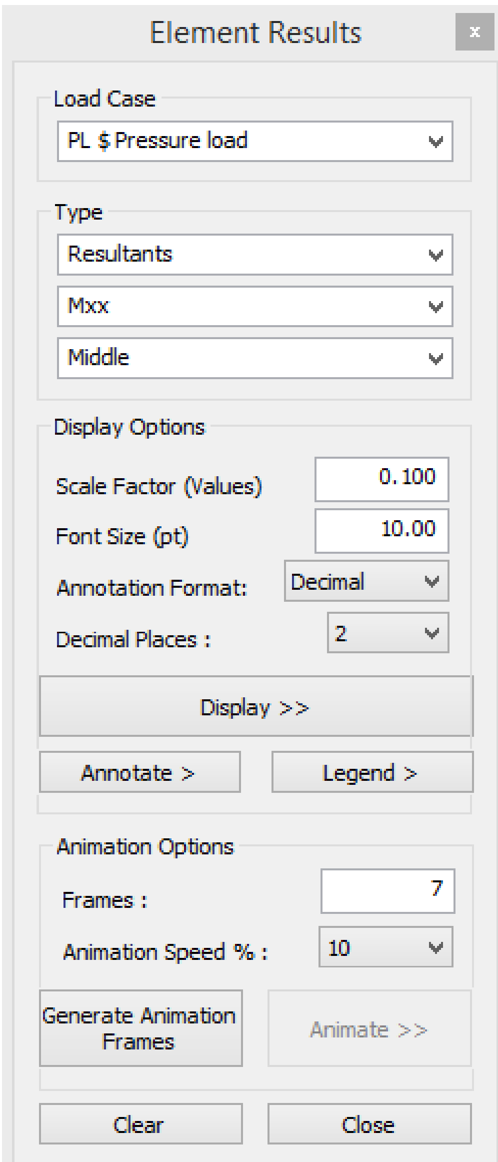
- The desired Load Case or Combination
- The Envelope option and the load cases that form the envelope.
- The Forces or Moments to be displayed (FX, FY, FZ, MX, MY, MZ)
- The Scale factor
- The *Font Size* (in pts) for Annotations and the Annotation Format.
- Automatically *Label Maximum and Minimum* 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  (Ribbon GTS Display) or from the menu "*GTS Display>Element Results*" or by typing `GTSDisplayElementResults` at the command prompt.

NOTE: Hardware graphics accelaration may cause AutoCAD to incorrectly display the colors of the contour. In such a case it is recomemded that you turn OFF Hardware Accelartion, 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 fist 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

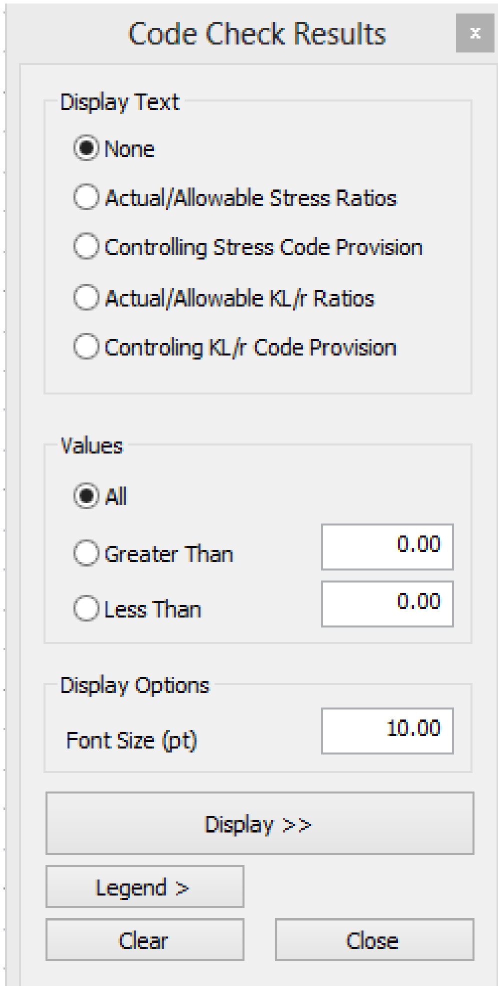
You can view the finite element results of selected elements from the icon Selection (Ribbon GTS Display) or from the menu "*GTS Display>Element Results Selection*" or by typing `GTSDisplayElementResultsSel` at the command prompt.

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.

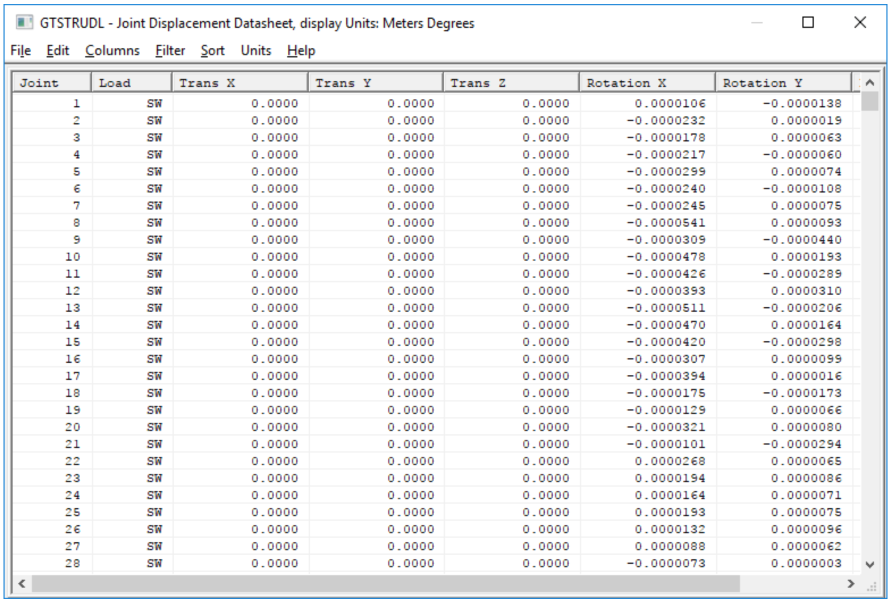
When you press the clear button, member colors remain blue and red for your convenience. If you want to change them go to GTS Display > Colors.

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	*GTS Display>Results Datasheets> Displacements*	`GTSDataSheetJointDisp`
Member Forces	*GTS Display>Results Datasheets> Member Forces*	`GTSDataSheetMemberForces`
Section Forces	*GTS Display>Results Datasheets> Section Forces*	`GTSDataSheetMemberForces`
Reactions	*GTS Display>Results Datasheets> Reactions*	`GTSDataSheetReactions`
Stresses	*GTS Display>Results Datasheets> Stresses*	`GTSDataSheetStresses`
Code Check	*GTS Display>Results Datasheets> Code Check*	`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:


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

2.6.85. Report Builder

You can generate your reports by calling Report Builder from the icon Report Builder (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.

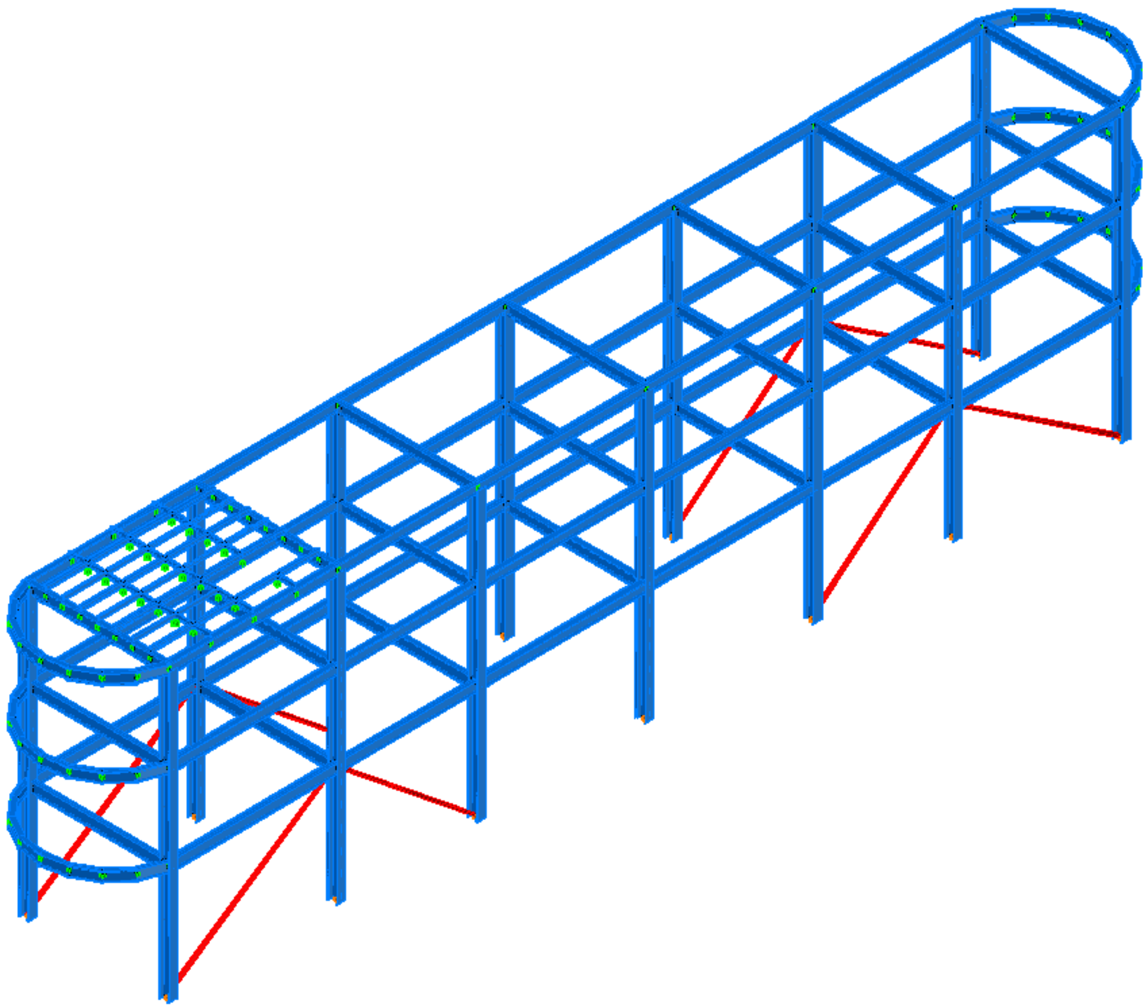
2.6.87. Version

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 xxxx`

3. Tutorial Example #1

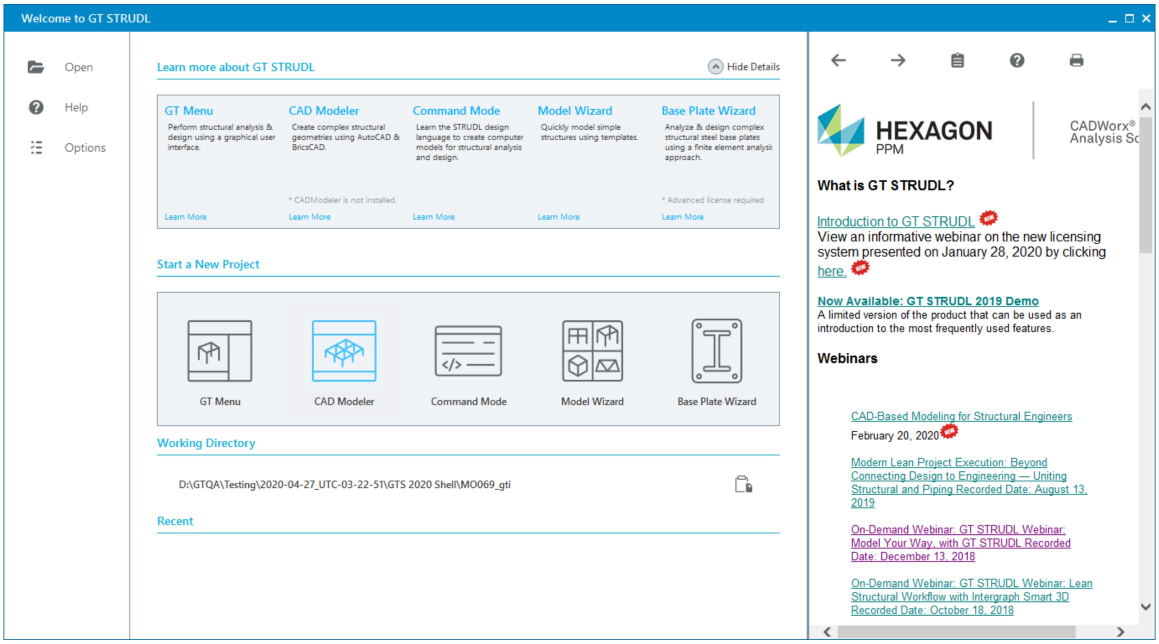
3.1. Introduction

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

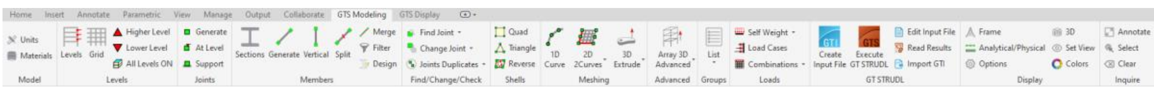


3.2. Open CAD Modeler and start working

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.



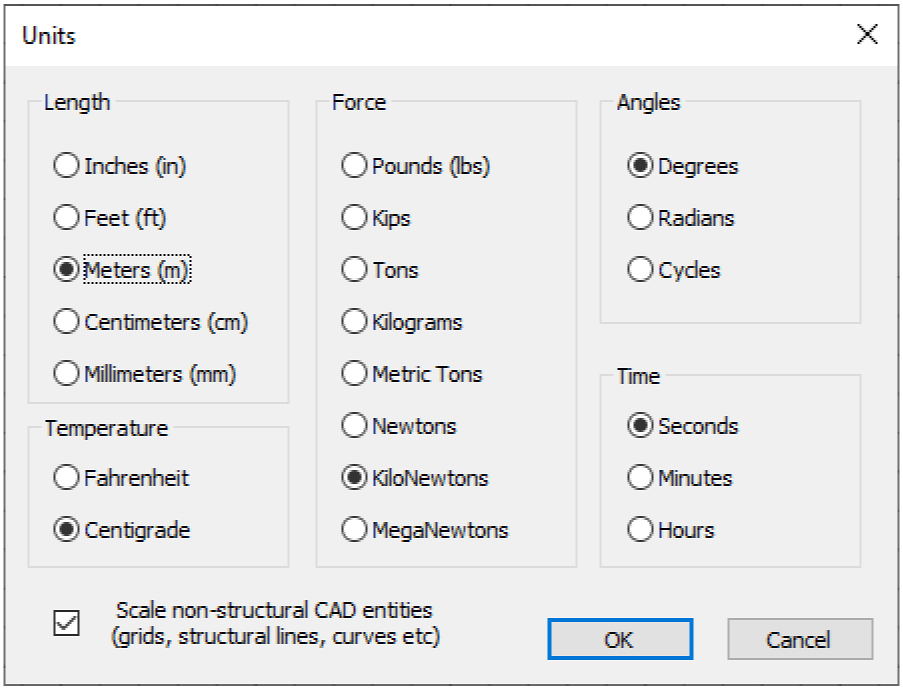
If AutoCAD's/BricsCAD's menu is not visible, type `MENUBAR` at AutoCAD's/BricsCAD's command prompt, then `1` and press `<ENTER>`.

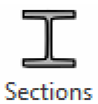
If AutoCAD's/BricsCAD's ribbon area is not visible, type `RIBBON` and press `<ENTER>`.

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.

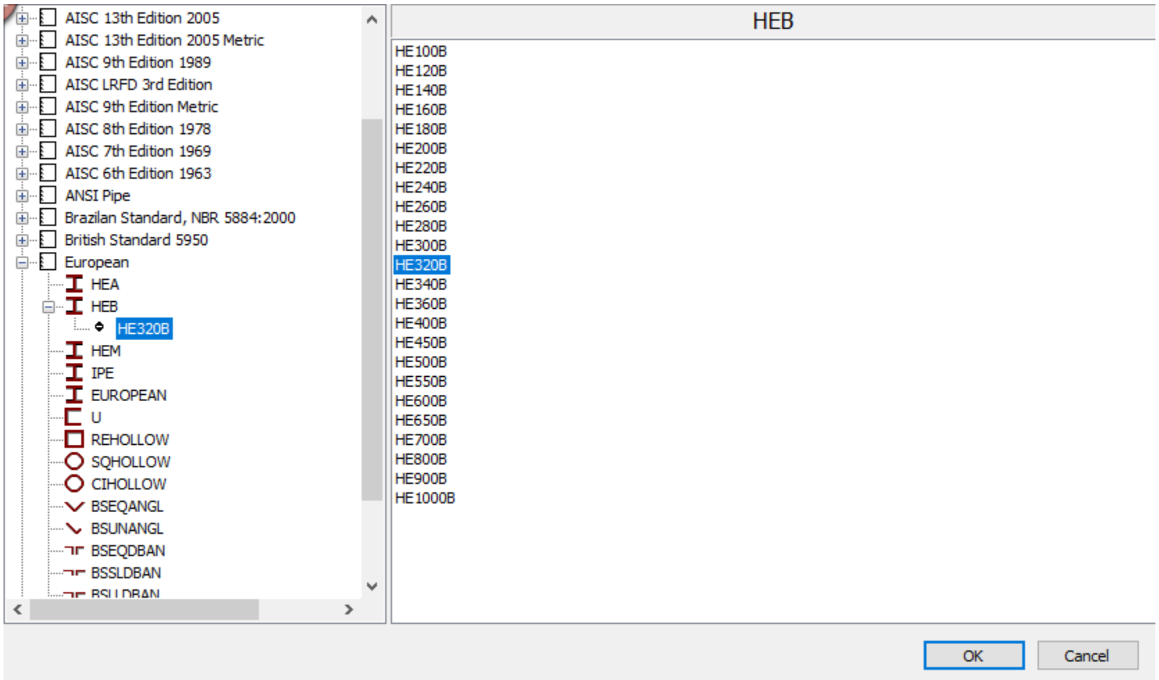
3.3. Define the basic geometry of the model

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



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

. 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.



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.

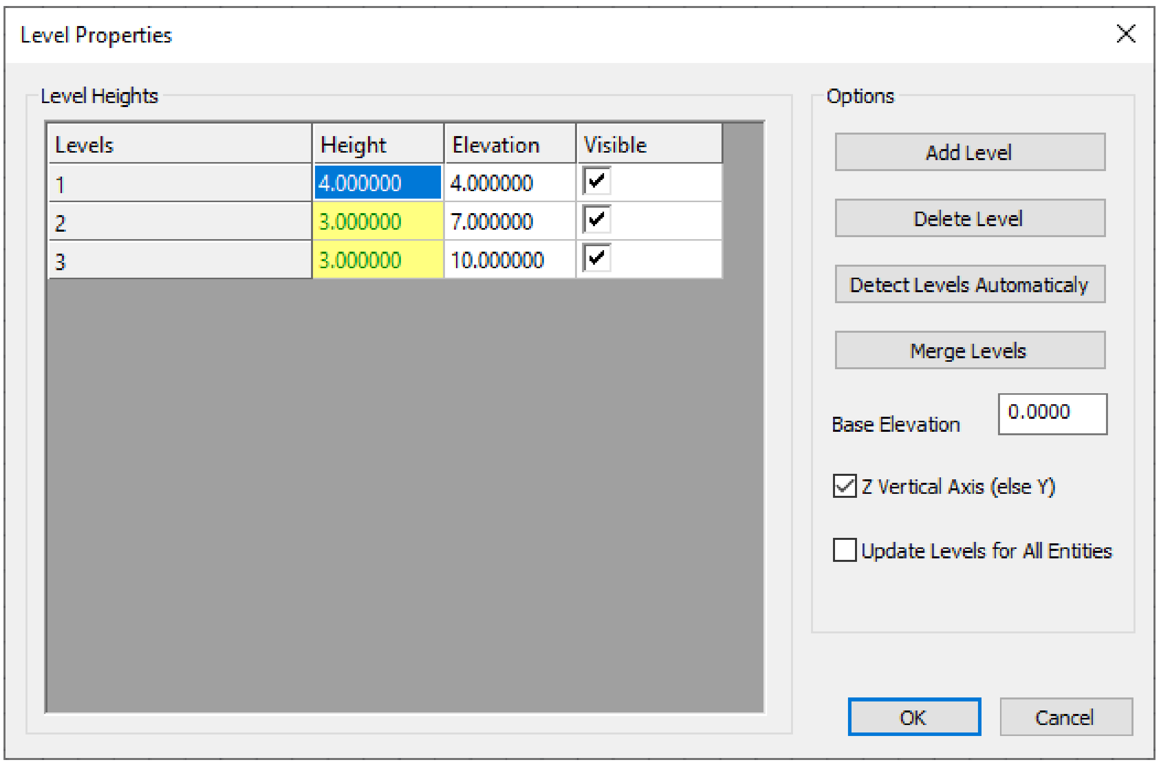
Note: You can add additional profiles at any time by following this procedure and also view the full list of profiles used in your model and add more profiles if needed.

Step #5. Define the 3 levels of the model by pressing the icon 

. 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`.

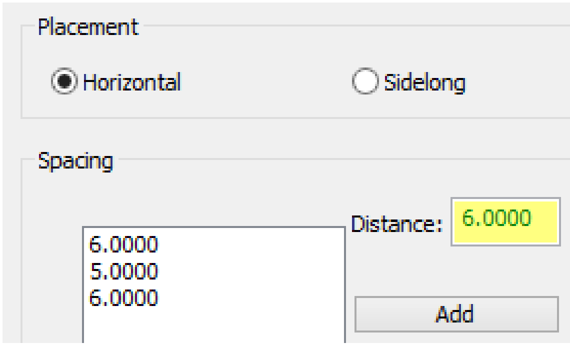
Note: Some Edit Boxes appear in yellow background and green fonts, like the one at the Level Heights. You can use mixed units in the yellow edit boxes. For more information about Mixed Units and the valid syntax please read GT STRUDL GT Menu Guide

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



Step #6. Enter a Grid that will help you enter the columns quickly by clicking on the icon 

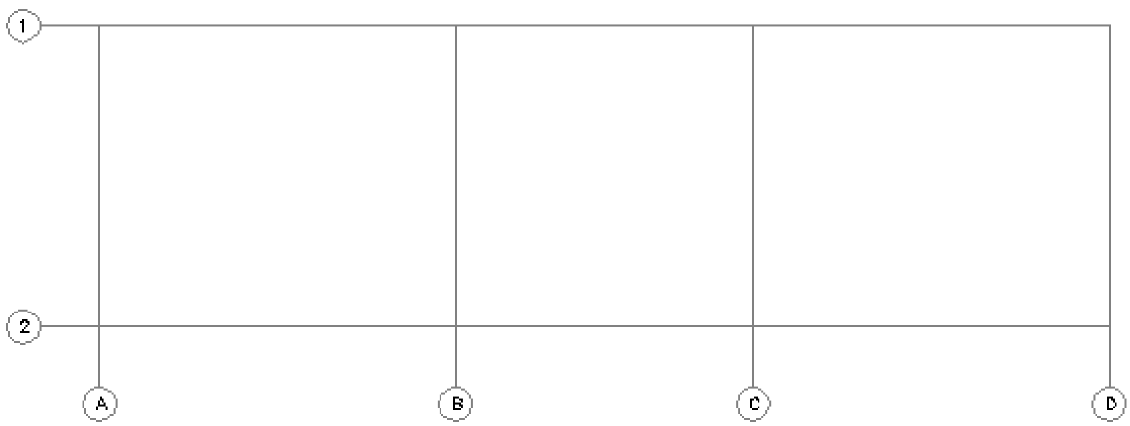
. The grid is going to have 3 spaces (6m, 5m, 6m) in the horizontal direction (X) and 1 space (5m) in the sidelong direction (Y). Enter `6` in the *Distance* text box and press the *Add* button. Repeat by entering `5` and *Add*, and `6` and *Add*.



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

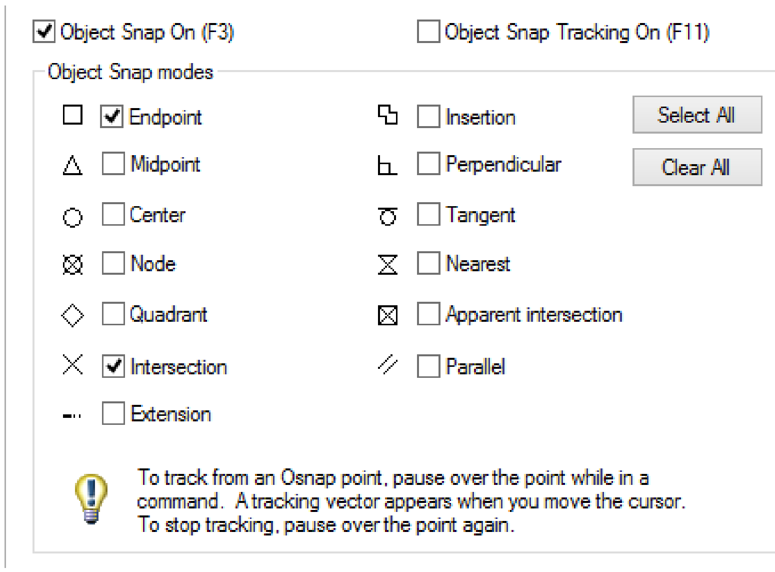
By pressing OK you are prompted (command prompt) to enter the Insert Point for the grid. Type 0,0,0 and press <ENTER>.

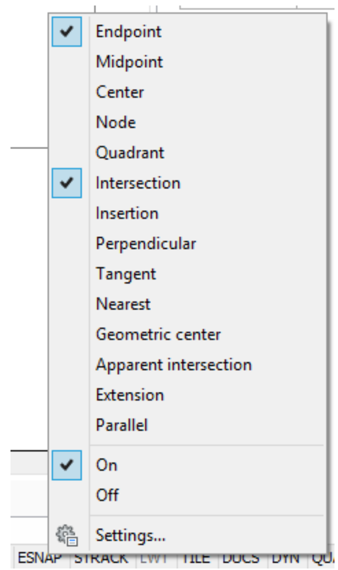
The grid is created, having its upper left corner A-1 at the point 0,0,0.



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>.

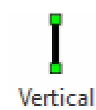
Note: In order to be able to snap at the intersection of the grid lines, while placing columns, make sure that the AutoCAD's/BricsCAD's Object Snap is ON, and the Intersection mode is enabled. Type OSNAP in both AutoCAD and BricsCAD to set the various snap settings. Shown below is the ObjectSnap tab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



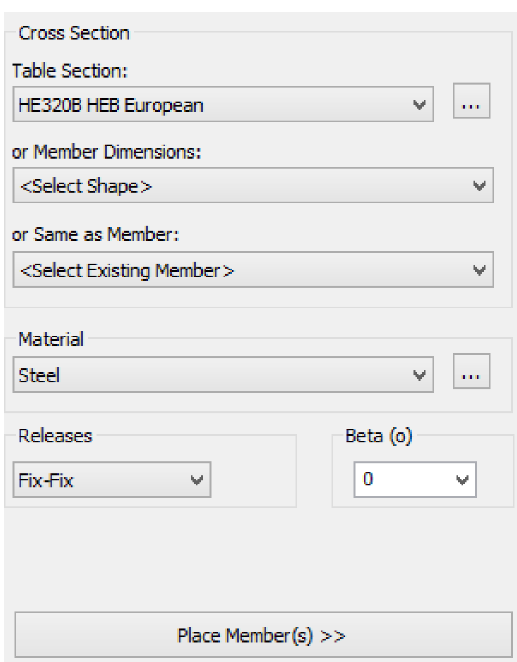


AutoCAD's Snap Setting BricsCAD's Snap Settings

3.4. Create the 1st floor

Step #7. Start entering the columns by clicking on the icon 

. The dialog *Place Member* appears that helps you to quickly select properties for the members that are going to be entered.



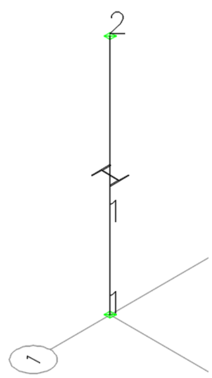
Select *HEB320B* as the cross section for the columns. Make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *0*. There is no need to close this dialog manually.

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.

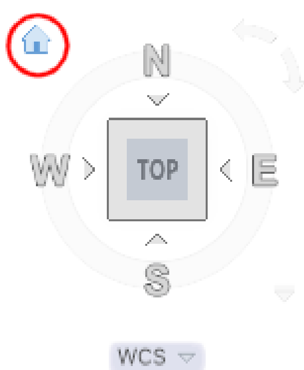
Repeat the same procedure by clicking at the points B-1, C-1, D-1, A-2, B-2, C-2 and D-2.

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

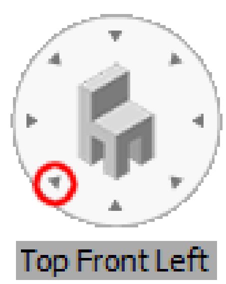


Note: Each time you create a member the orientation of the cross section will appear in the middle of the element, unless you clear it with command "Clear" (see 2.6.86).

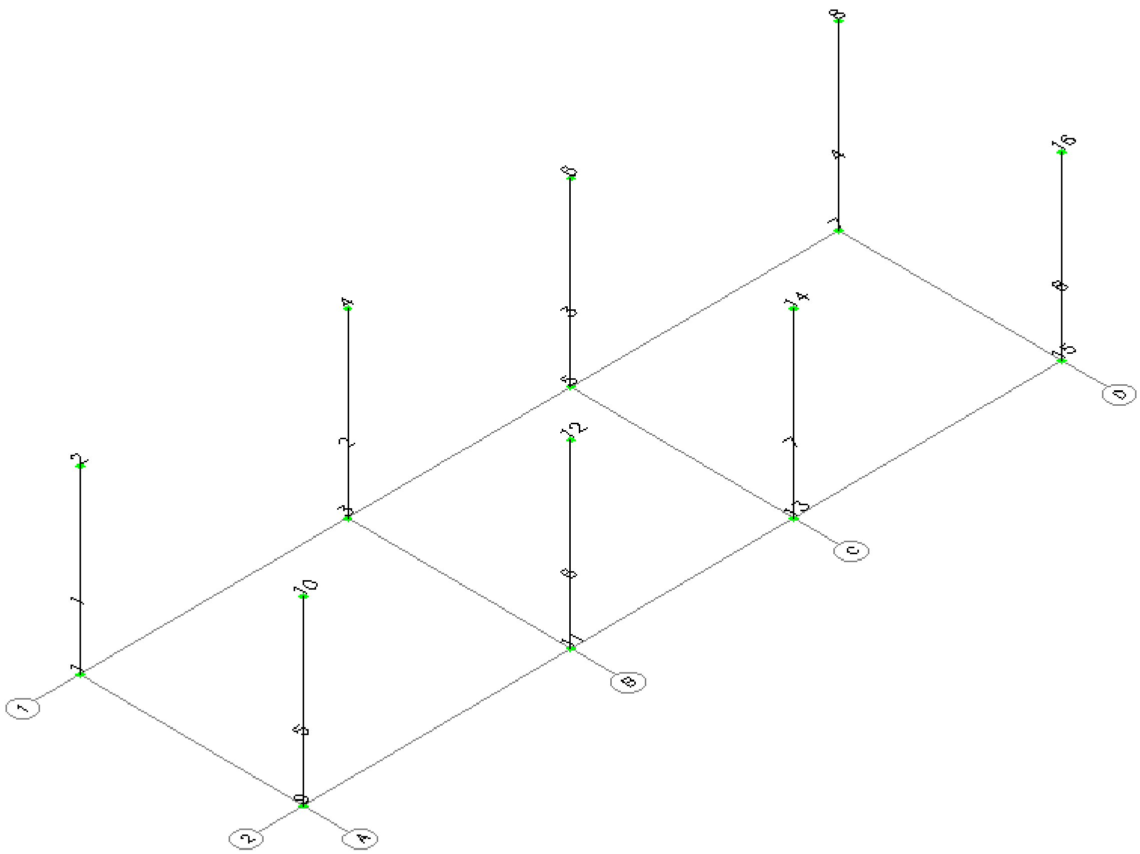
Step #8. You can easily change to an isometric view of the structure by pressing the small house icon in AutoCAD's View Cube or the small arrow "Top Front Left" of the following image in BricsCAD's View Chair. As you can see in the isometric view below, column members 1 to 8 were created together with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



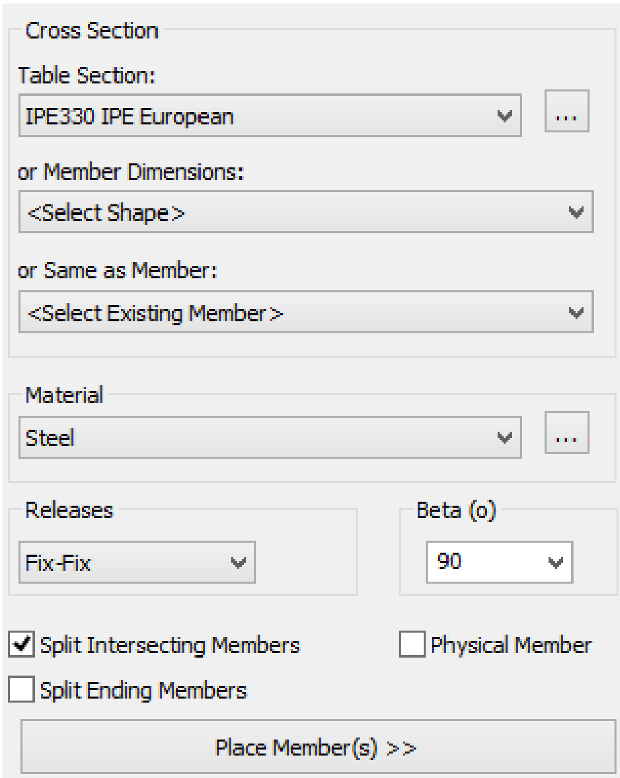
AutoCAD's View Cube for Isometric View



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.

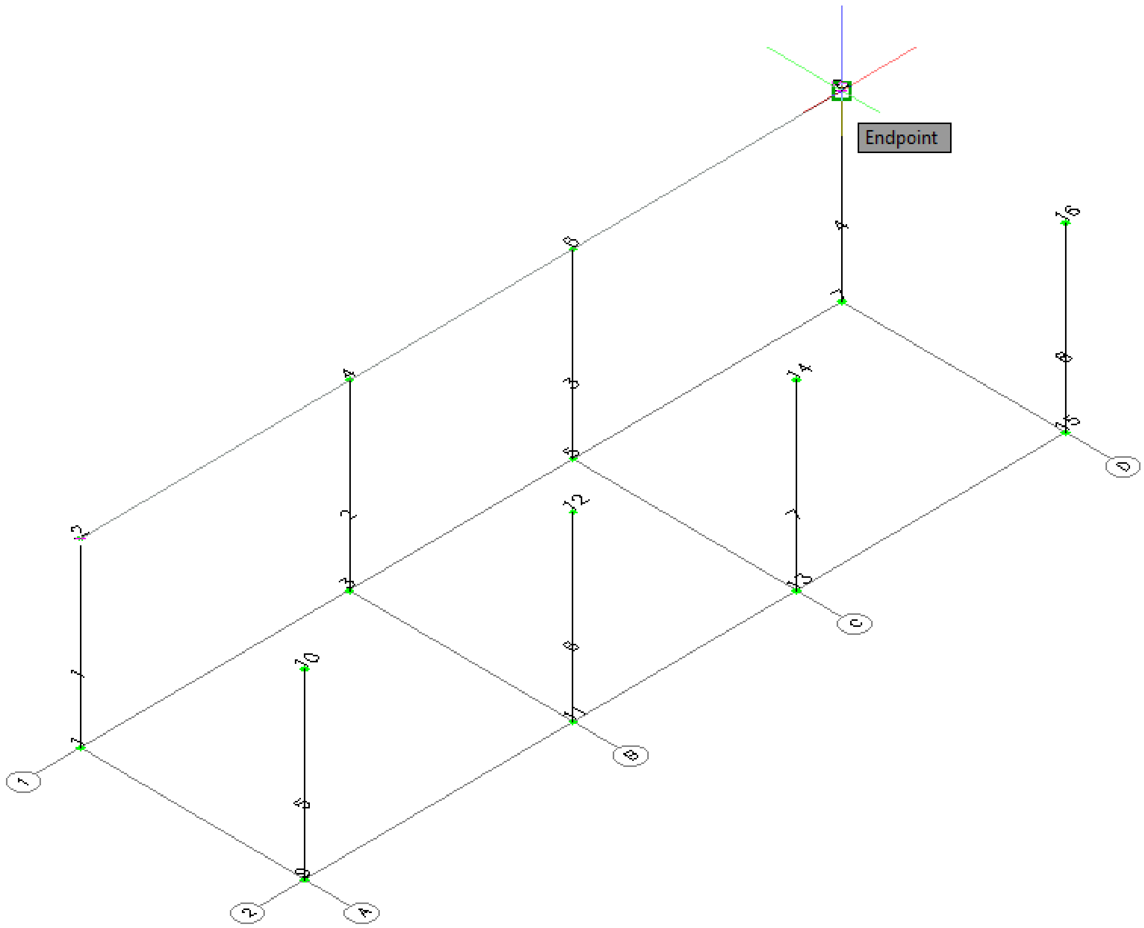


Select *IPE330* as the cross section and make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *90*. You have to set the Beta angle equal to 90 degrees in order to orientate the local Y axis of the IPE cross section along the Z global axis. Moreover, make sure that the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

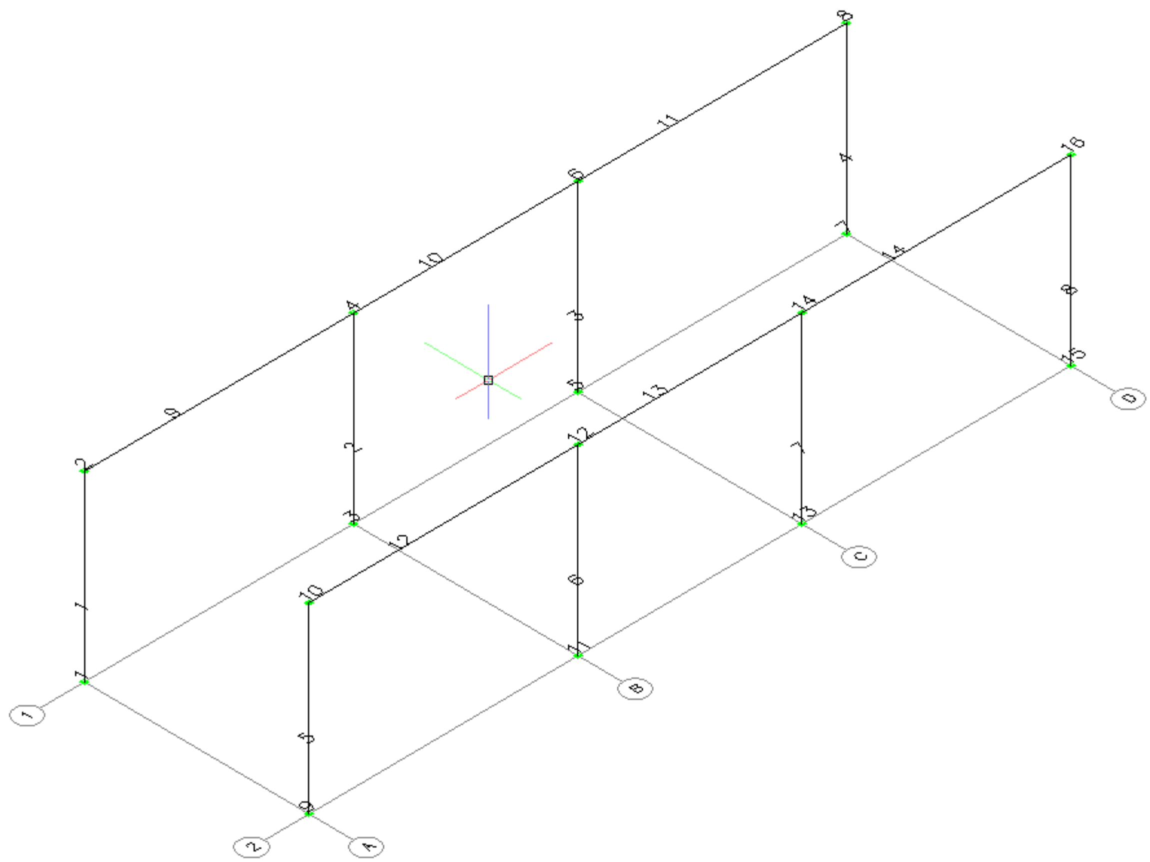
Then you have to define the *Ending Point (x,y,z)* so click on joint 8, as shown in the picture below.



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.

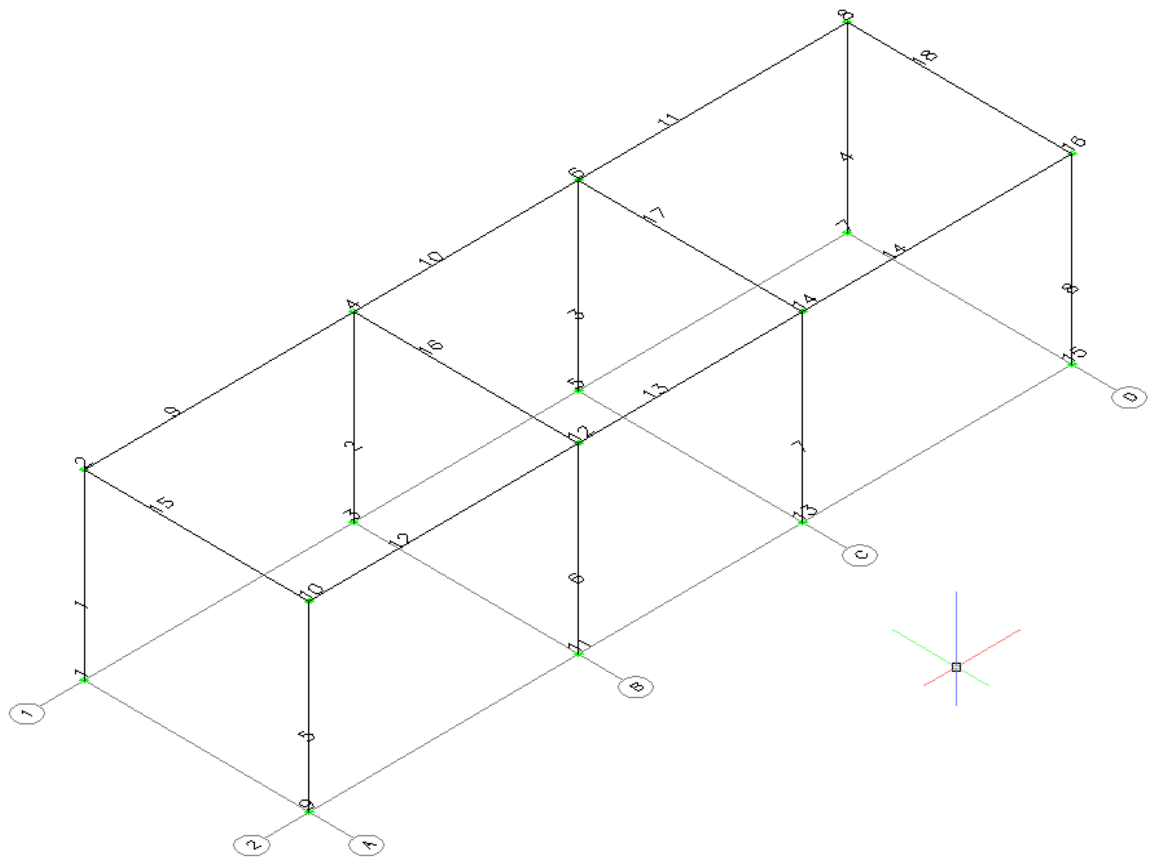


Step #10. Enter the beams, along Y axis. The command Generate Beams should be still active, else you can call it again by clicking again on the icon 

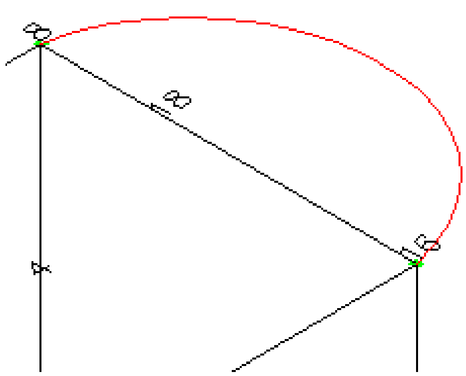
. Keep the same settings at the Place Member Form, as in the previous step, regarding the cross section and Beta angle, but do NOT click on *Split Intersecting Members*. Press the “Place Member(s) >>” button.

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.

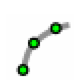


Step #11. Create an arc on the right side of the structure:

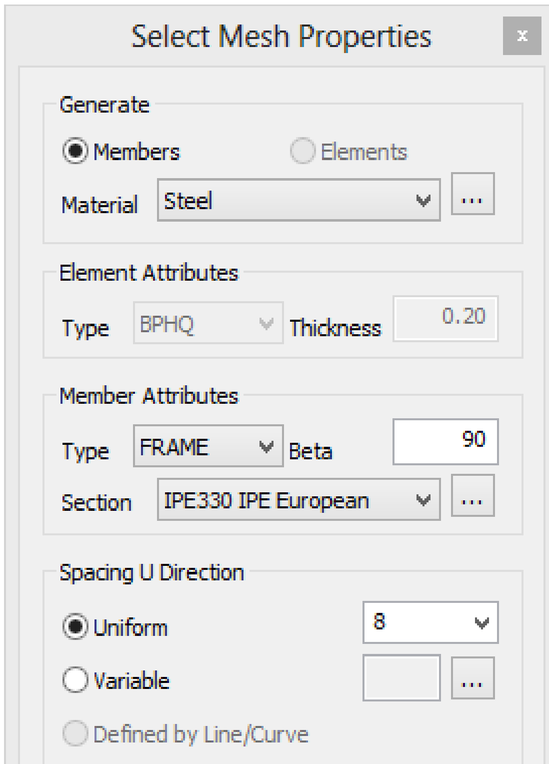


Type `ARC` and

- 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 

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 form appears where you enter:

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

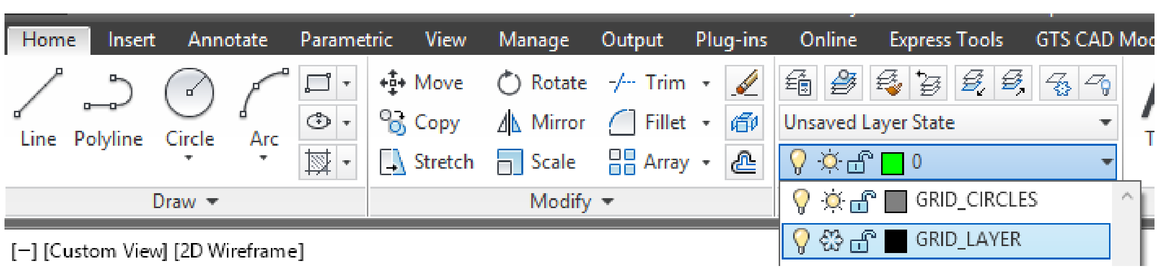
This command is going to generate 8 linear members equally spaced along the arc.

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

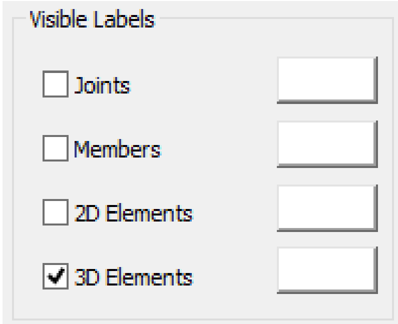
Press the Create button and 8 members, named 19 to 26, were created and 9 joints, named 17 to 25 were created along the arc.

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 

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.

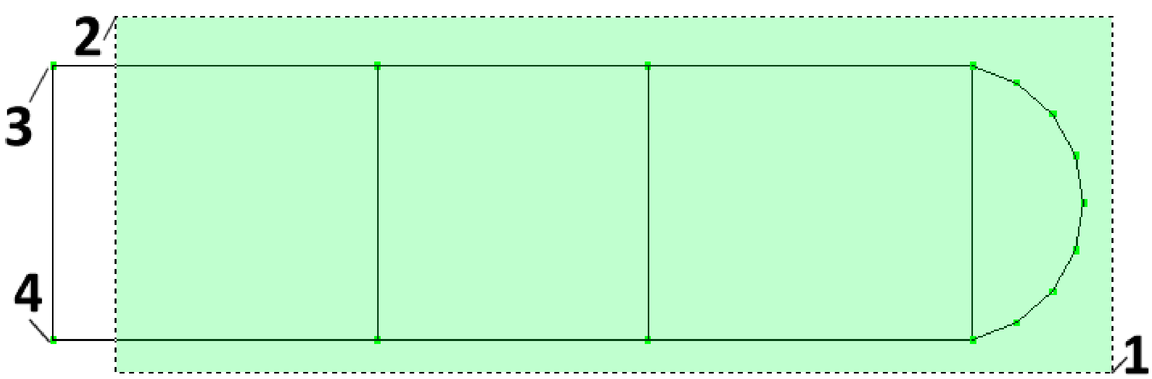
Note: You can also delete or hide the Arc line as it is no longer needed. It can be deleted by selecting it with the mouse, and then presssing the keyboard button <Delete>. Be carefull not to select any members or joints, but only the Arc line.

Step #15. Mirror the structure: Switch to a floor pan 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.

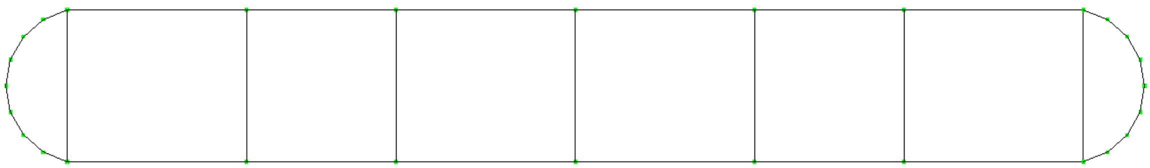
NOTE: This option is very useful when mirroring structures having shell finite elements and the mirror line lies along the element's XY plane. If you choose to maintain the incidence order, then after the mirroring the local Z axis of the source and copied finite elements have the same orientation.

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).



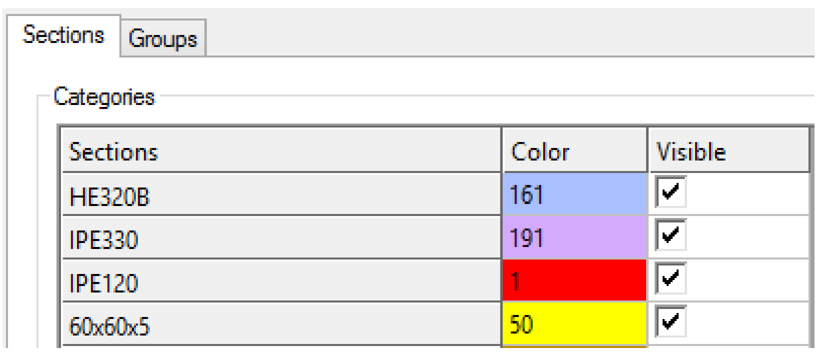
When you get the message *Specify first point of mirror line* (AutoCAD) or *Start of mirror line* (BricsCAD): click on the joint at points 3 and then *Specify second point of mirror line* (AutoCAD) or *End of mirror line* (BricsCAD): click on the joint at point 4 as shown in the picture above.

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 

to set different colors for each profile.



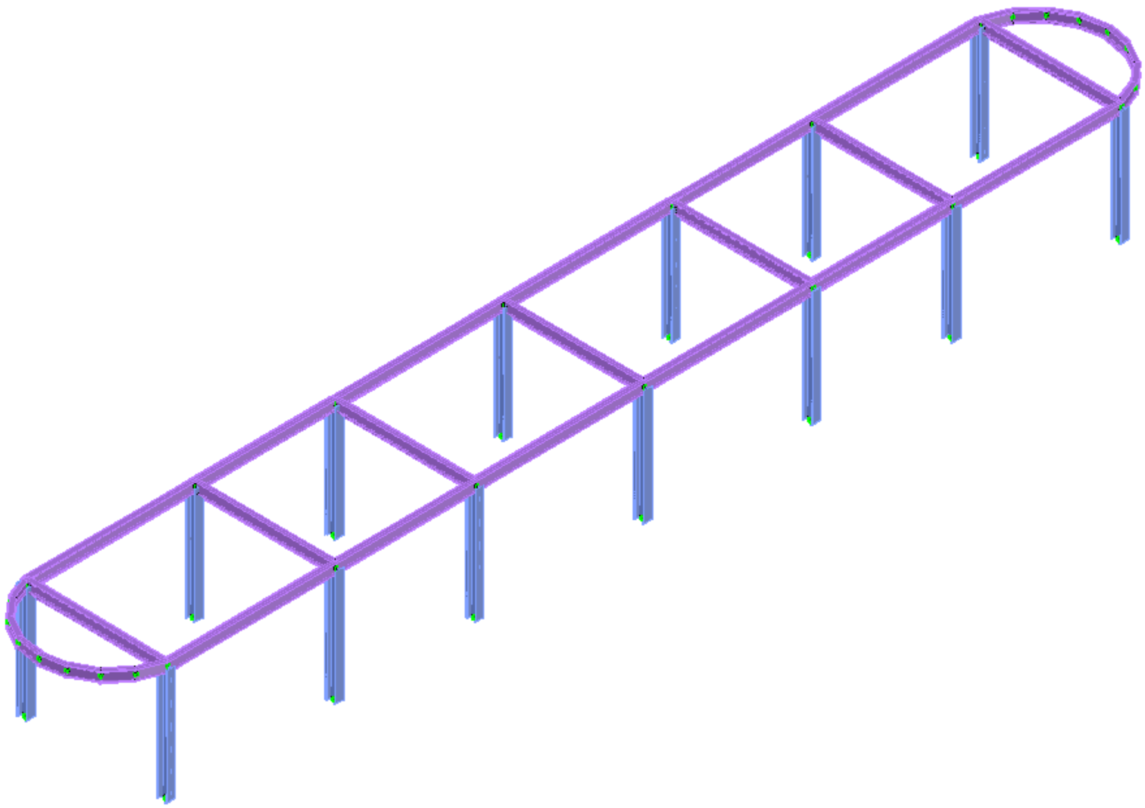
Sections | Groups

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:

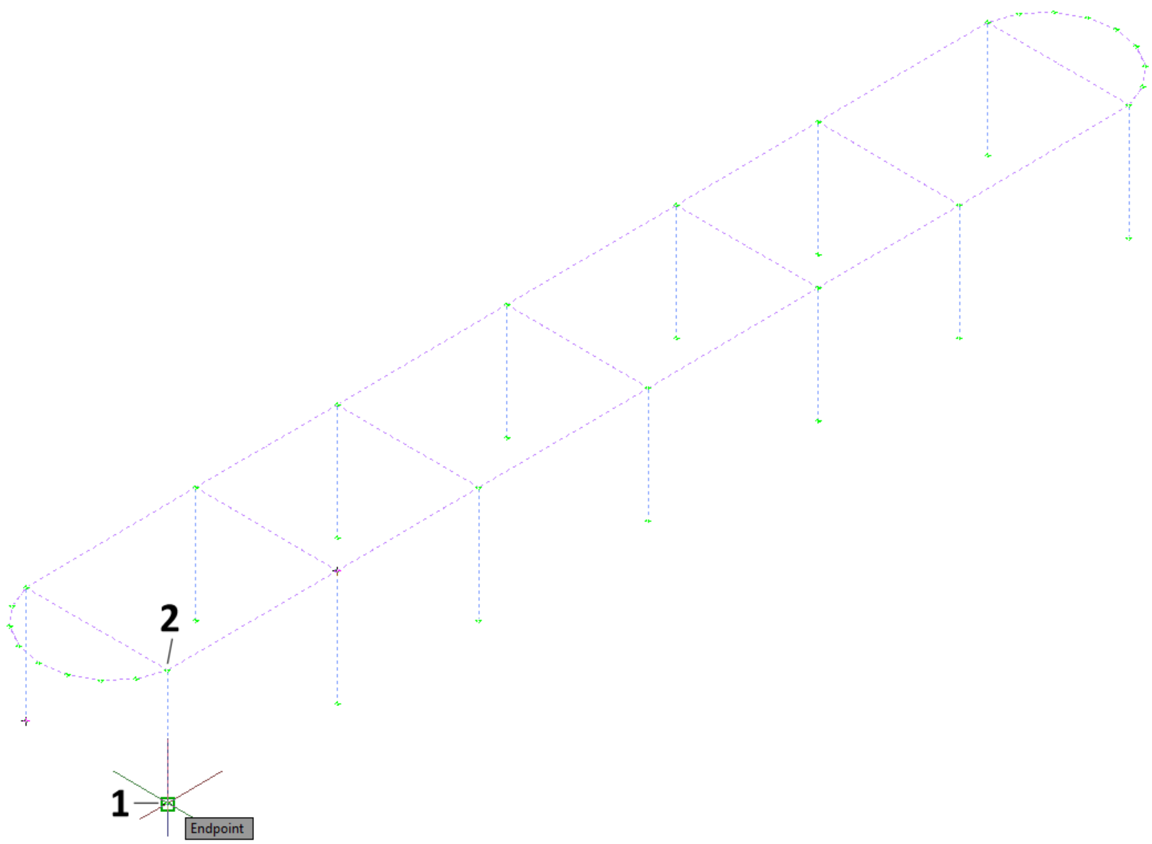


Press the icon  to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

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.

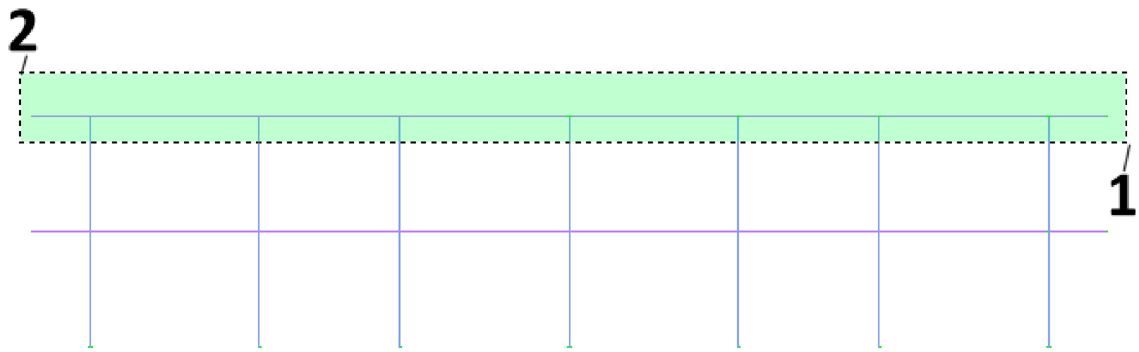
In order to *Enter second point :* click at the top of the same column such as point 2 of the picture and then press ESC in order to terminate the copy function.

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`.

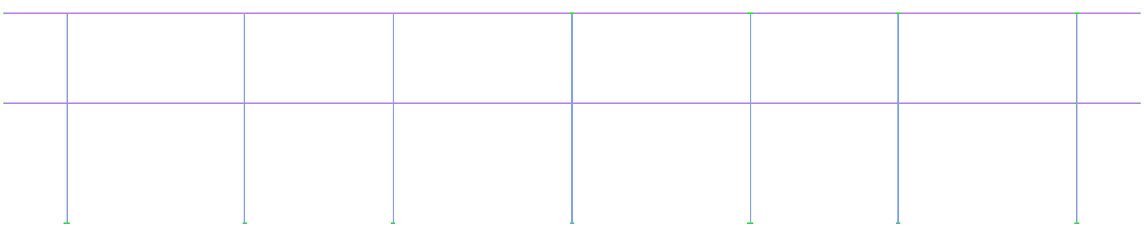
Type `MOVE` in order to initiate AutoCAD's/BricsCAD's move command and when asked to Select objects (or entities), click on points 1 and 2, as shown in the picture below, selecting all the entities that belong to the top of the 2nd floor. You will get a notification about the selected entities and press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

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

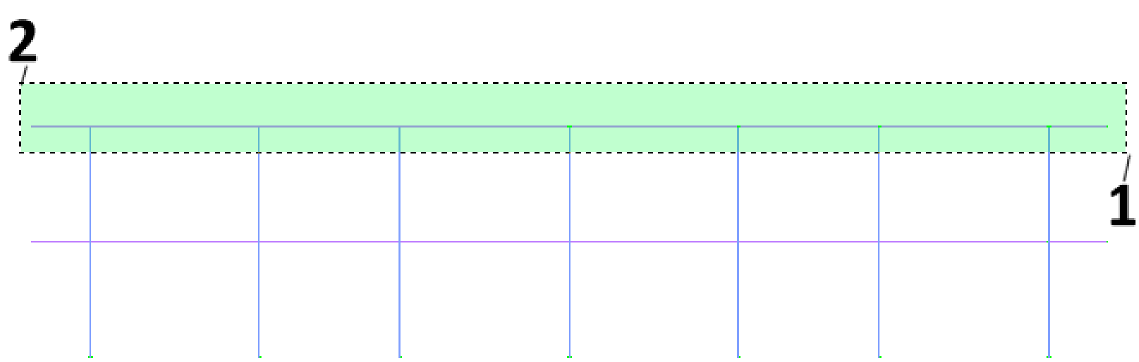
Now the height of the 2nd floor is correct and equal to 3.00 meters.



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/entites:* click on the similar two points that were used in the previous MOVE command as shown in the following figure.

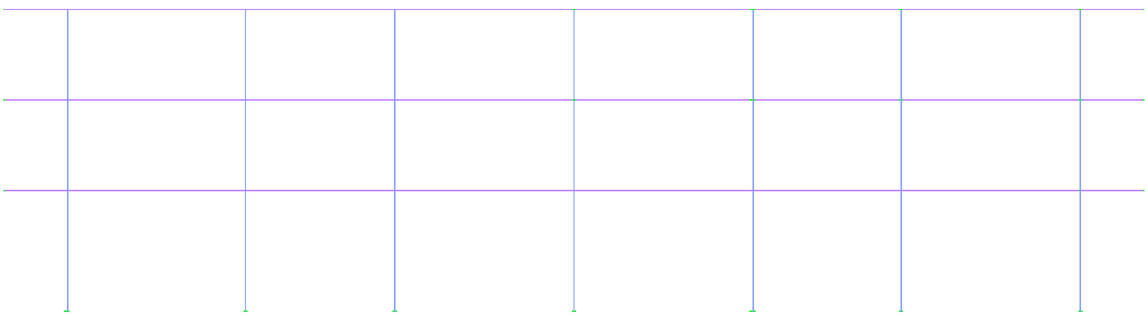


Since this is a crossing window, the columns are automaticallly selected. You will get a notification about the selected entities and press <ENTER>.

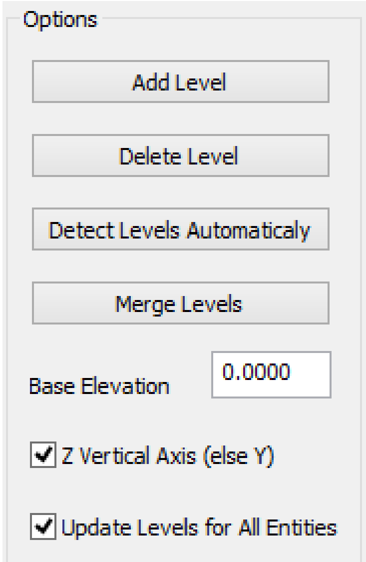
In order to *Enter base point:* click ANYWHERE on the screen. It makes no difference where you click since relative coordinates will be used to define the displacement.

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.

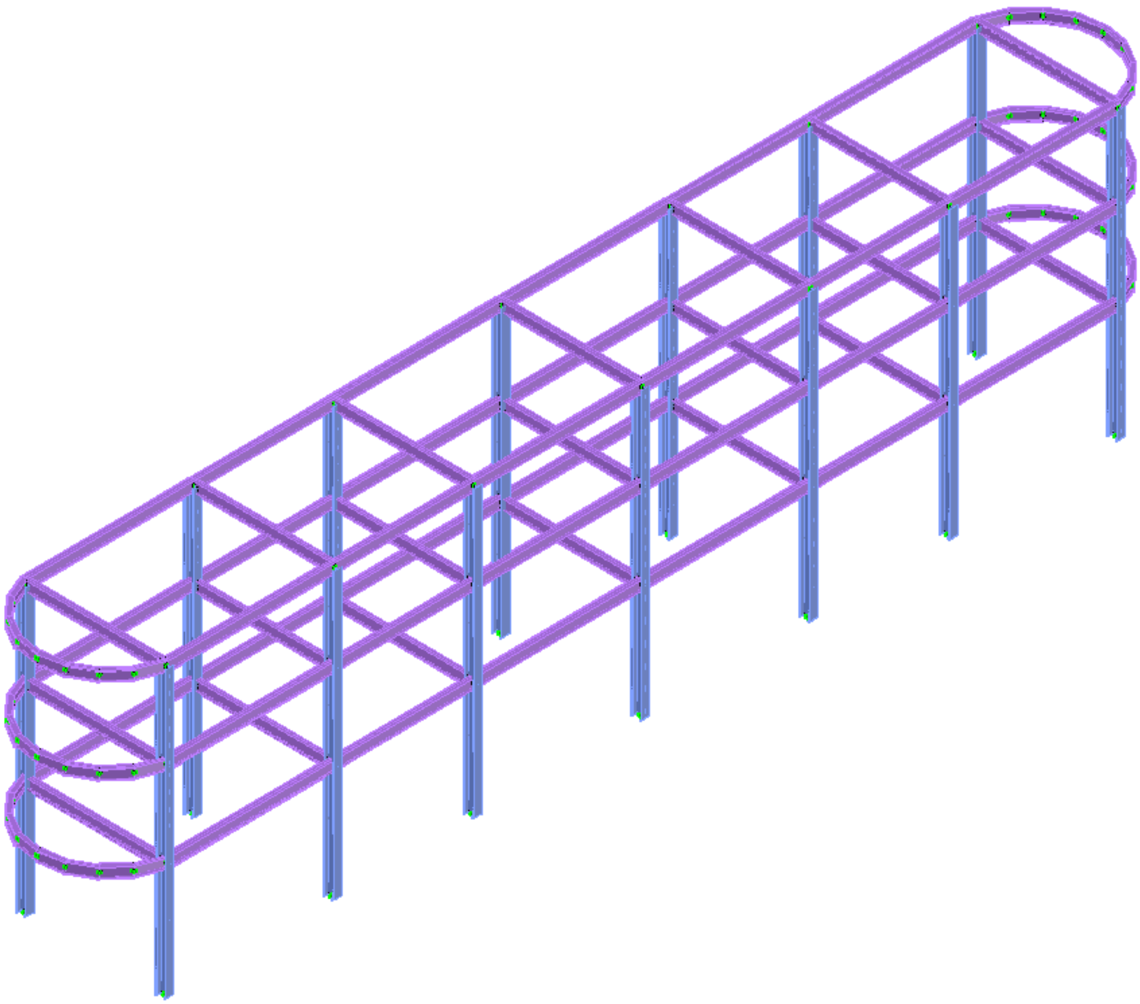
Now every Member or Joint has the correct Level property depending on its Z coordinate.

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.

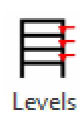
3.7. Create bracing

Step #23. Place bracing members at the front:

Press the icon 

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 

, 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 

and 

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 

and the Place Member form appears.

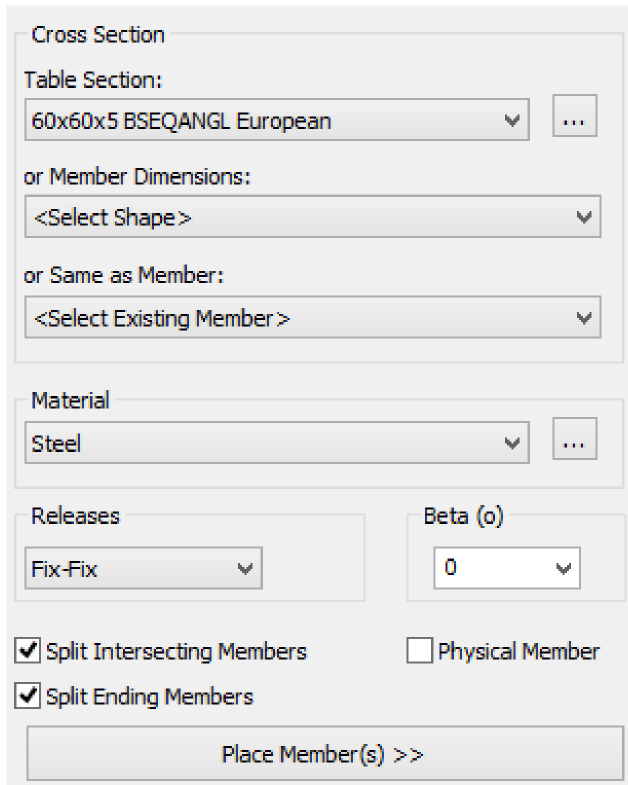
Select *60x60x5* as the cross section and make sure that Material is set to *Steel*, Releases to *Fix-Fix* and Beta angle is *0*.

Press the "Place Member(s)>>" button.

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

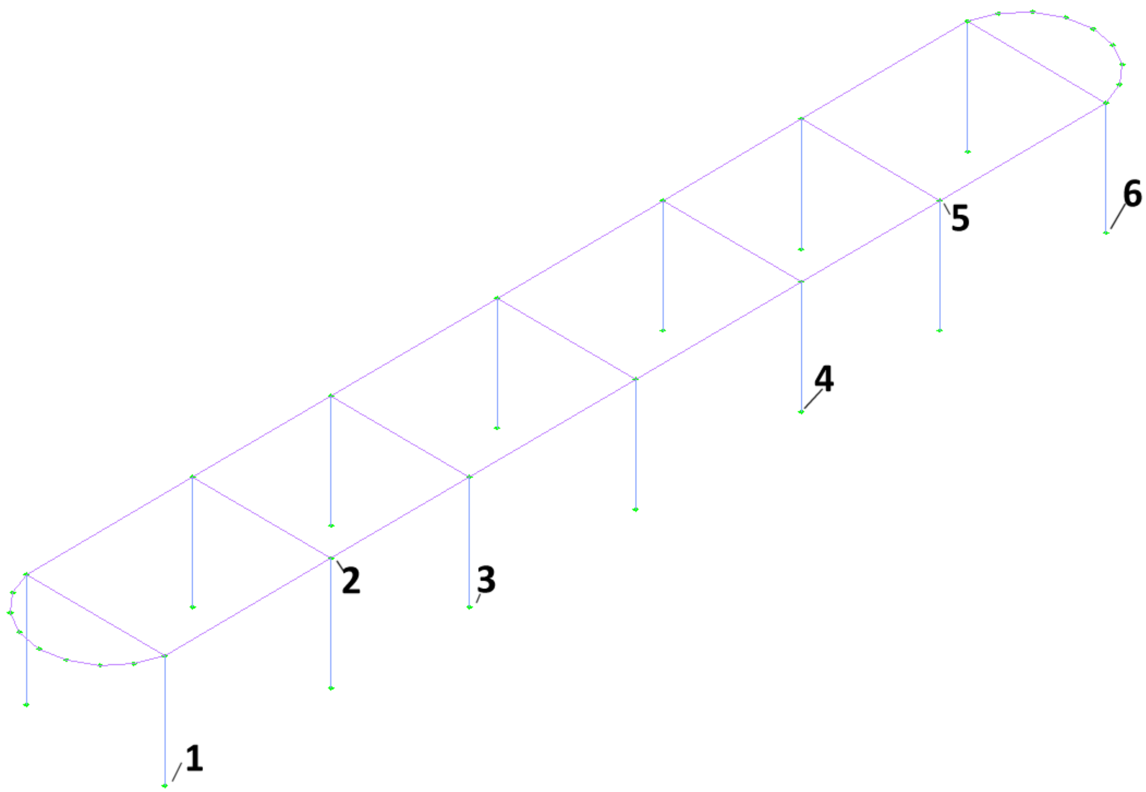
Click on joint located at Point 2 and the first bracing member is created.

Click <u>again</u> 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.

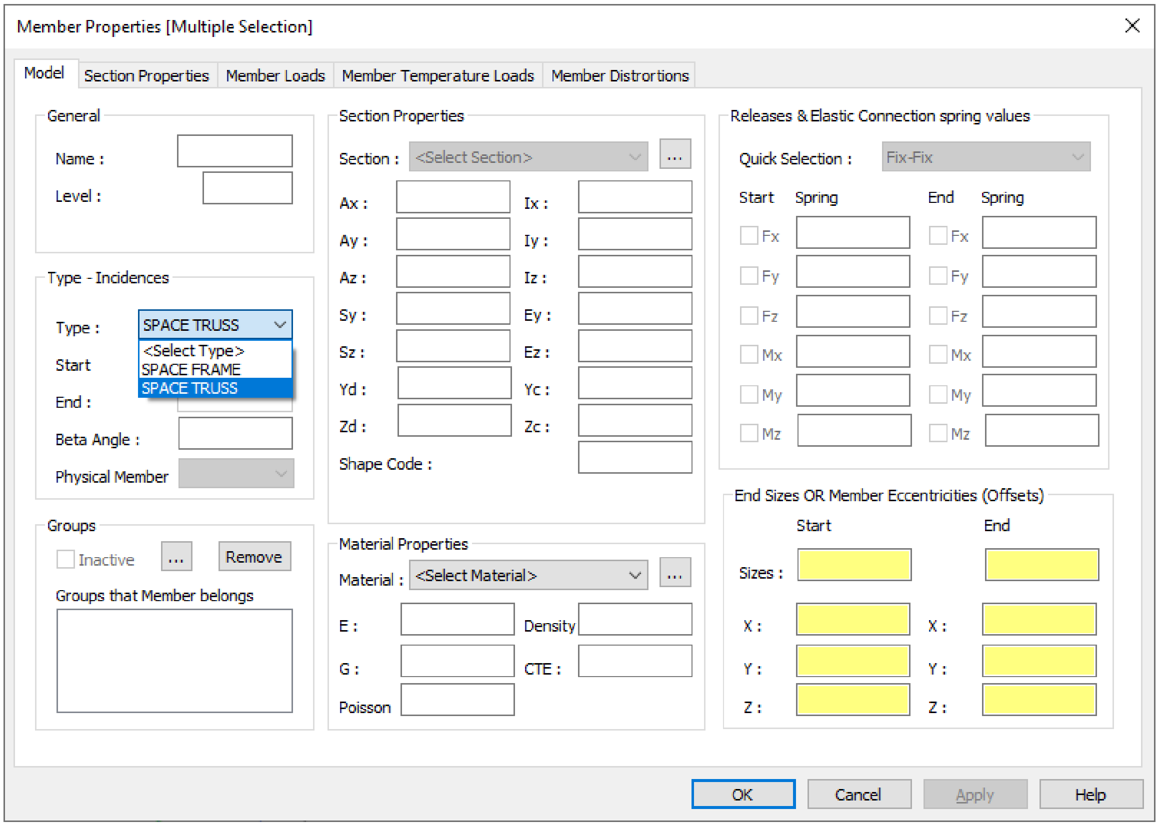
Click <u>again</u> on the joint located at Point 5, click on the joint located at Point 6 and the fourth bracing member is created.



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.



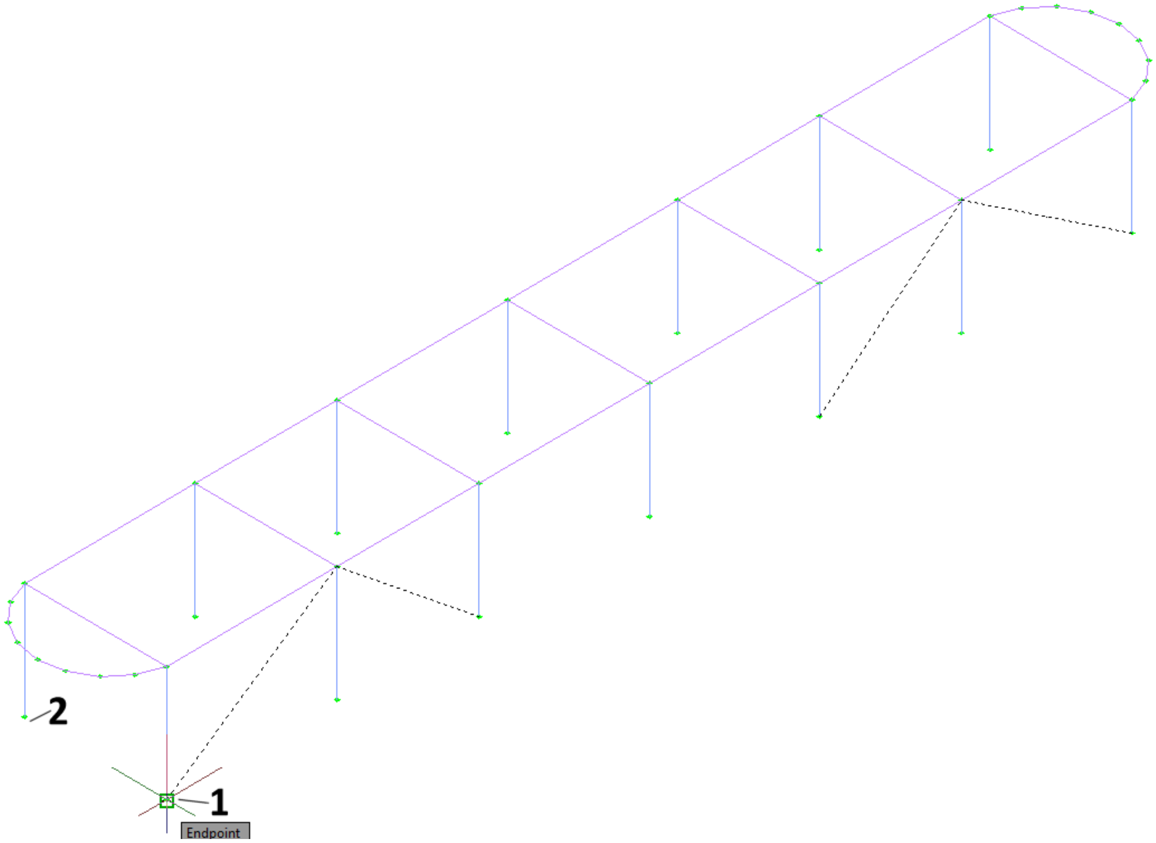
This modification applies to all selected members.

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>`.

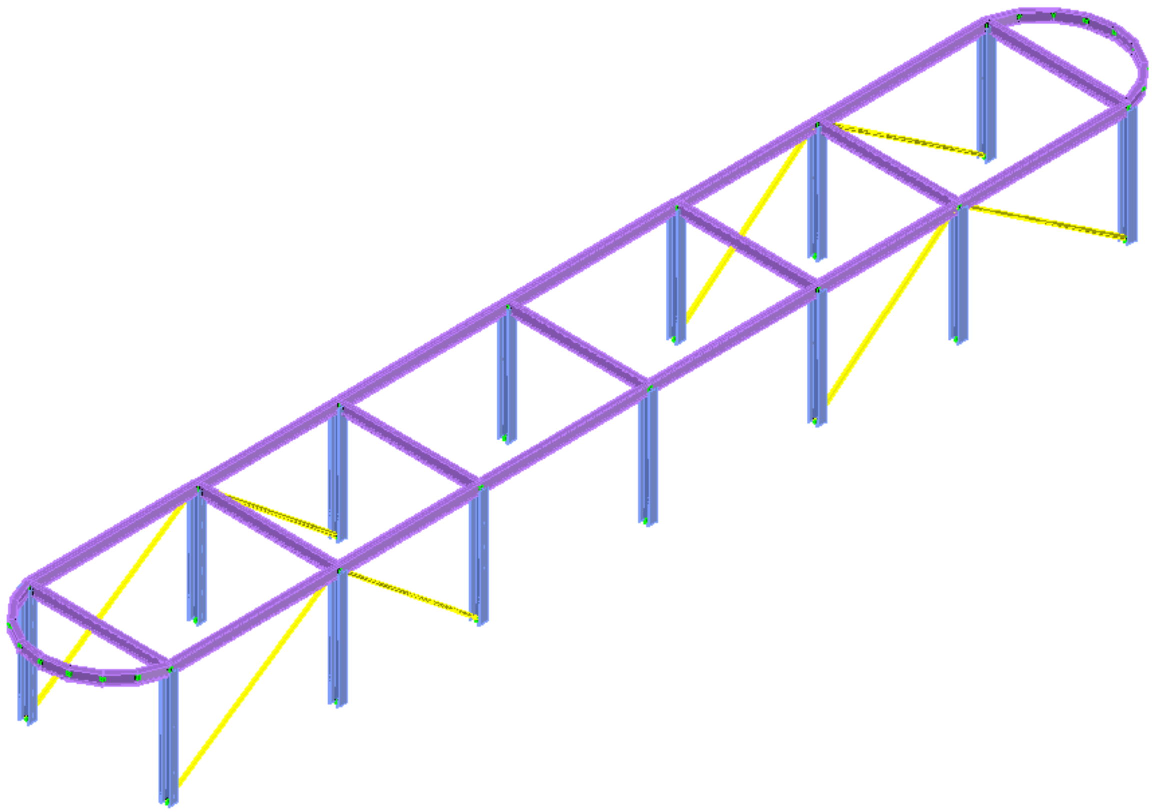
In order to *Enter base point:* click on the Joint at Point 1 of the following image.

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  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:

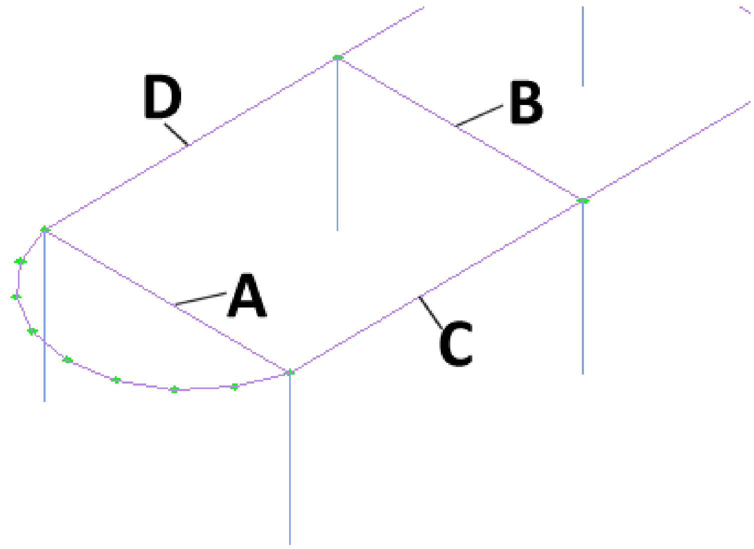
GTS CAD Modeler | M KN DEG CEN SEC | Level: 3

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 spliting 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.

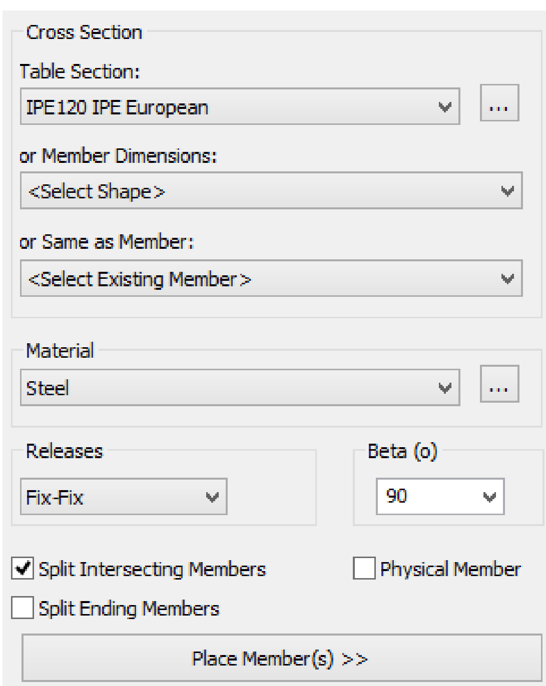
Click again on the icon Split and click on the members C and D as shown at the following image, and then press <ENTER>.

In order to define the Distance for spliting 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 

and Place Member form appears.



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.

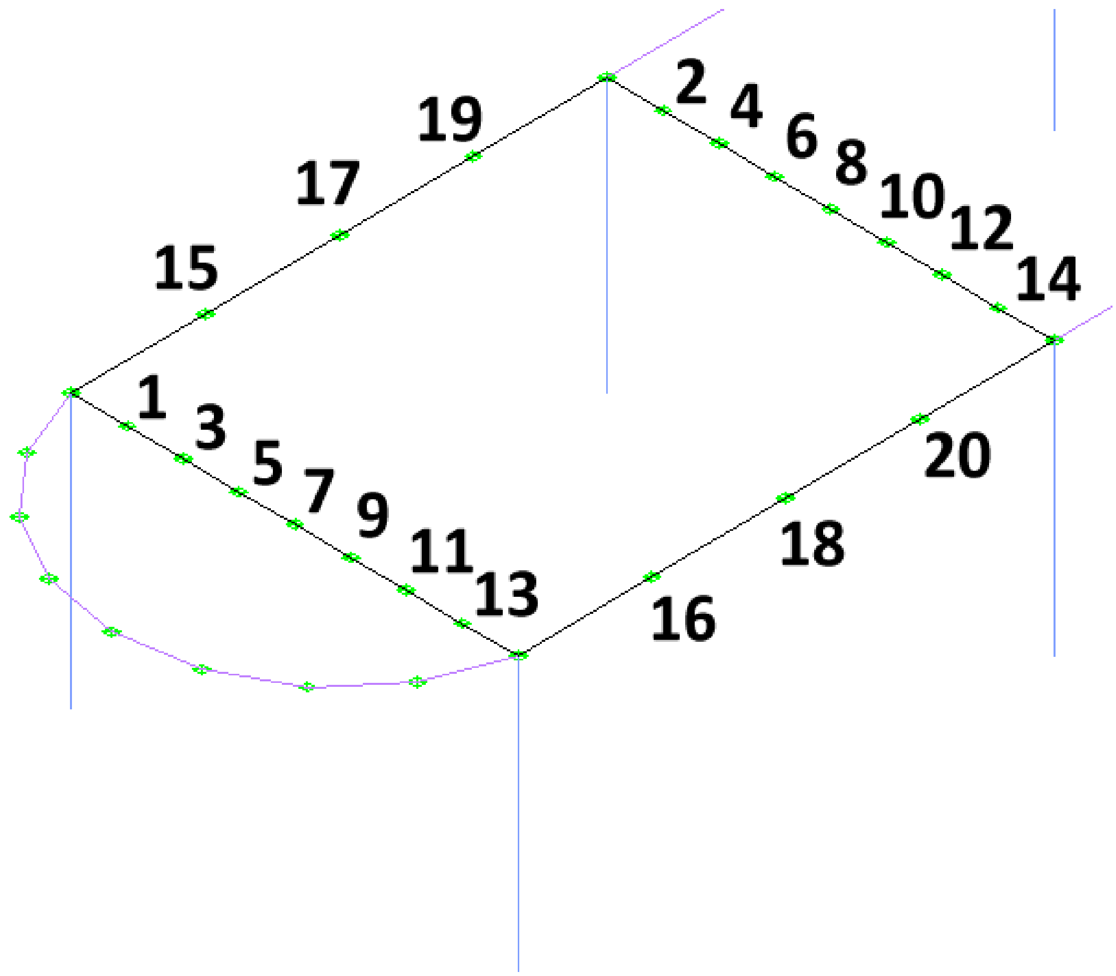
Continue by clicking on joints at points 13 and 14 and another girder member is generated.

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:

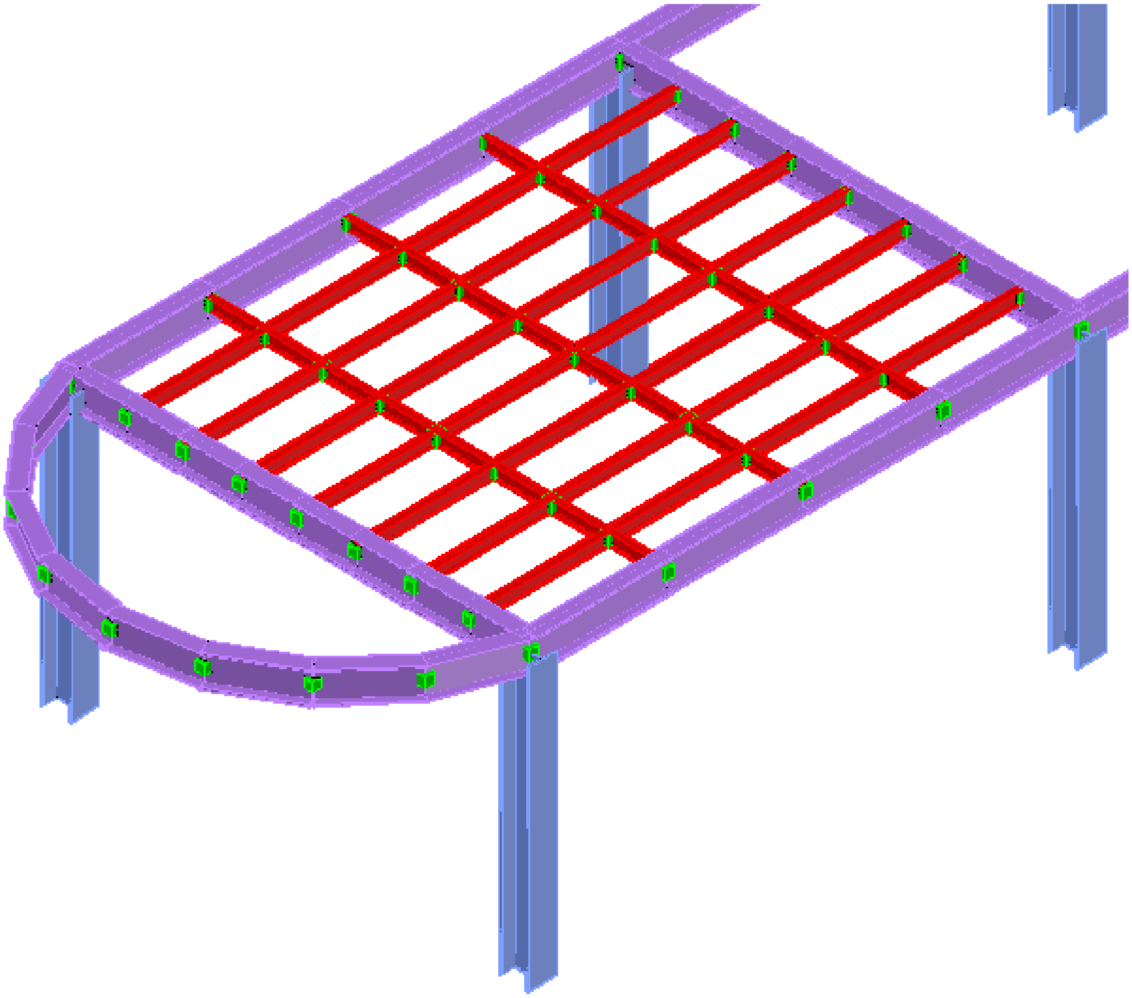
Click on the joint located at point 17 and then click on the joint at point 18 and the girder member is generated. Existing girders are split.

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.



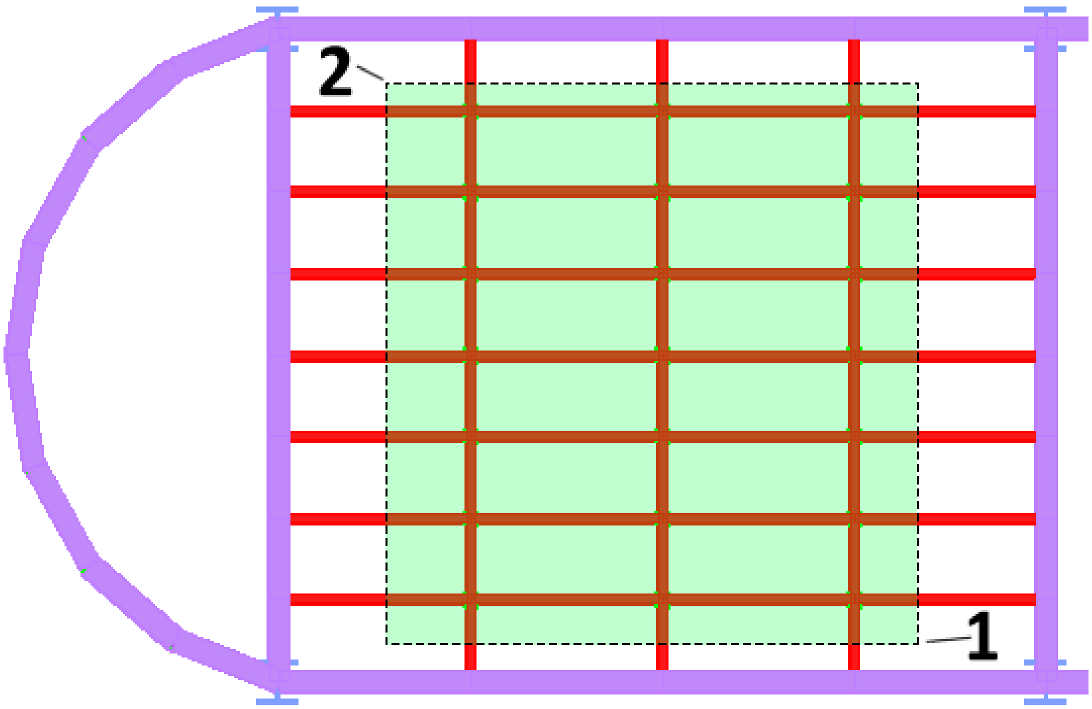
Press ESC to terminate the Generate Beam Command.

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:

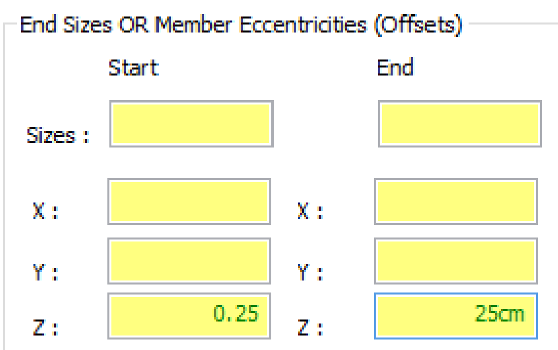


Click on the Top Icon on AutoCAD's View Cube or BricsCAD's View Chair to switch to the top view in order to be able to select girder members easily.

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.

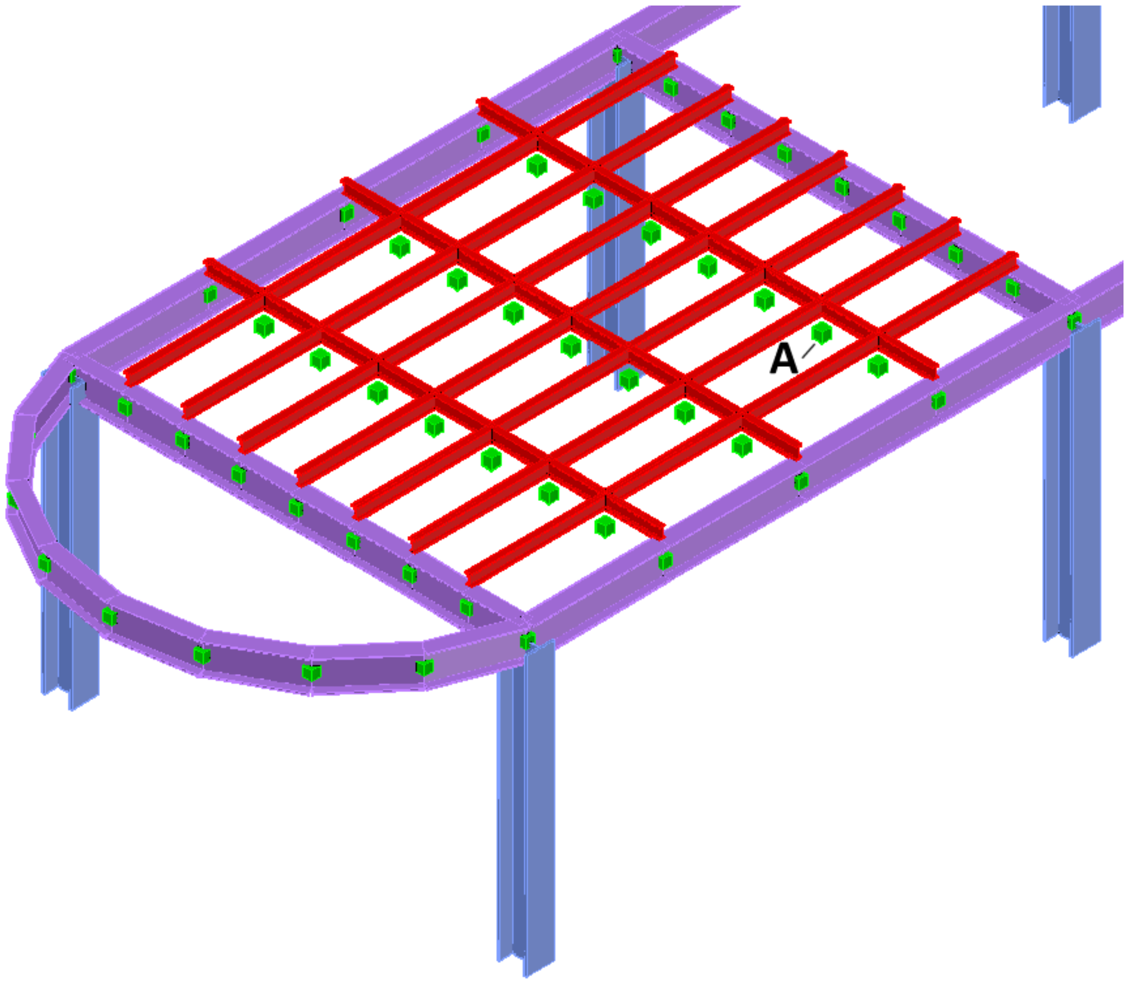


Note: Some Edit Boxes appear in yellow background and green fonts, like Eccentricities. You can use mixed units in the yellow edit boxes. To understand this feature enter 0.25 for the Z Starting Eccentricity and 25cm as the Z Ending Eccentricity.

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.

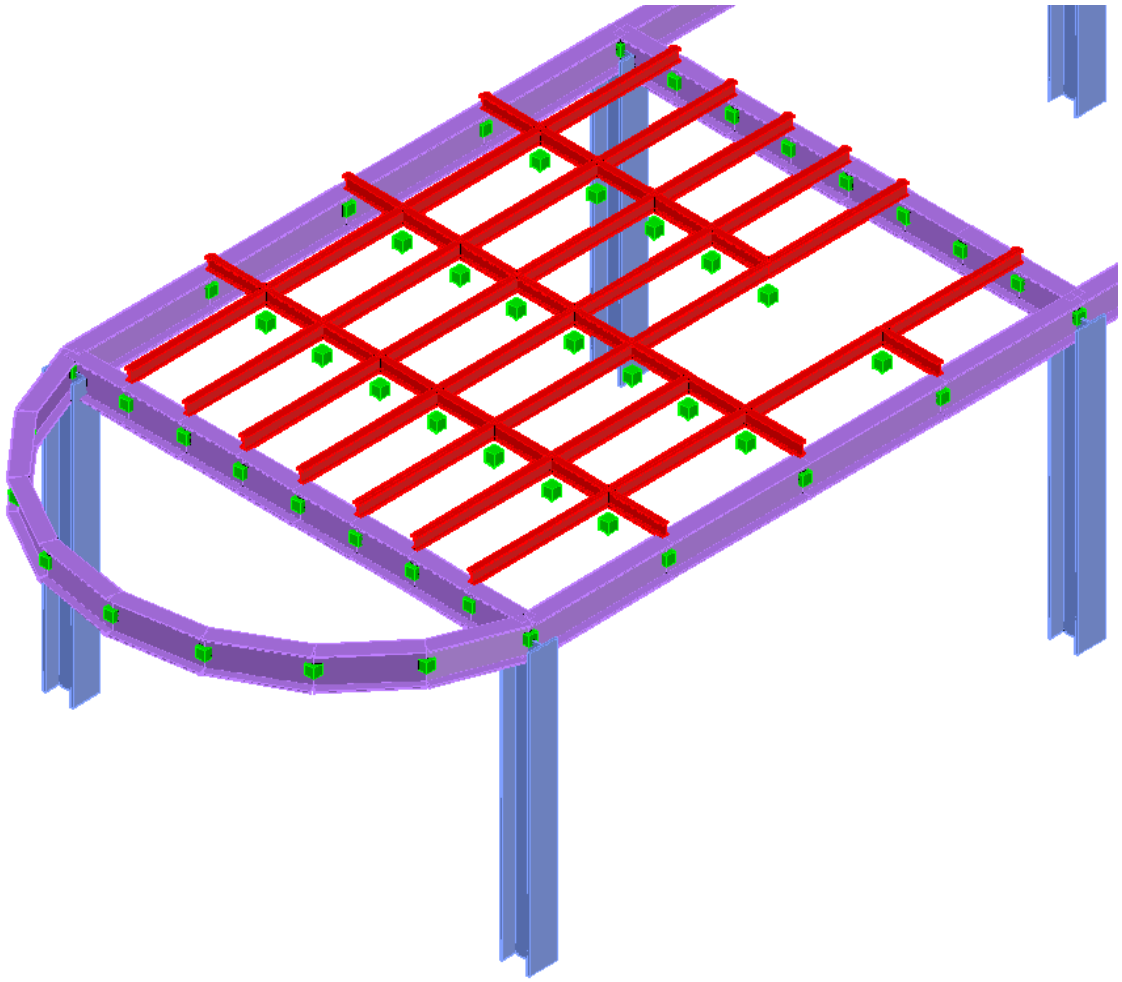


3.9. Create an opening

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.



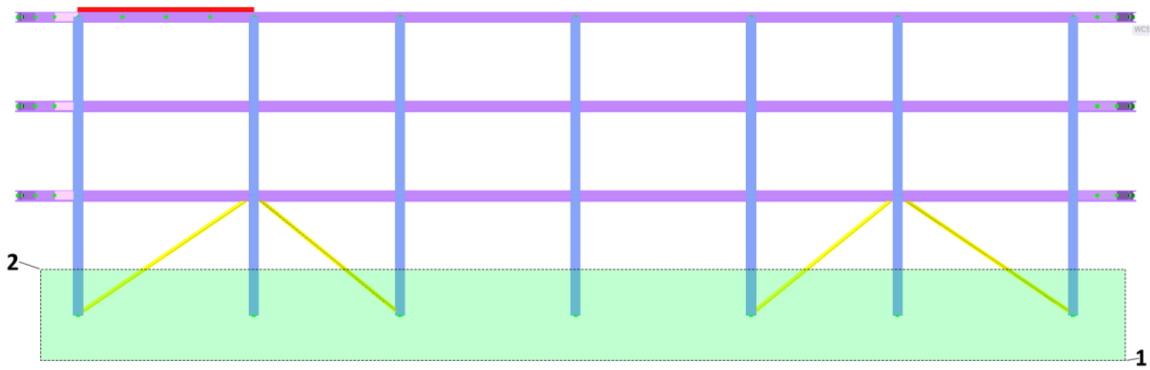
3.10. Create Supports

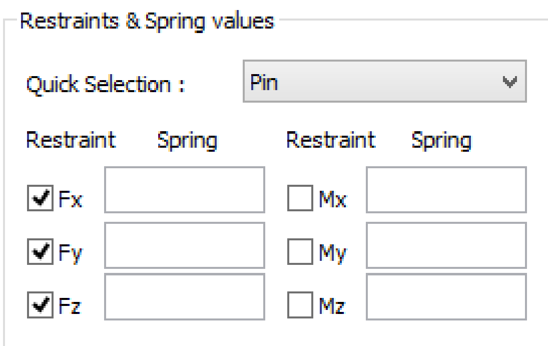
Step #31. Support the joints at the base of the model:

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.





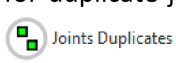
The Joint Properties [Multiple Selection] form appears.

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.

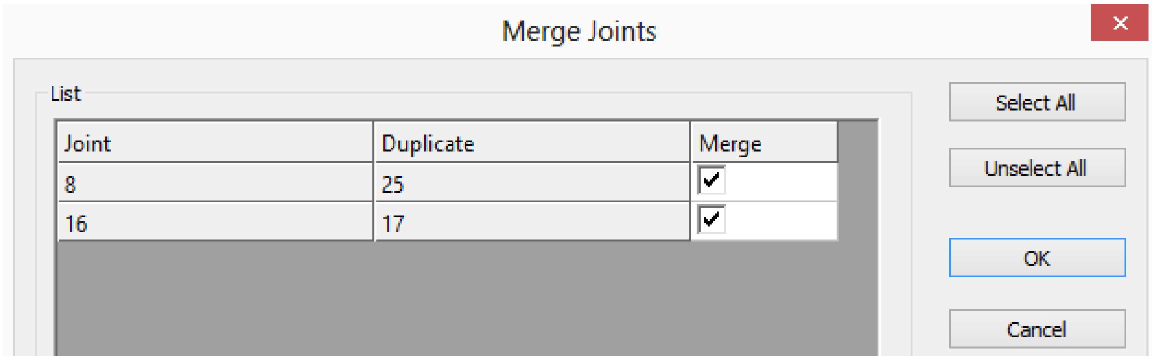
3.11. Check the model

Step #32. Check for duplicate joints: In order to check for joints having the same coordinates, click on the icon 

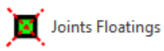
, under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area.

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 2^{nd} 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 

, 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*.

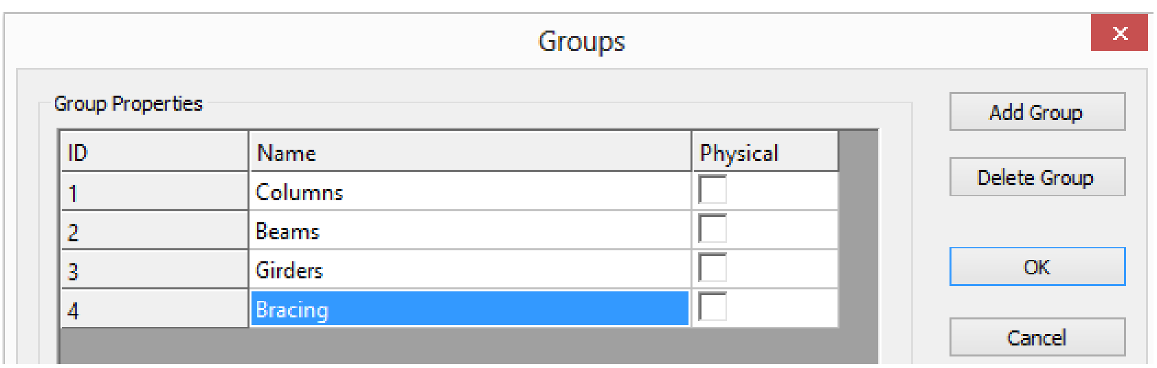
Note: You can also run all other checks of the same drop list, to check for Interference Joints, Duplicate Members, Zero Length Members, Duplicate Names and Database Integrity. You should not get any errors or warnings.

3.12. Define Groups

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

Click on the icon 

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.

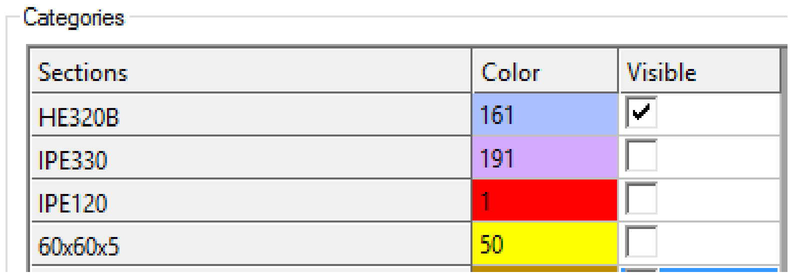
Press the *Add Group* button and enter `Bracing` as *Name* of the group.

Press OK to close the form.

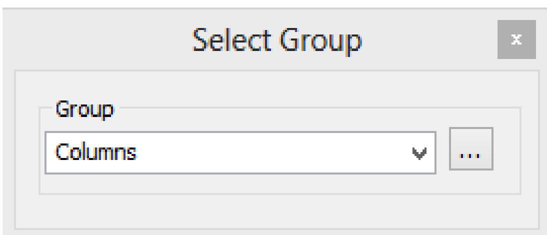
Step #35. Add Columns to their Group.

Click on the icon 

and make only profiles HE320B visible by unckecking all others.



Press OK.

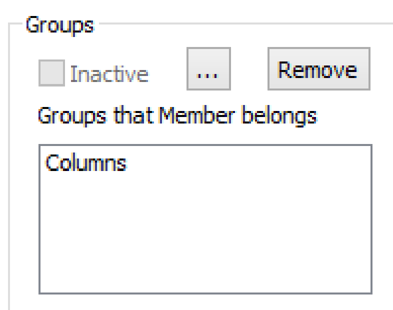


Click on the icon 

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.

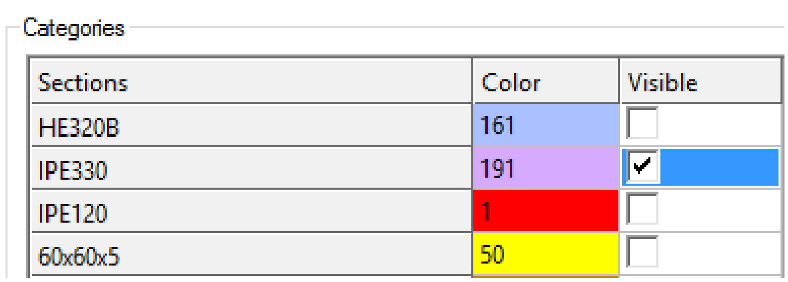
To verify that the group is applied correctly, you can double click on a column and in the Member Properties form, you will be able to see its Group definition.



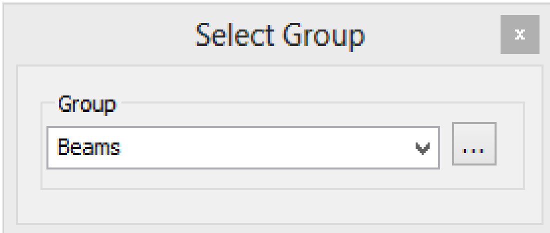
Step #36. Add Beams to their Group:

Click on the icon 

and make only profiles IPE330 visible by unckecking all others.



Press OK.

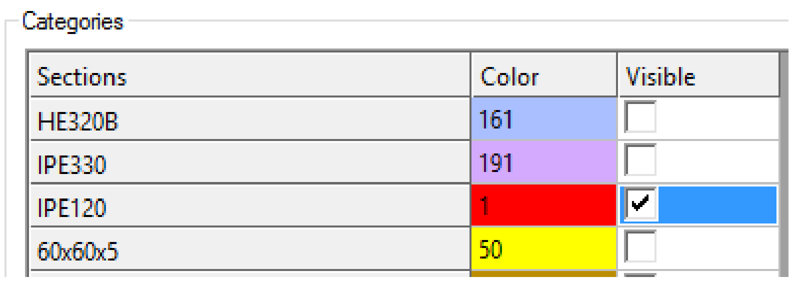


Click on the icon +Members and make sure that the Group Beams is selected as the active group.

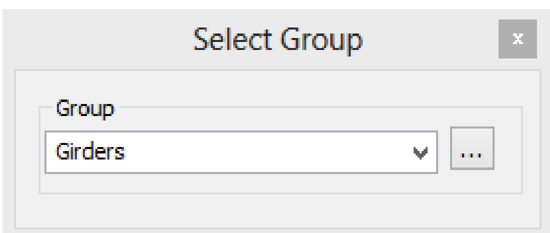
Select everything on the screen in a full window and press the Enter key.

Step #37. Add Girders to their Group:

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



Press OK.

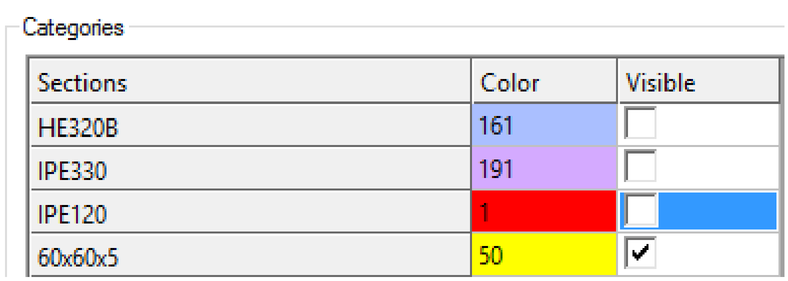


Click on the icon +Members and make sure that the Group Girders is selected as the active group.

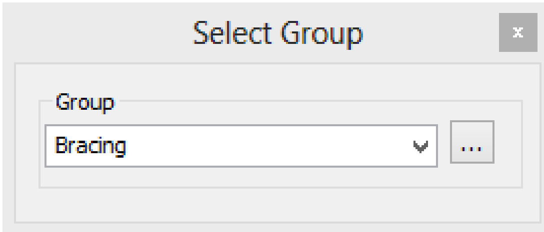
Select everything on the screen in a full window and press the Enter key.

Step #38. Add Bracing to their Group:

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



Press OK.



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

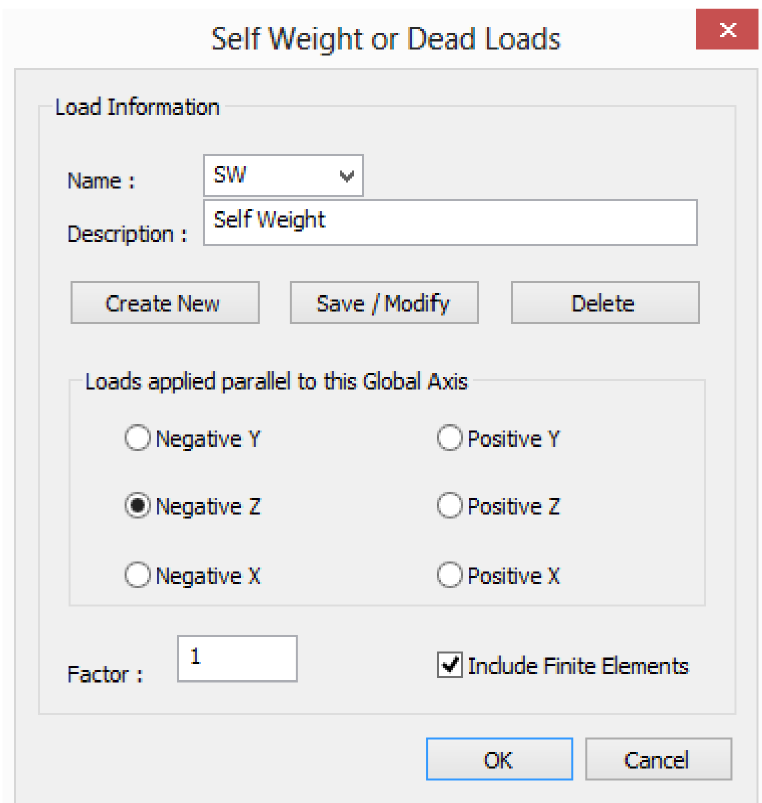
Select everything on the screen in a full window and press the Enter key.

3.13. Define Loads

Step #39. Define Self Weight: Click on the icon  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:

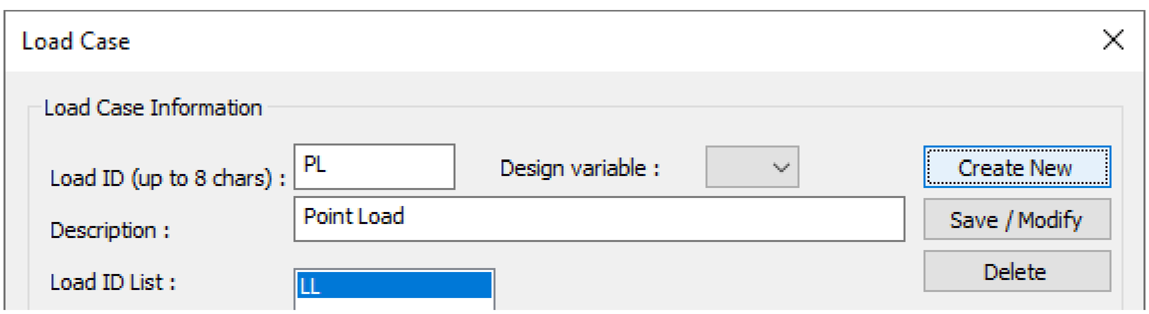
- *LL* as Name
- *Live Load* as the Load Description
- *L* as Design variable

and press Create New.

Enter:

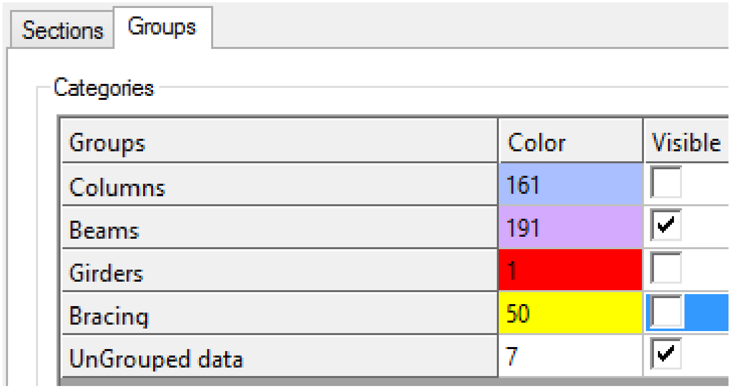
- *PL* as Name
- *Point Load* as the Load Description

and press Create New.



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.



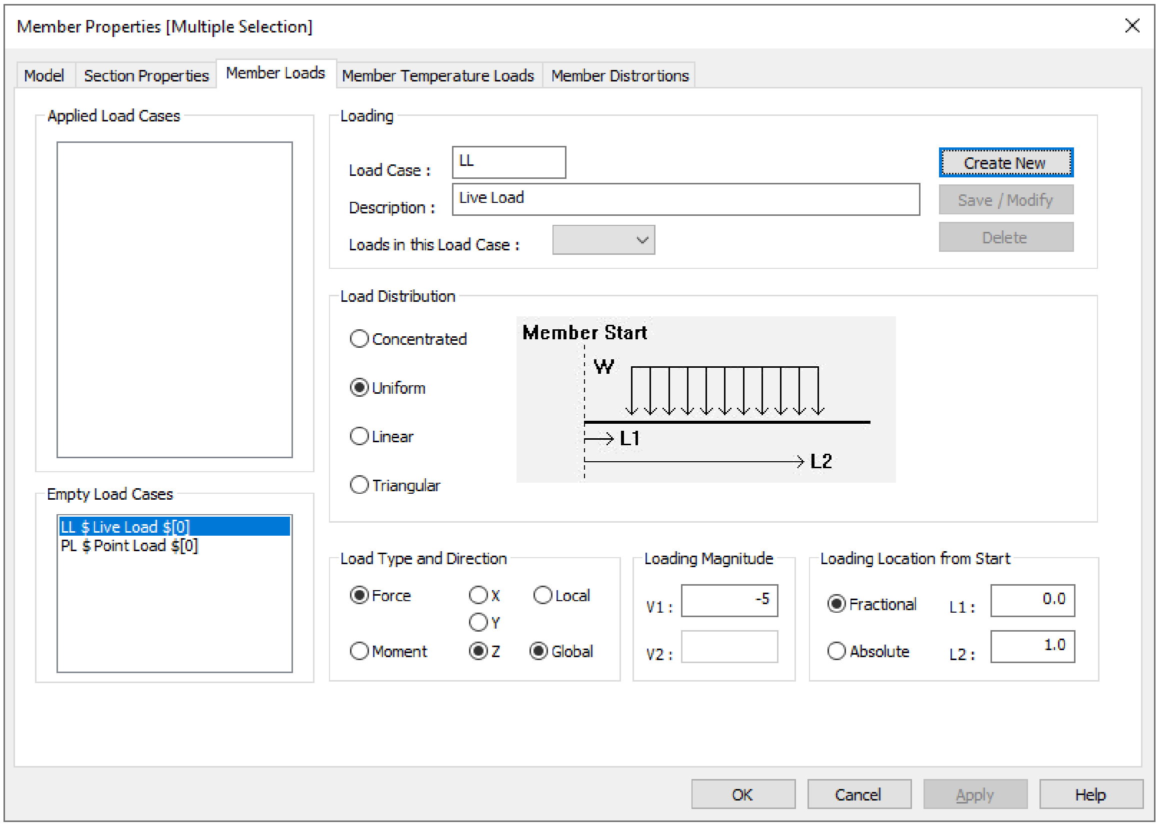
Click on the icon Member . Using a full window, select all entities that appear on screen and press <ENTER> to finish with the selection.

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

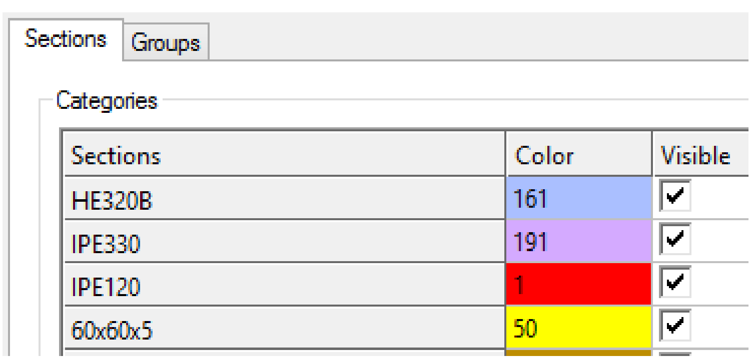
Press **Create New** and the number 125 appears next to the $ symbol the Empty Load Cases list box. This is a notification that 125 members are loaded in Load Case LL.



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

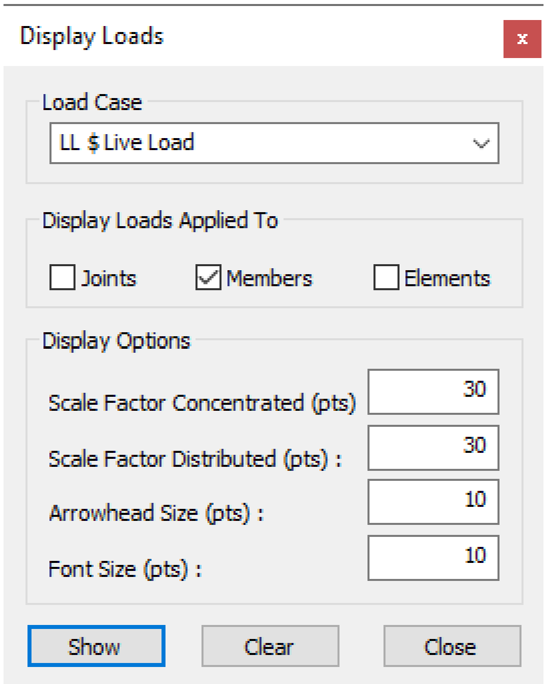
By double clicking on one beam and then selecting the Member Loads tab and clicking on LL at the Applied Load Cases List Box, you will be able to view and edit the existing loading values of the specific beam.

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.



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

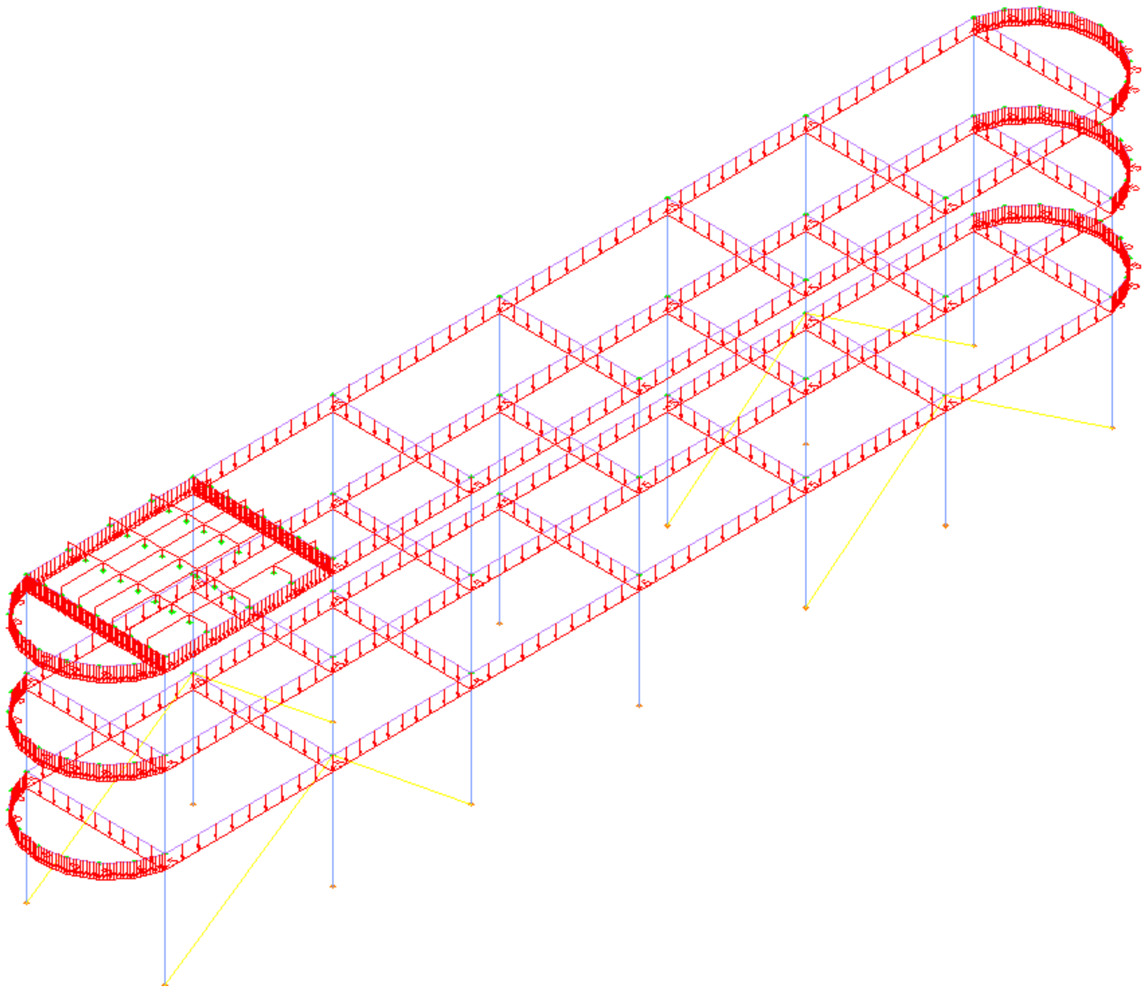
On the Menu Bar, click on GTS Display>Display>Member Loads and the Display Loads form appears:



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

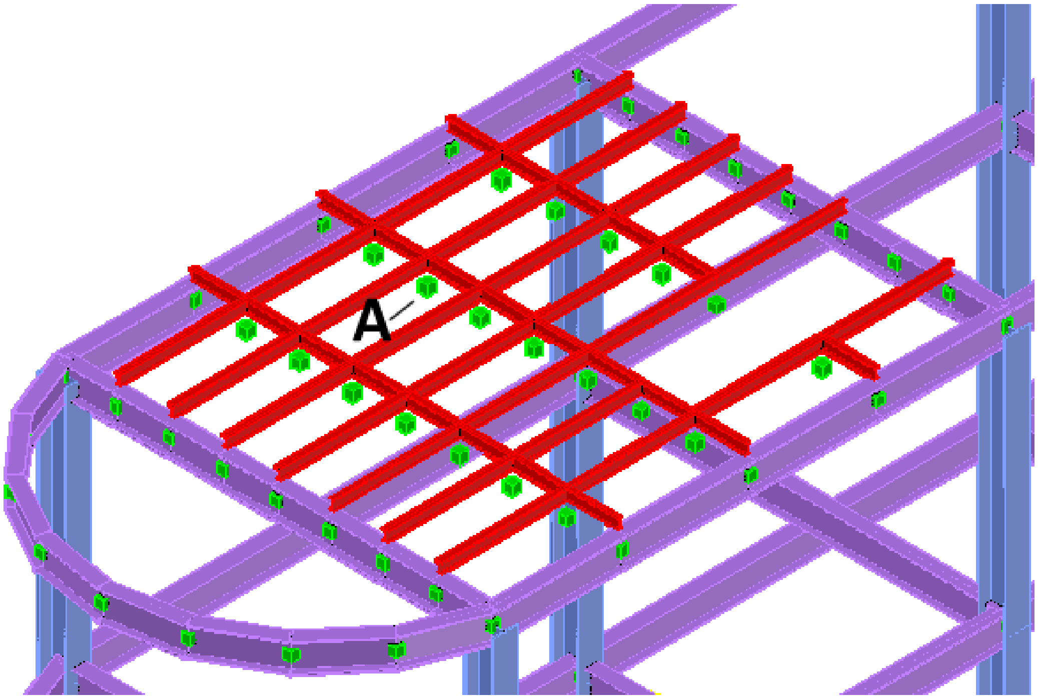
Press Show and the loading arrows are displayed.

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.



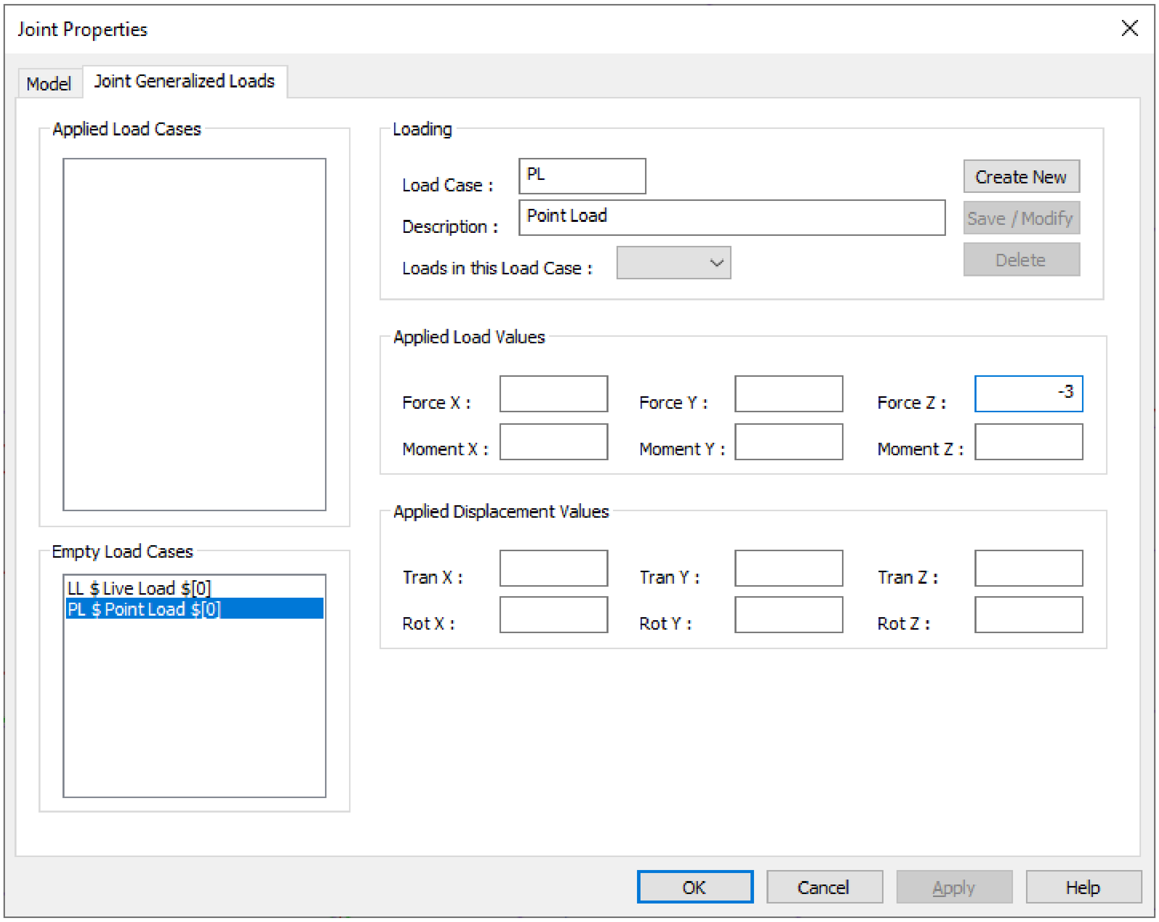
Zoom closer to the specific point using AutoCAD's/BricsCAD's zooming functions.

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.

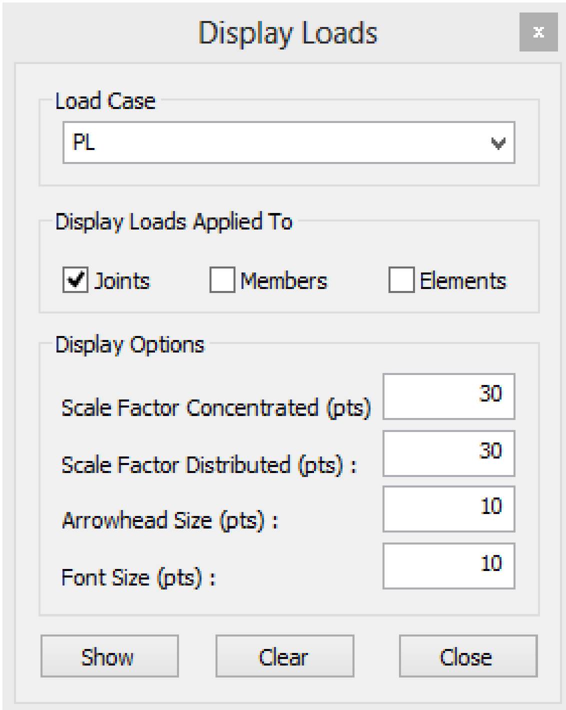
Press Create New, and the number 1 appears next to the $ symbol in the Load Cases list box. This is a notification that 1 joint is loaded under the Load Case PL.



Step #44. View Joint Load: Click at the icon 

to switch to wireframe view.

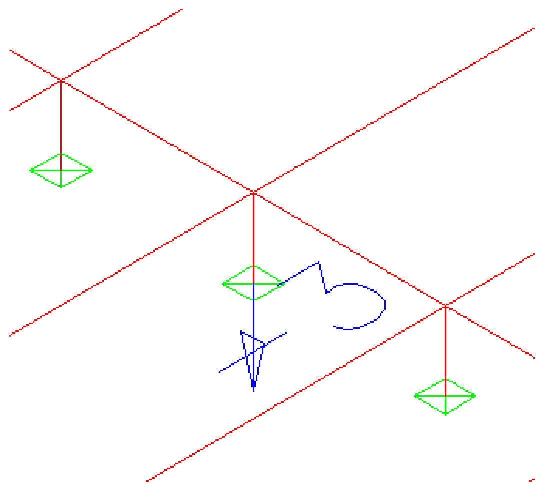
On the Menu Bar, click on GTS Display>Jont Loads and the Display Loads form appears:



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

Press Show and the loading arrows are displayed.

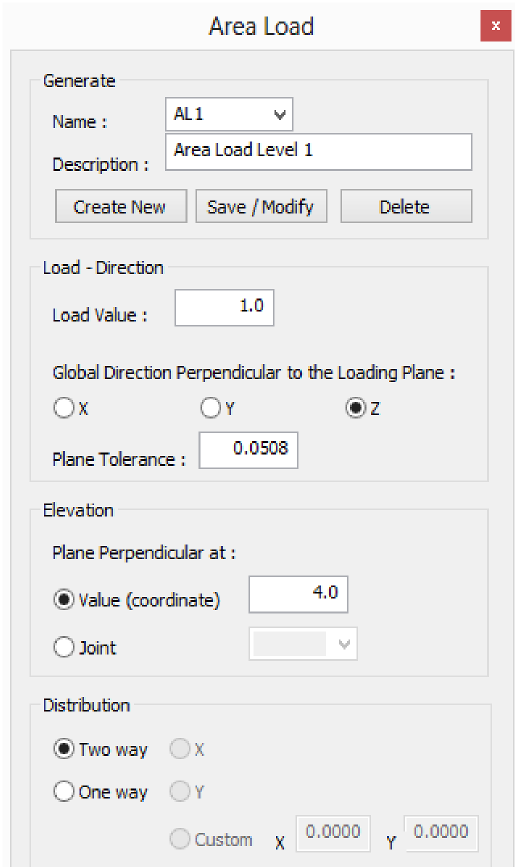
Click on Clear when you are done and Close to exit from the dialog.



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.

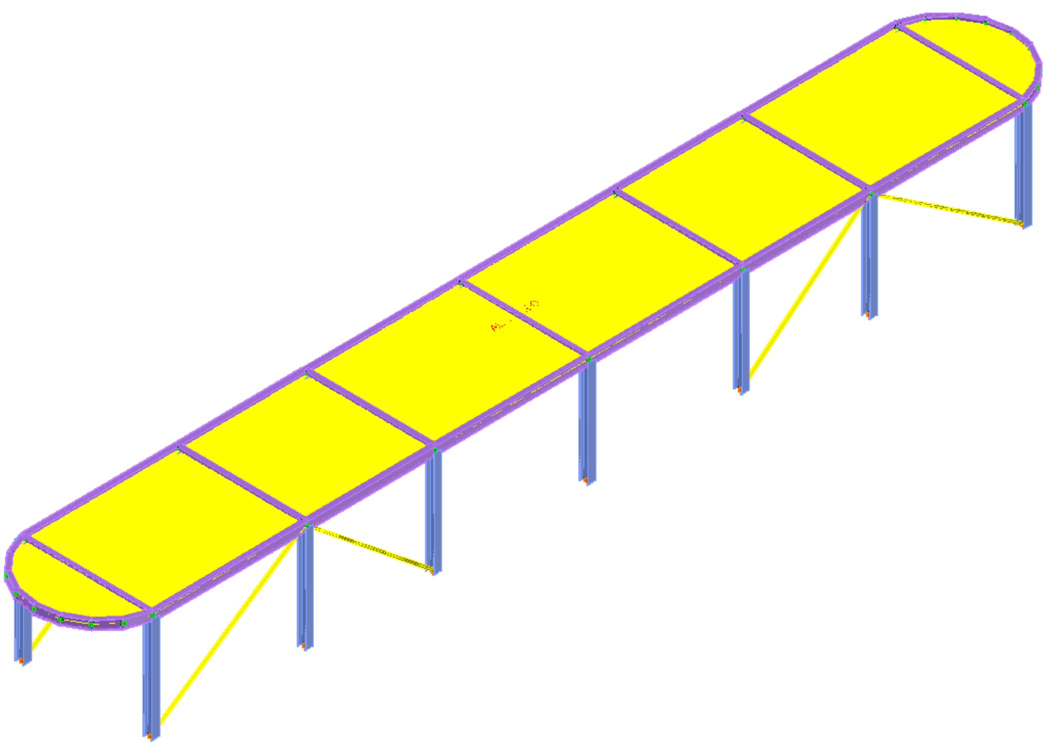
Click on the icon Area Load in the Loads panel of the ribbon.



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

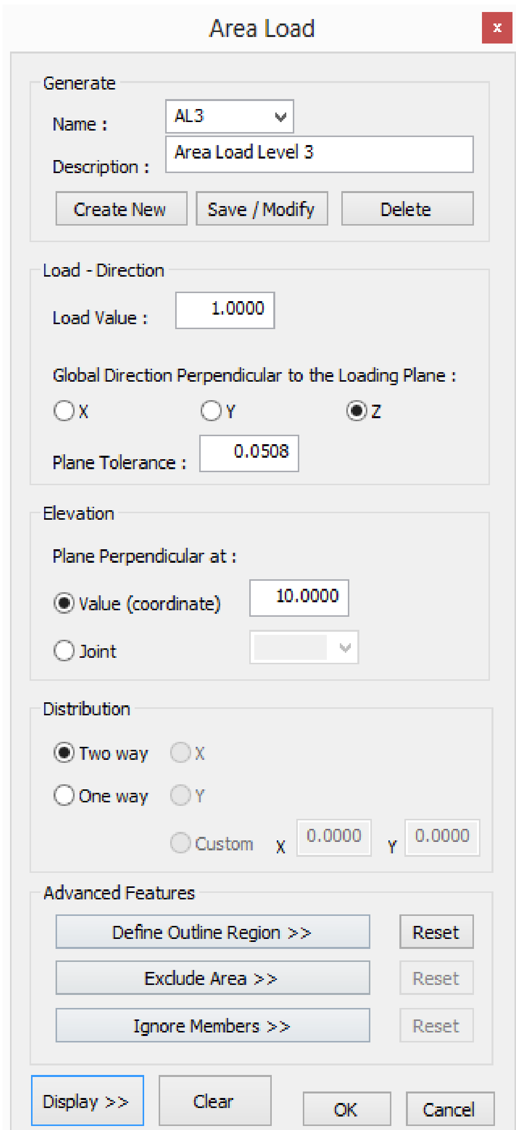
Press Display >> and the loaded areas that were automaticaly detected are displayed in yellow solid hatch, as shown below.



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 untill "Level 3" is displayed.

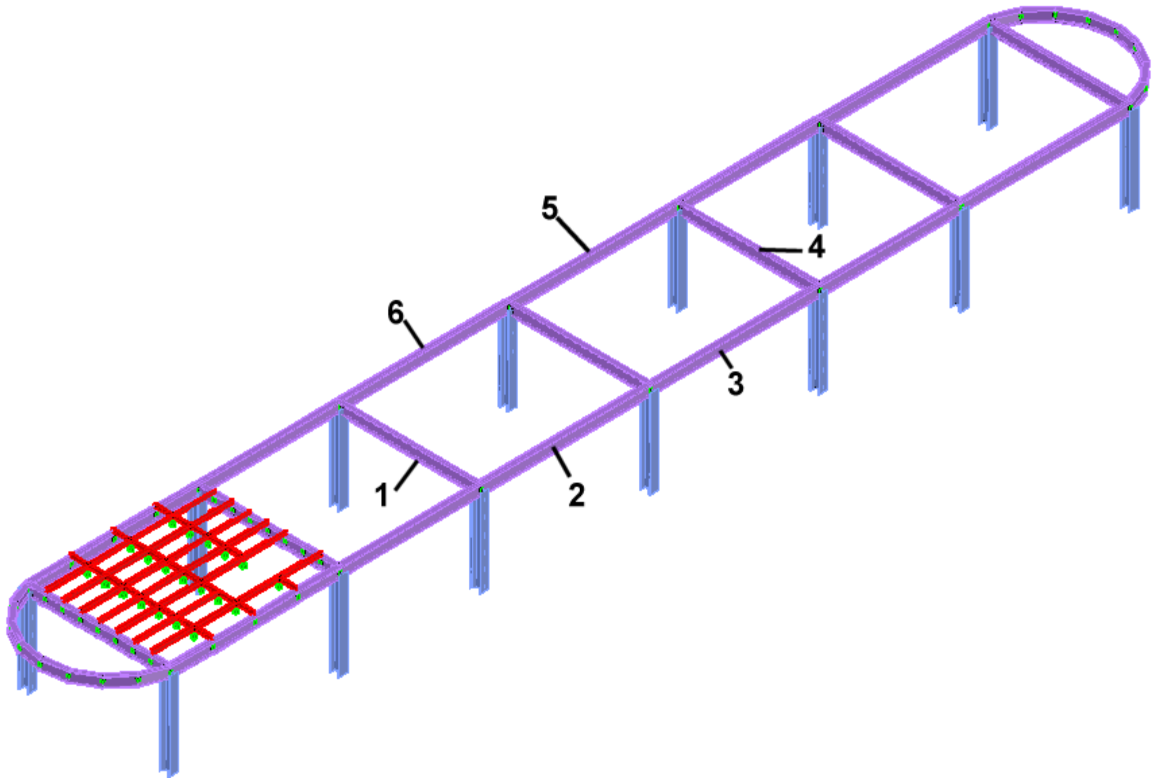
Click on the icon Area Load in the Loads panel of the ribbon.



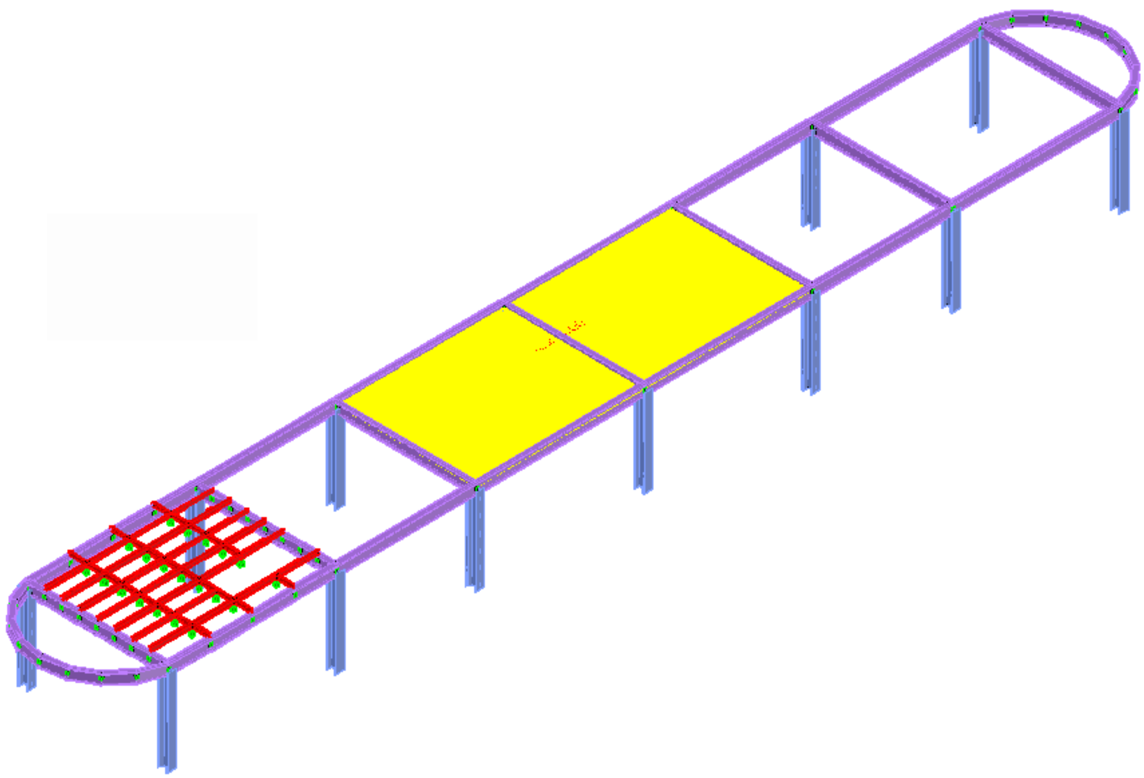
Type:

- *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.



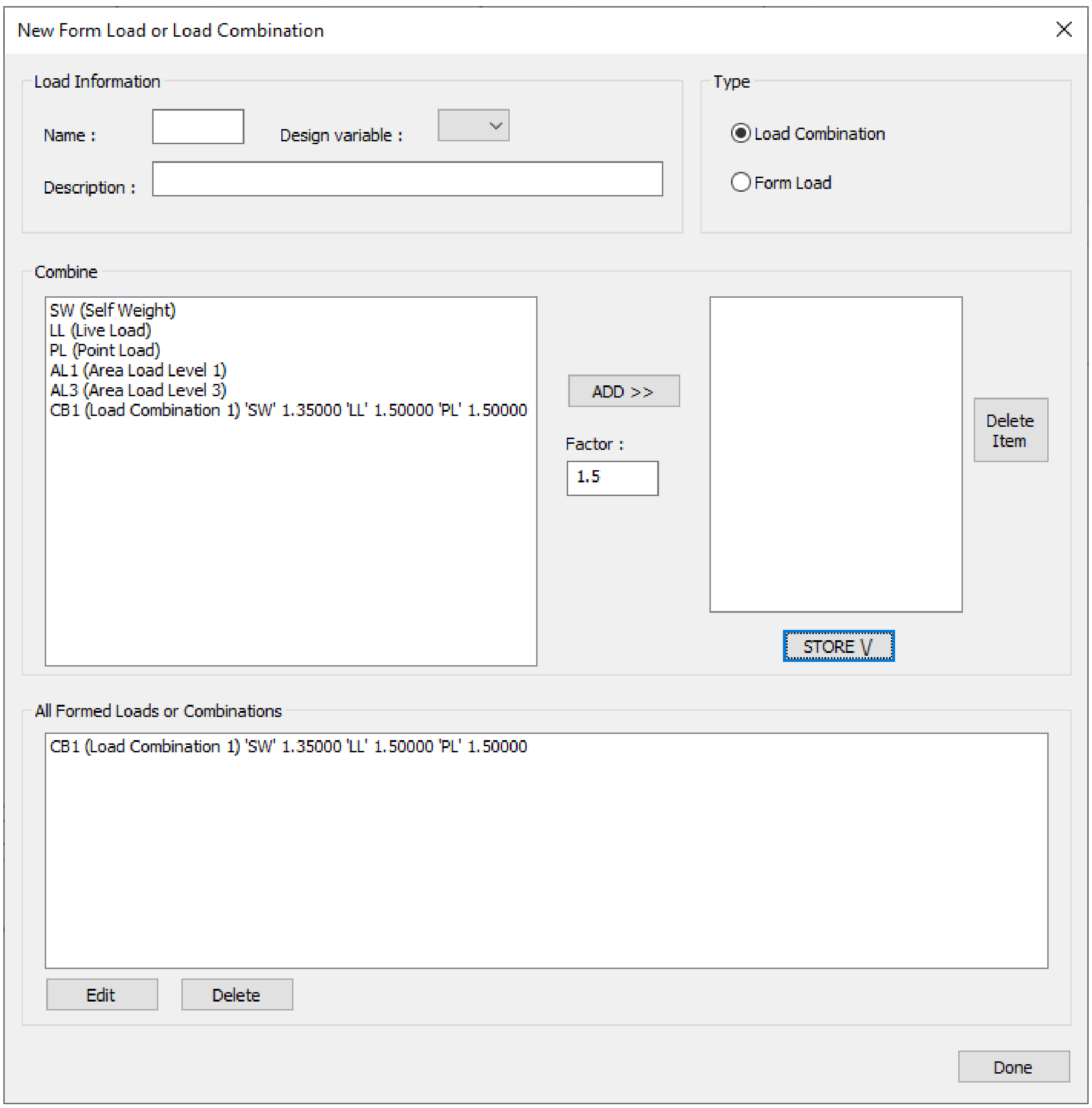
Press Display >> and the loaded area will be displayed in yellow solid hatch, as shown below.



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

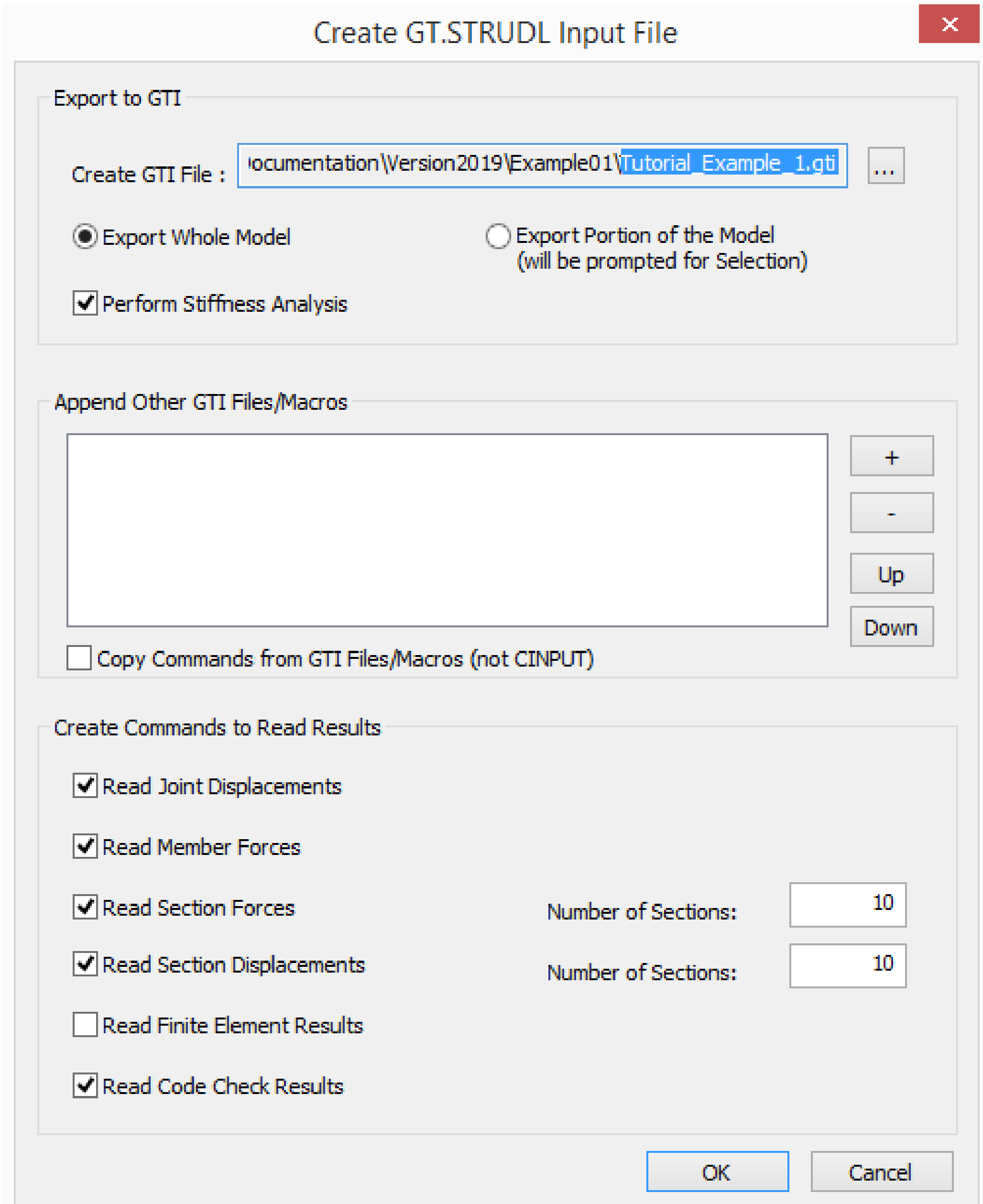
Step #47. Define Load Combinations: Click on the icon Combinations and the Load Combination dialog appears. Enter:

- *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.



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.



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.

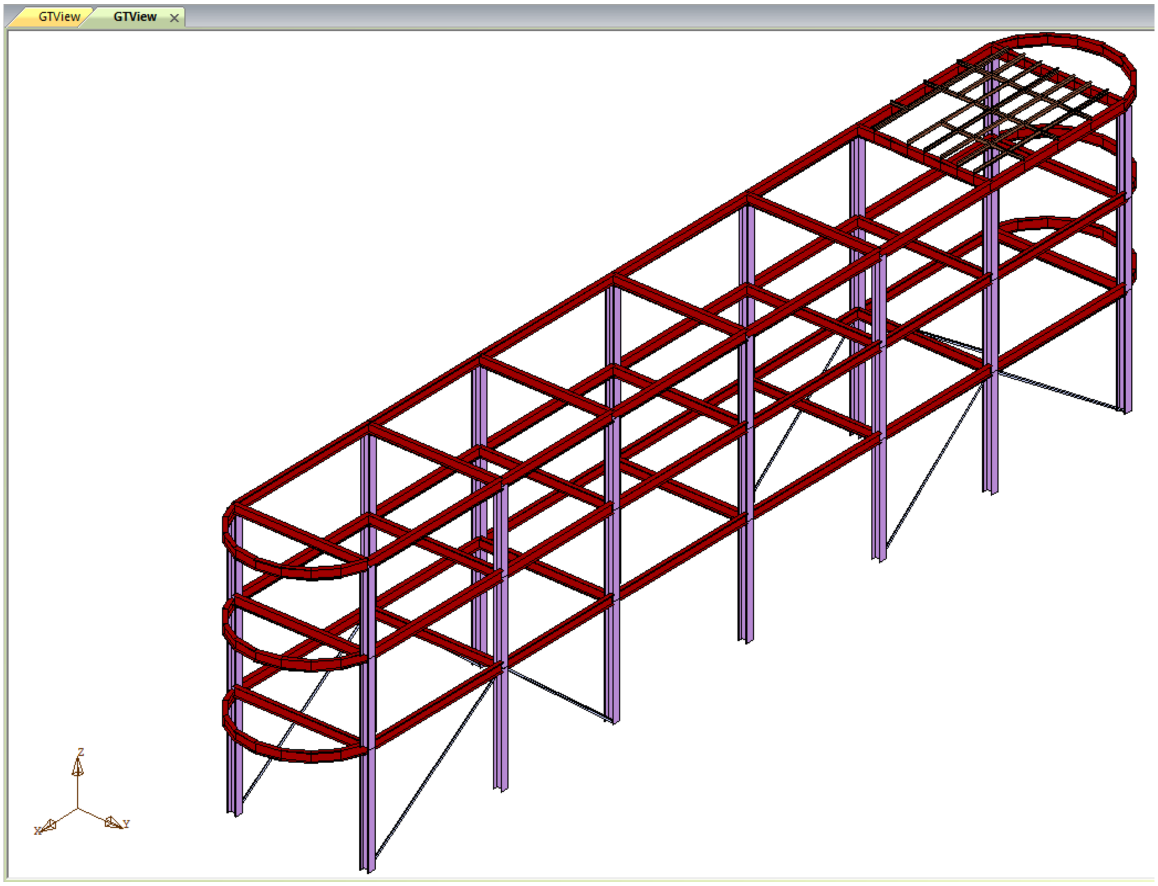
Stiffness analysis is automaticaly performed and DBX result files are automaticaly created.

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)

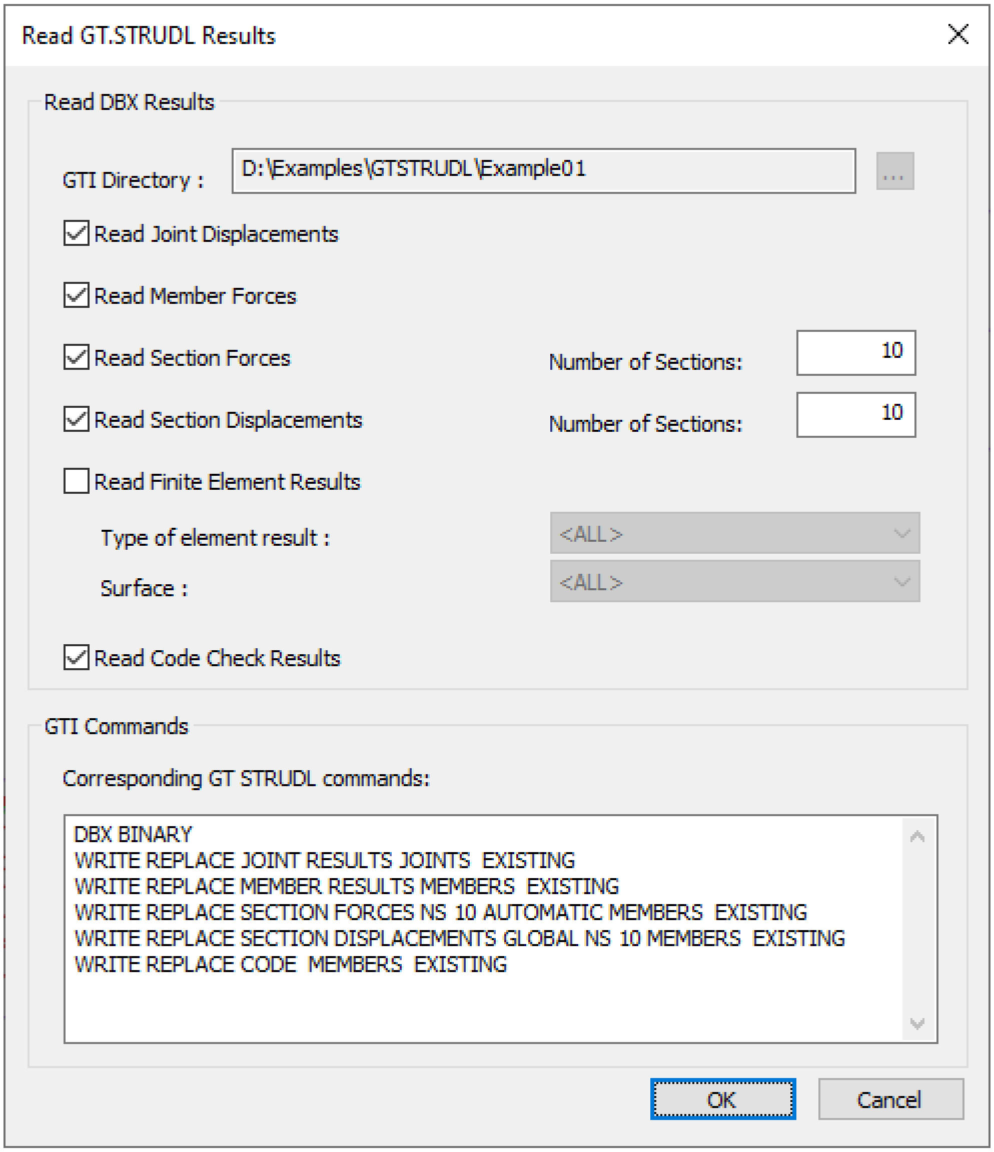
```
PARAMETERS

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.

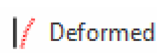


Step #51. Read Results from GT STRUDL: In CAD Modeler, click on the icon Read Results and the Read GT.STRUDL Results dialog appears. Check all options except “Read Finite Element Results” as shown below and press OK.

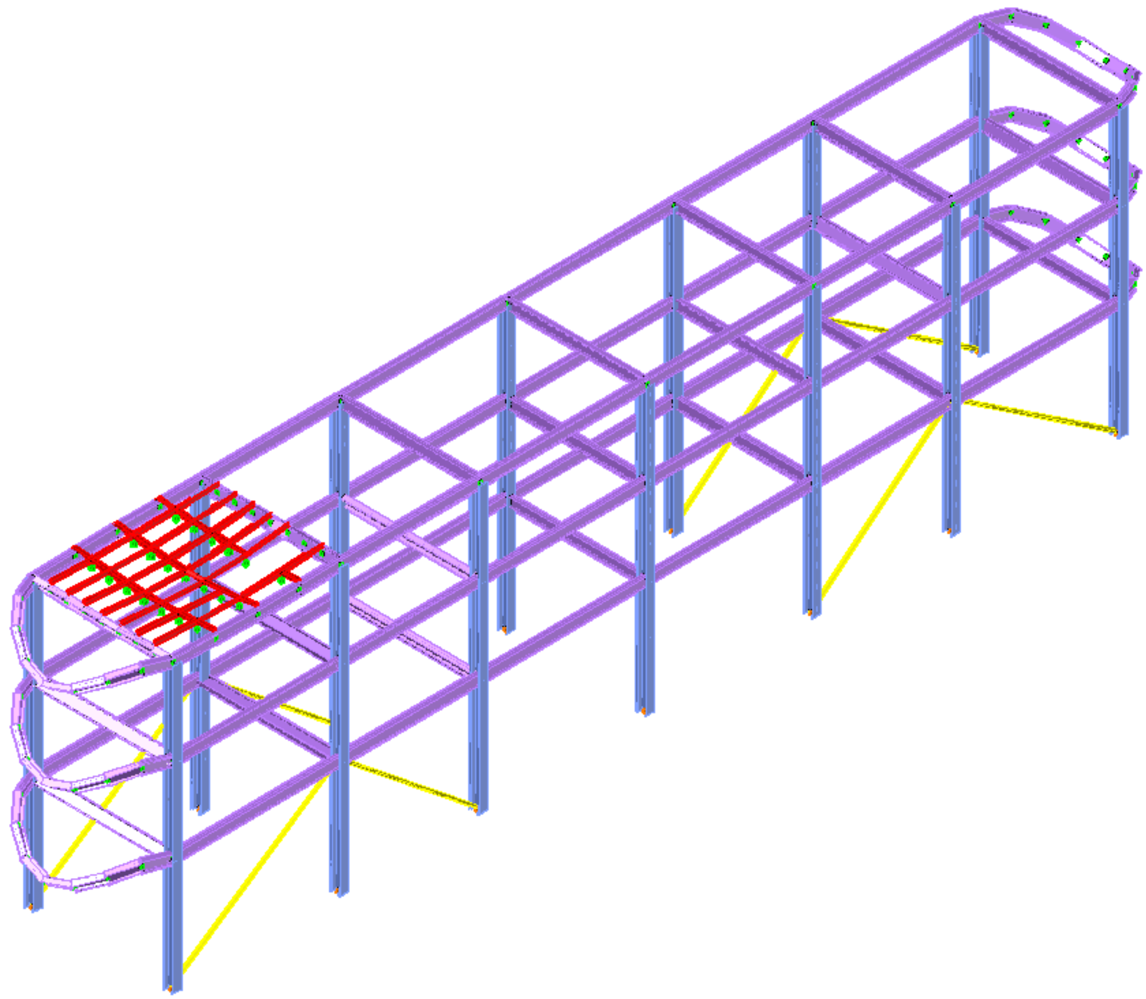


Note: You should get an information message “Results Loaded Successfully”

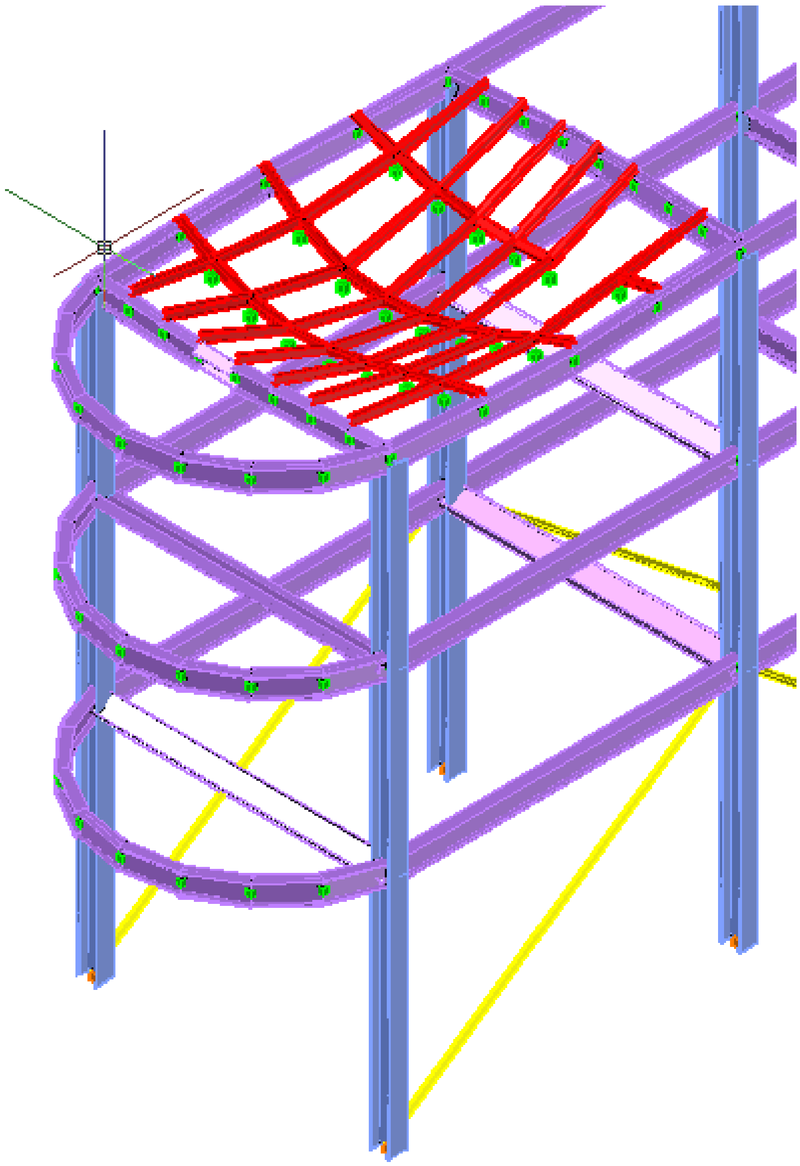
3.15. Display Results

Step #52. Display Deformed Model: on the Menu Bar, click on 

(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:



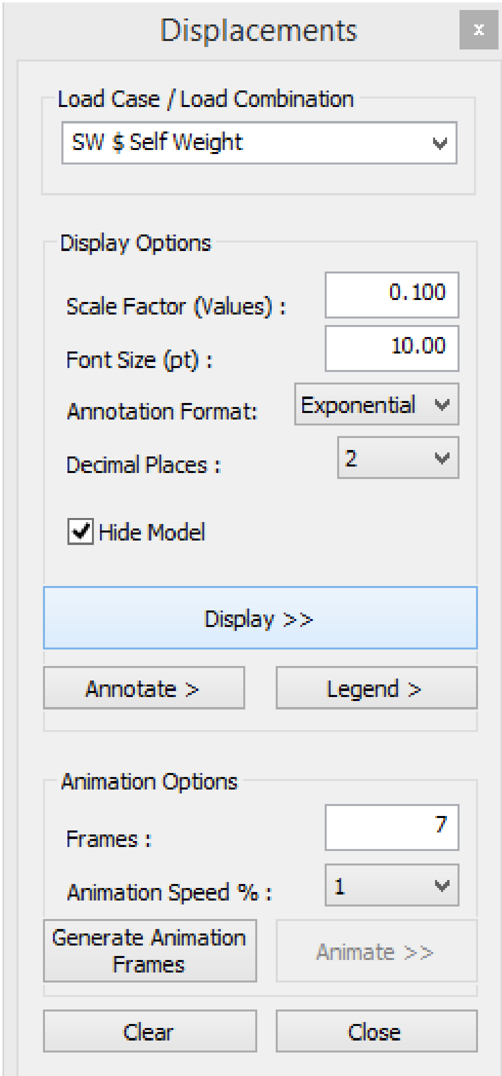
Note: You can Annotate the Joint displacements by clicking on the Annotate Displacements icon, that is located under the Deformed in the ribbon and the selecting a joint and annotation possition.

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

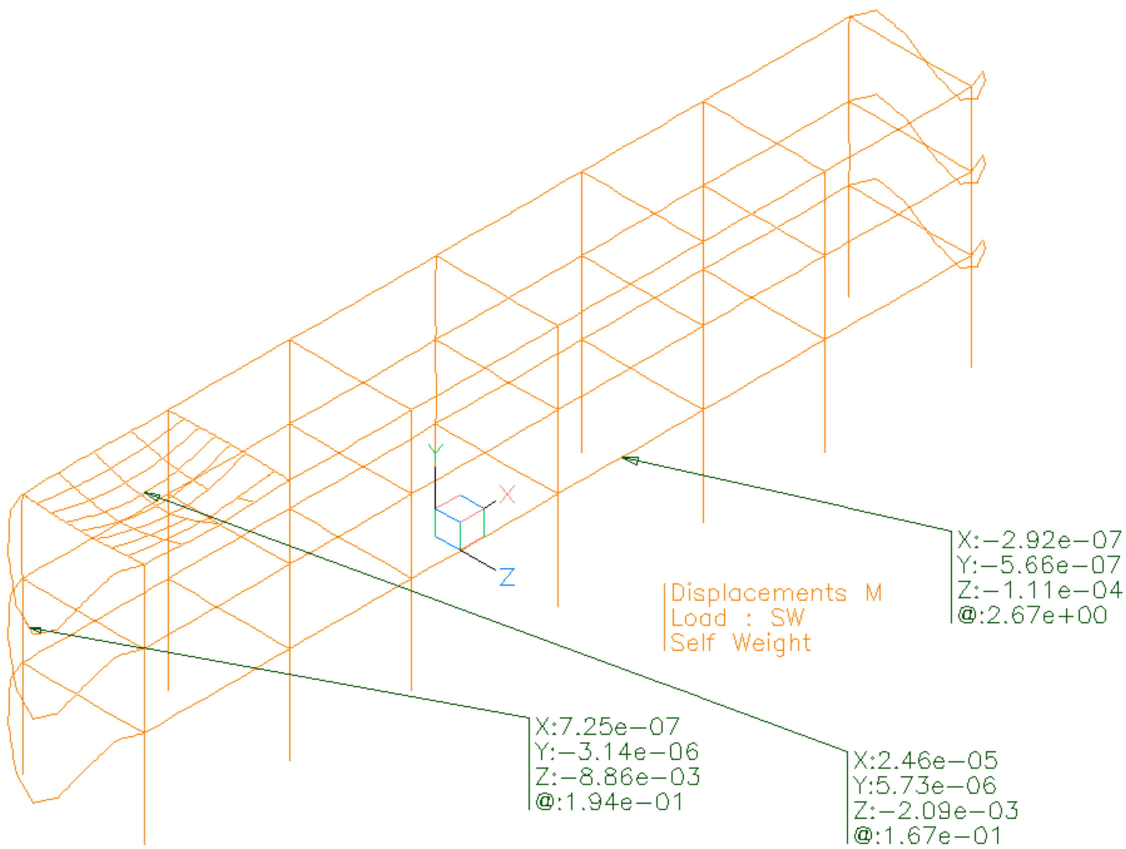


- *0.1* as Scale Factor
- *10.00* as Font Size (default)
- Annotation Format: Exponential
- *Check Hide Model*

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.

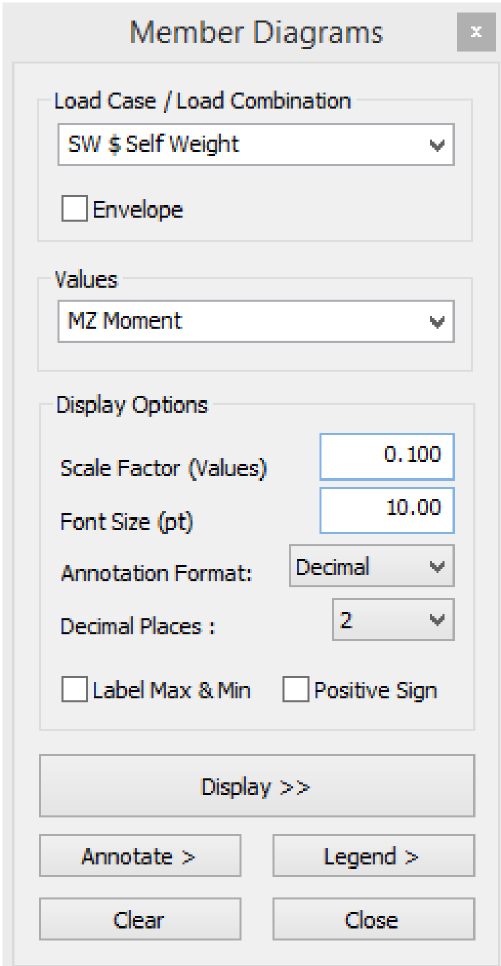


In order to display the animation press “Generate Animation Frames” and then “Animate >>”. To terminate the animation press “Stop” button.

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

Step #54. Display Member Diagrams: Click on 

(ribbon tab “GTS Display”).



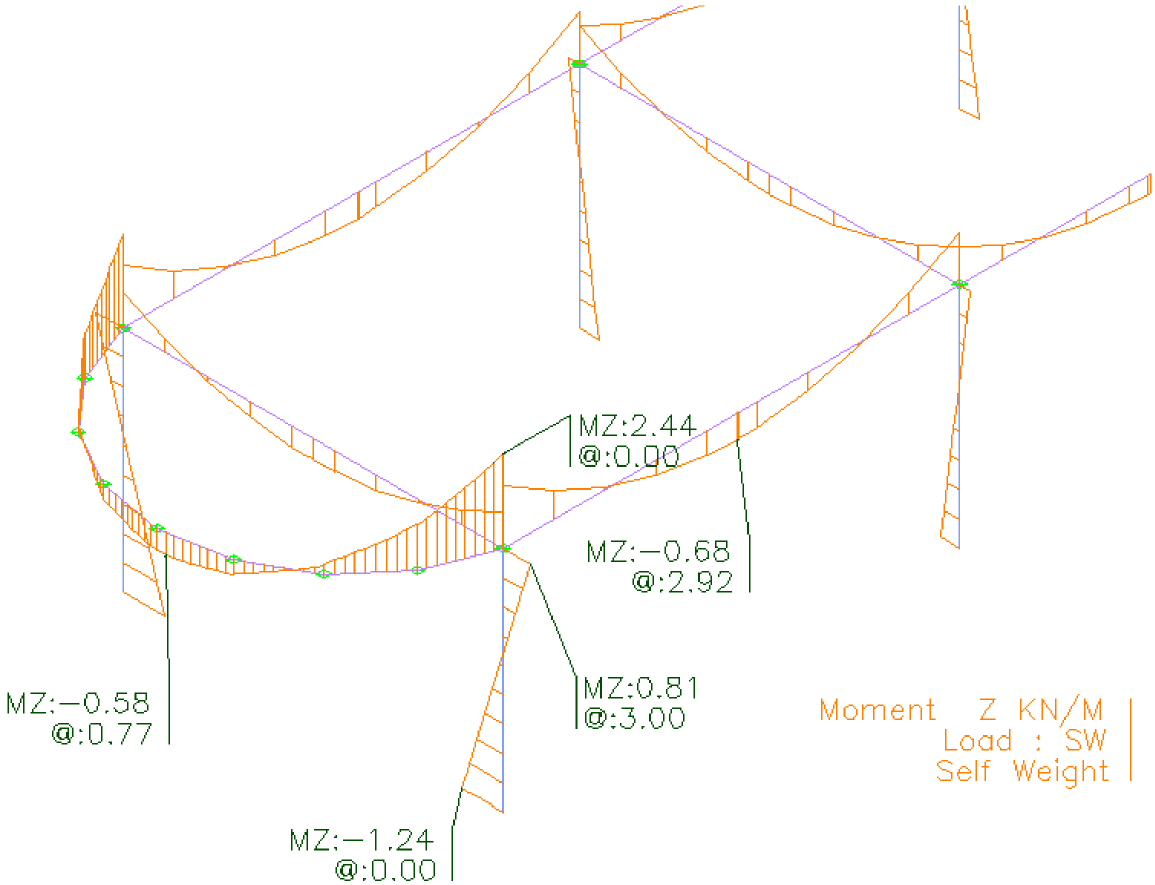
Select:

- *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.

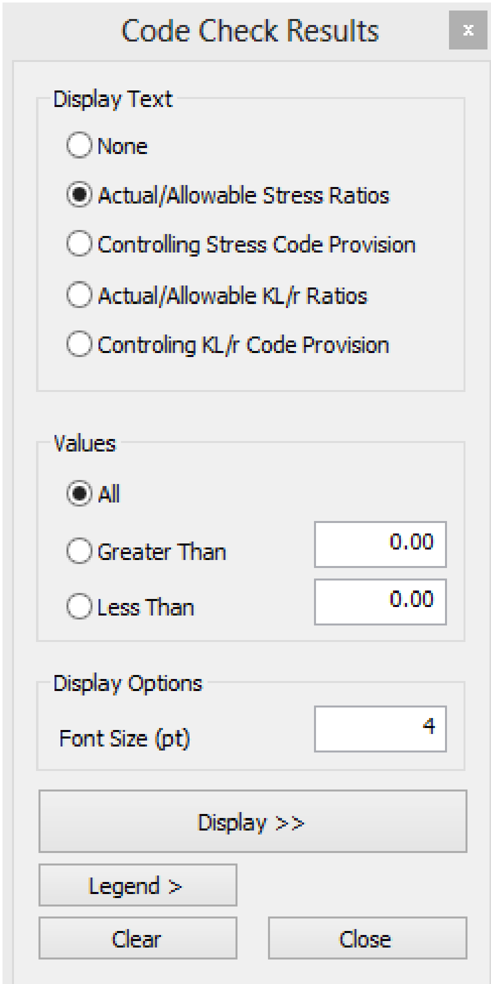
Press “Annotate >” and click on any part of the yellow MZ Moment curve and then at the position that you want the annotation to be displayed.

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.



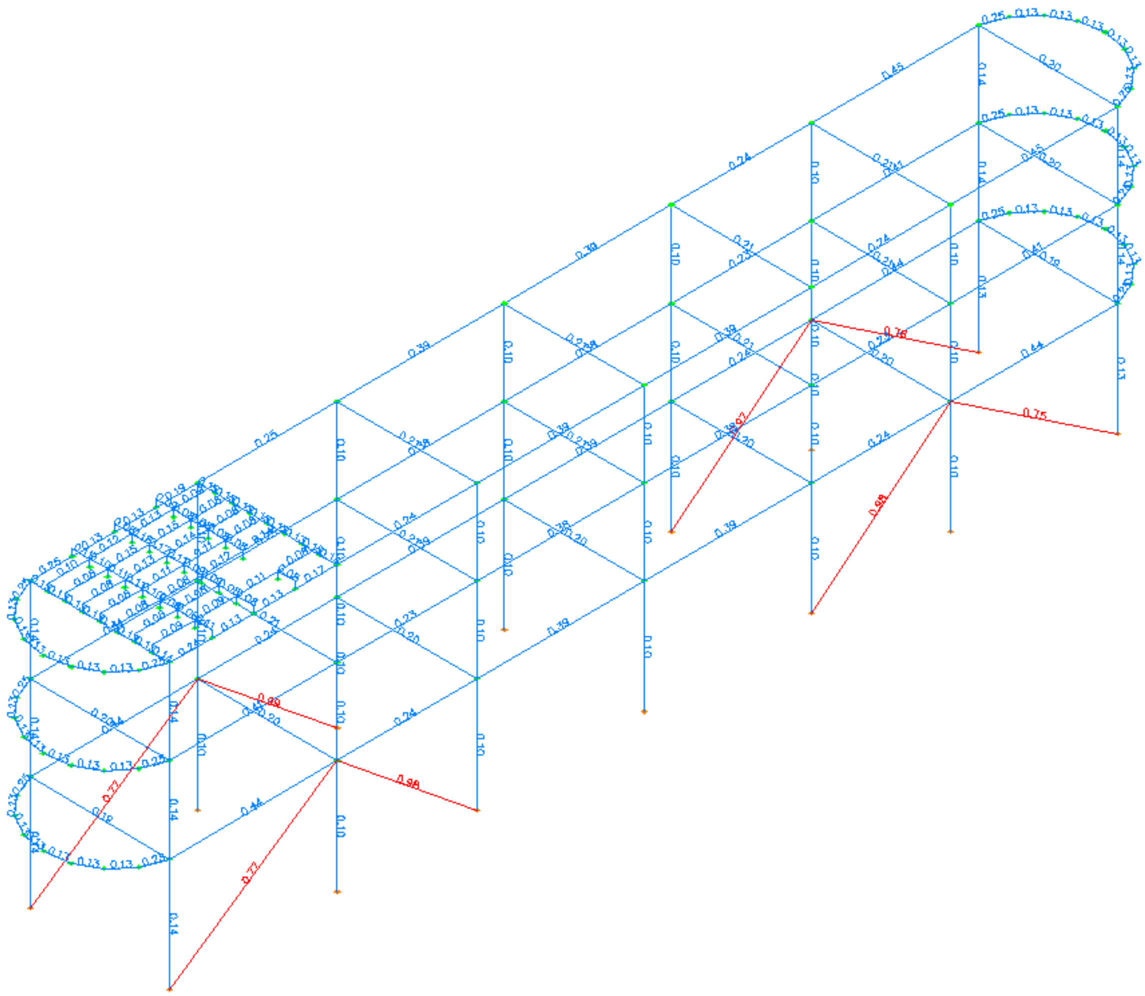
Select:

- *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”.

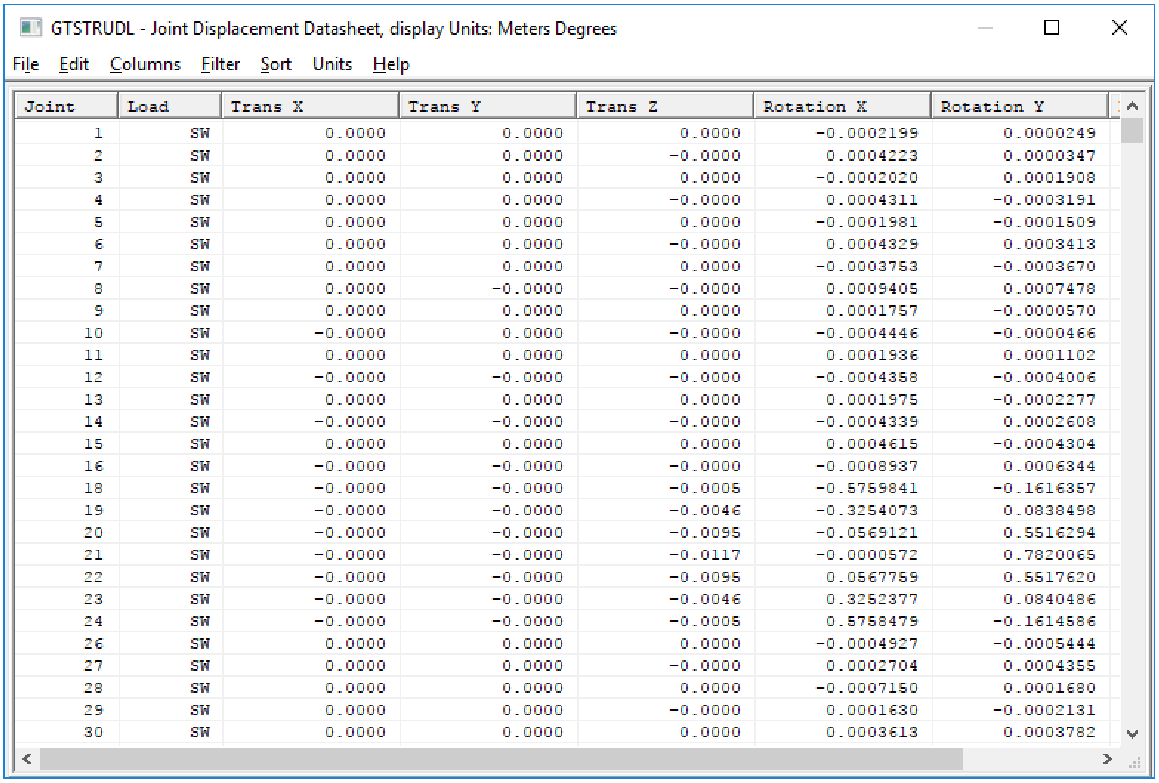
.



3.16. Results Datasheets

Step #56. Except from 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.

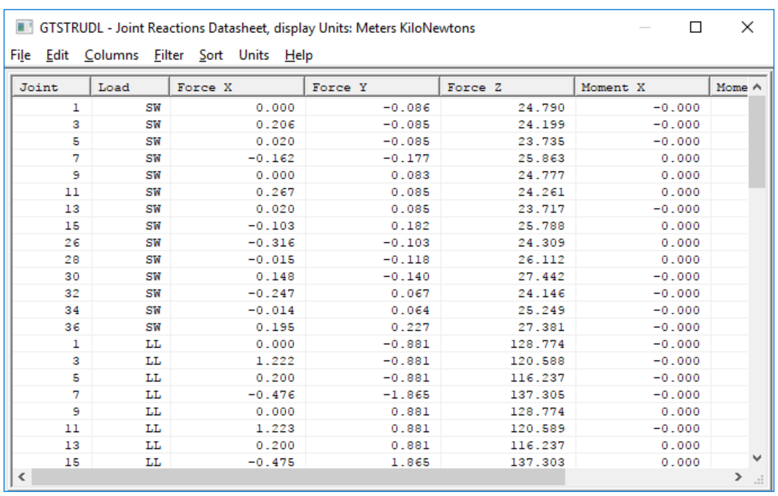


GTSTRUDL - Joint Displacement Datasheet, display Units: Meters Degrees

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	-0.0002199	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	Mome
1	SW	0.000	-0.086	24.790	-0.000	
3	SW	0.206	-0.085	24.199	-0.000	
5	SW	0.020	-0.085	23.735	-0.000	
7	SW	-0.162	-0.177	25.863	0.000	
9	SW	0.000	0.083	24.777	0.000	
11	SW	0.267	0.085	24.261	0.000	
13	SW	0.020	0.085	23.717	-0.000	
15	SW	-0.103	0.182	25.788	0.000	
26	SW	-0.316	-0.103	24.309	0.000	
28	SW	-0.015	-0.118	26.112	0.000	
30	SW	0.148	-0.140	27.442	-0.000	
32	SW	-0.247	0.067	24.146	-0.000	
34	SW	-0.014	0.064	25.249	-0.000	
36	SW	0.195	0.227	27.381	-0.000	
1	LL	0.000	-0.881	128.774	-0.000	
3	LL	1.222	-0.881	120.588	-0.000	
5	LL	0.200	-0.881	116.237	-0.000	
7	LL	-0.476	-1.865	137.305	-0.000	
9	LL	0.000	0.881	128.774	0.000	
11	LL	1.223	0.881	120.589	-0.000	
13	LL	0.200	0.881	116.237	0.000	
15	LL	-0.475	1.865	137.303	0.000	

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

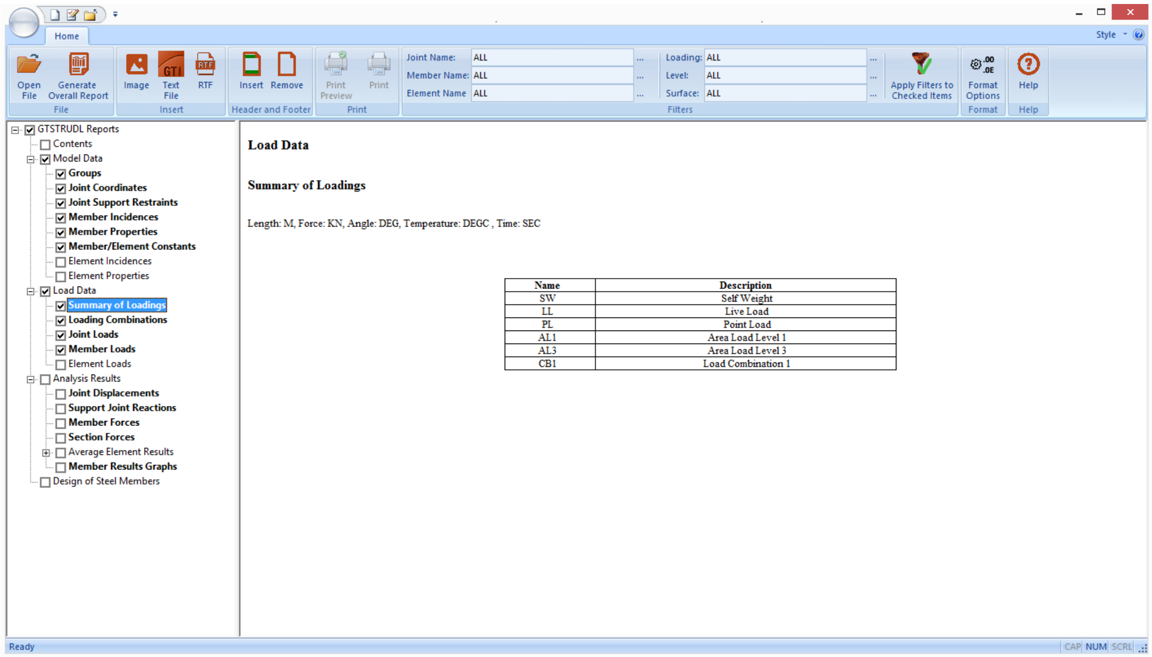
GT STRUDL Code Check Results

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/r	0.766	5.5.1	3.395	KL/r	60x60x5
149	FAIL	CB1	6.403	3.015	KL/r	0.981	5.5.1	3.015	KL/r	60x60x5
150	FAIL	CB1	6.403	3.015	KL/r	0.977	5.5.1	3.015	KL/r	60x60x5
151	FAIL	CB1	7.211	3.395	KL/r	0.751	5.5.1	3.395	KL/r	60x60x5
152	FAIL	CB1	7.211	3.395	KL/r	0.768	5.5.1	3.395	KL/r	60x60x5
153	FAIL	CB1	6.403	3.015	KL/r	0.992	5.5.1	3.015	KL/r	60x60x5
154	FAIL	CB1	6.403	3.015	KL/r	0.973	5.5.1	3.015	KL/r	60x60x5
155	FAIL	CB1	7.211	3.395	KL/r	0.755	5.5.1	3.395	KL/r	60x60x5
1	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
2	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
3	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
4	Pass	CB1	4.000	0.293	KL/r	0.134	5.5.4(2)	0.293	KL/r	HE320B
5	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
6	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
7	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
8	Pass	CB1	4.000	0.293	KL/r	0.134	5.5.4(2)	0.293	KL/r	HE320B
9	Pass	CB1	0.000	0.939	KL/r	0.391	5.5.4(2)	0.939	KL/r	IPE330
10	Pass	CB1	0.000	0.782	KL/r	0.243	5.5.4(2)	0.782	KL/r	IPE330
11	Pass	CB1	6.000	0.939	KL/r	0.439	5.5.4(2)	0.939	KL/r	IPE330
12	Pass	CB1	0.000	0.939	KL/r	0.391	5.5.4(2)	0.939	KL/r	IPE330
13	Pass	CB1	0.000	0.782	KL/r	0.243	5.5.4(2)	0.782	KL/r	IPE330
14	Pass	CB1	6.000	0.939	KL/r	0.439	5.5.4(2)	0.939	KL/r	IPE330
15	Pass	CB1	0.000	0.782	KL/r	0.205	5.5.2	0.782	KL/r	IPE330
16	Pass	CB1	0.000	0.782	KL/r	0.205	5.5.2	0.782	KL/r	IPE330
17	Pass	CB1	5.000	0.205	5.5.2	0.205	5.5.2	0.000	********	IPE330
18	Pass	CB1	5.000	0.188	5.5.2	0.188	5.5.2	0.000	********	IPE330
19	Pass	CB1	0.000	0.249	5.5.3	0.249	5.5.3	0.000	********	IPE330
20	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	********	IPE330
21	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	********	IPE330
22	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	********	IPE330
23	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	********	IPE330
24	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	********	IPE330
25	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	********	IPE330
26	Pass	CB1	0.871	0.249	5.5.3	0.249	5.5.3	0.000	********	IPE330
27	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
28	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
29	Pass	CB1	4.000	0.293	KL/r	0.135	5.5.4(2)	0.293	KL/r	HE320B
30	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
31	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B

3.17. Report Builder

Step #57. Click on the icon Report Builder 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).

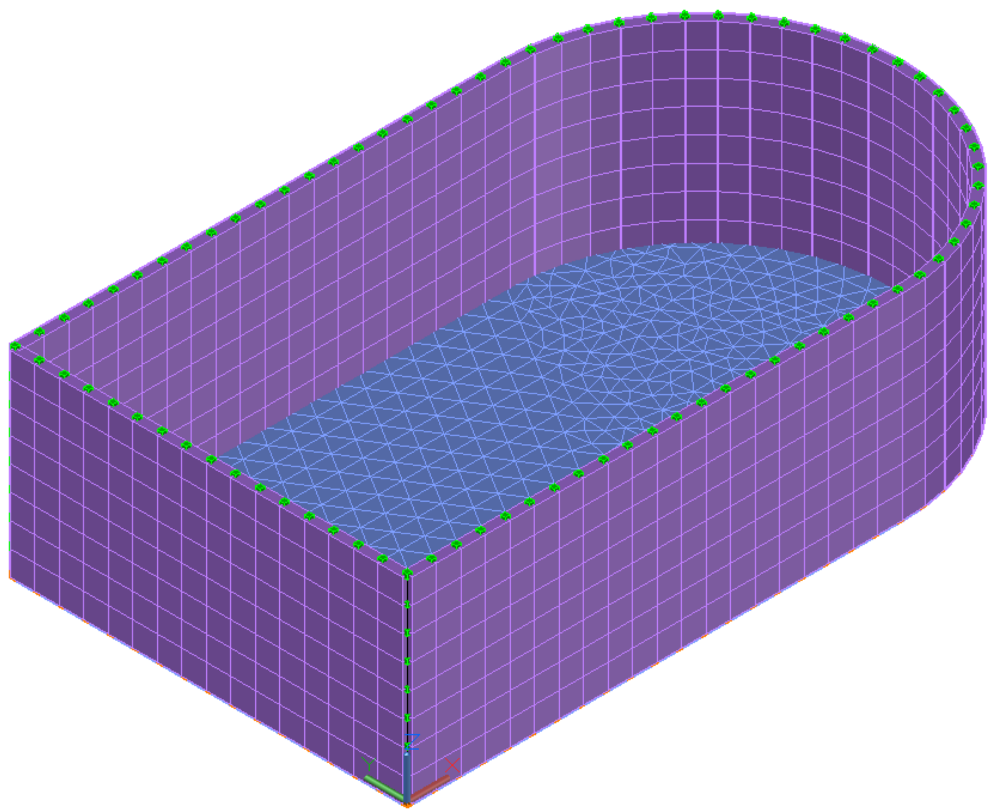


Name	Description
SW	Self Weight
LL	Live Load
PL	Point Load
AL1	Area Load Level 1
AL3	Area Load Level 3
CB1	Load Combination 1

4. Tutorial Example #2

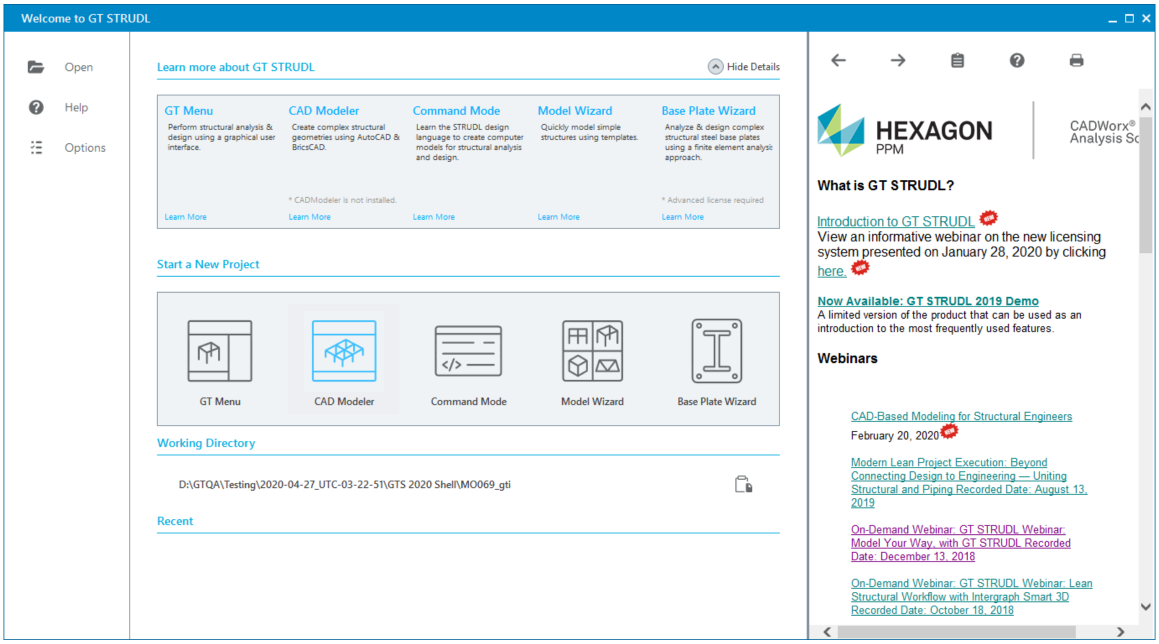
4.1. Introduction

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.



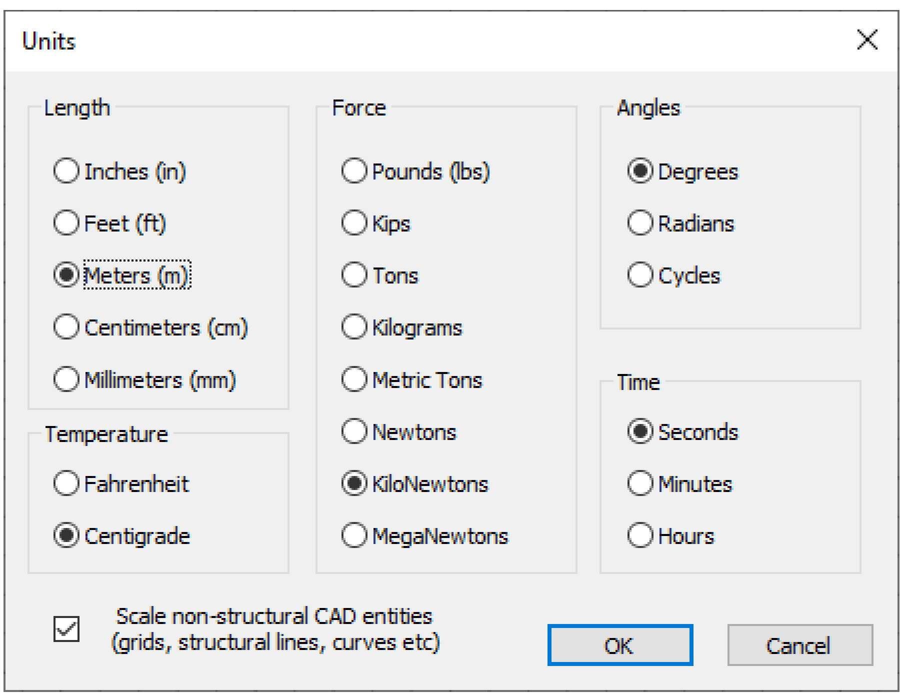
4.2. Open CAD Modeler and start working

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

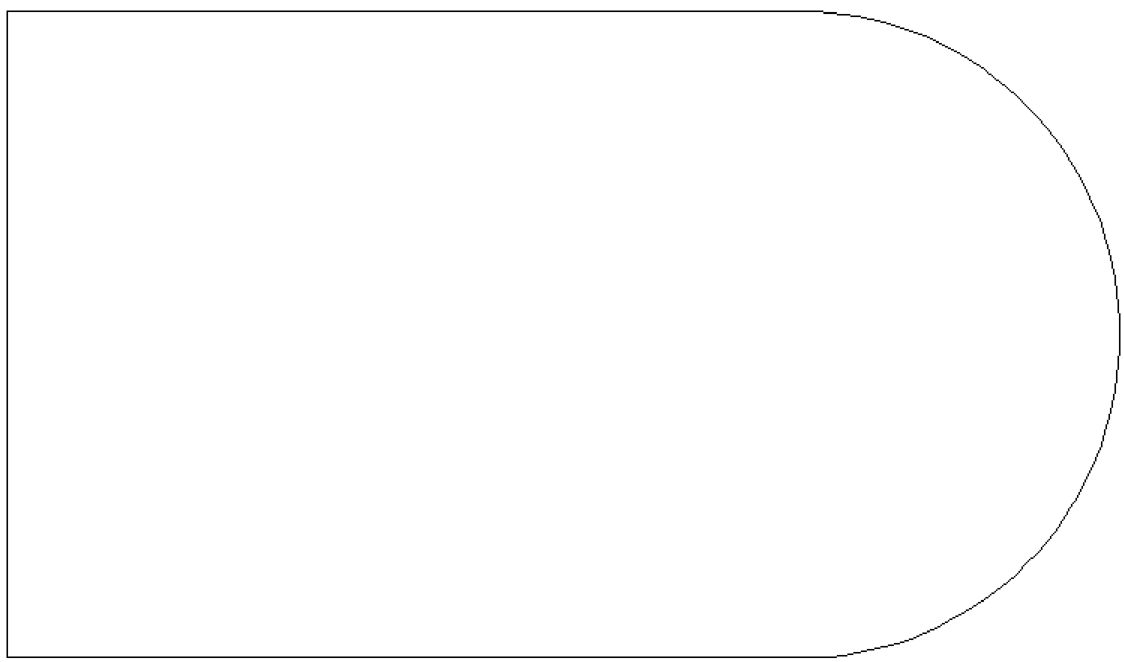
Step #2. Define the correct Units by pressing the icon Units and select *Meters (m)* and *KiloNewtons* in the *Units Form*.



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
```

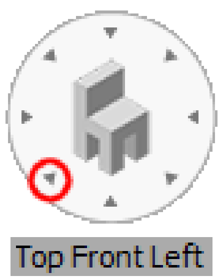
The polyline shown in the picture below is created after entering Zoom and Extents at the command prompt.



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.

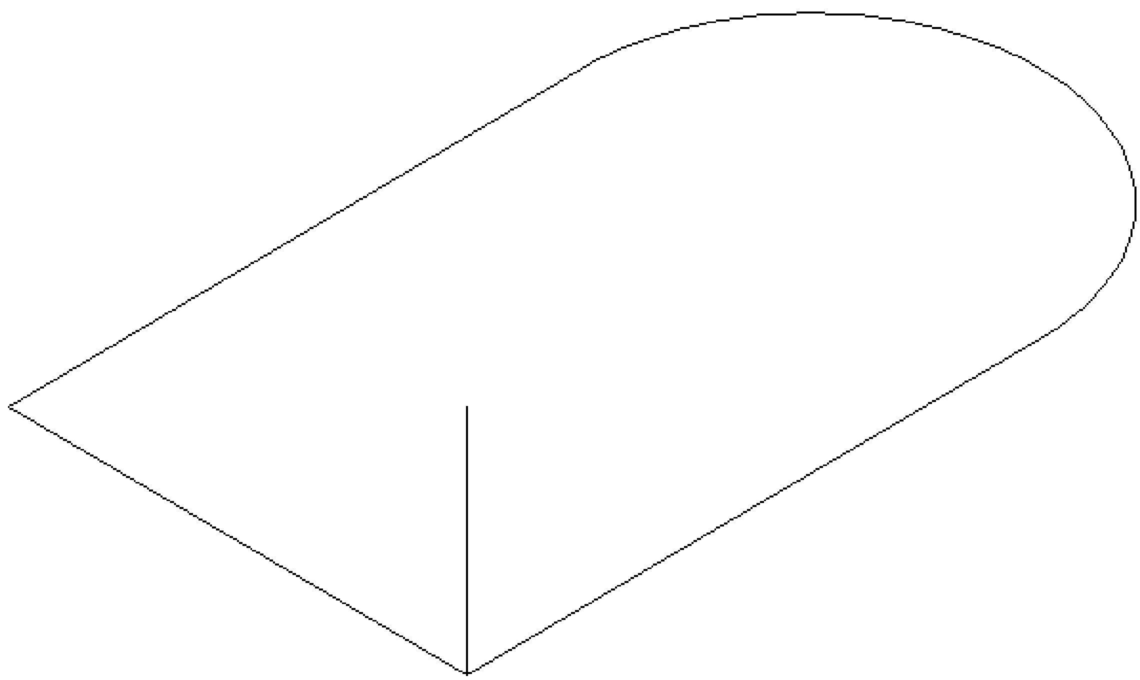
Type the following commands at the command prompt (each command is followed by an <ENTER>):

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



The line shown at the picture below is created.

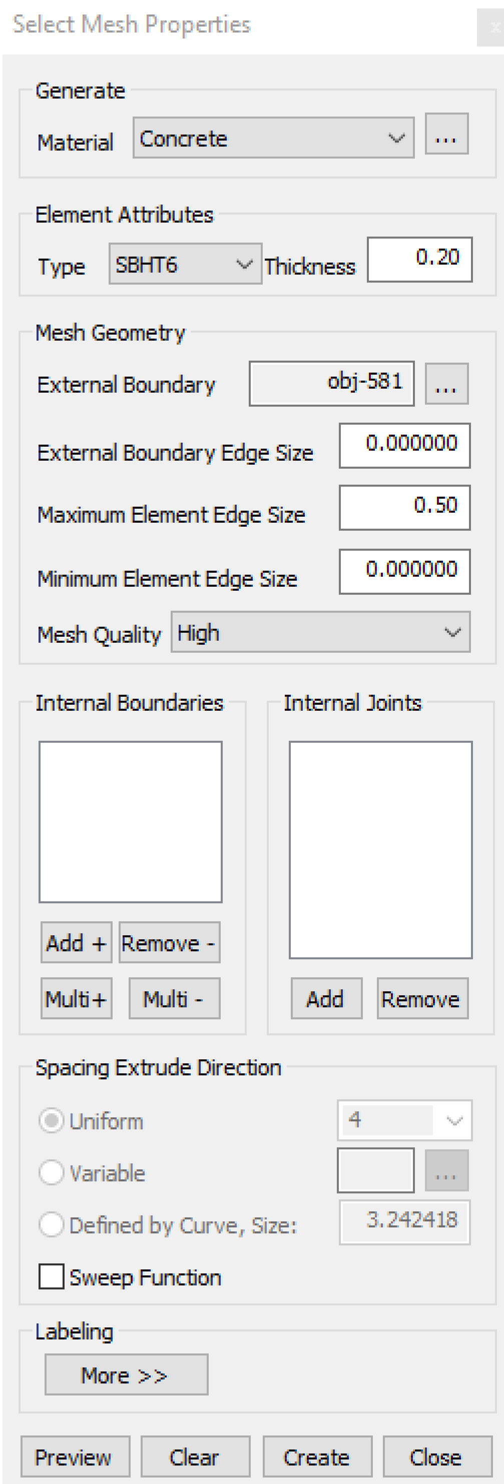
Click on the TOP icon of the AutoCAD's Cube / BricsCAD's Chair in order to switch back to floor plan view.



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 

, 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.



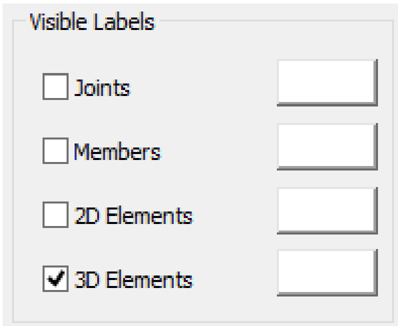
The Select Mesh Properties form appears where you have to enter:

- 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:

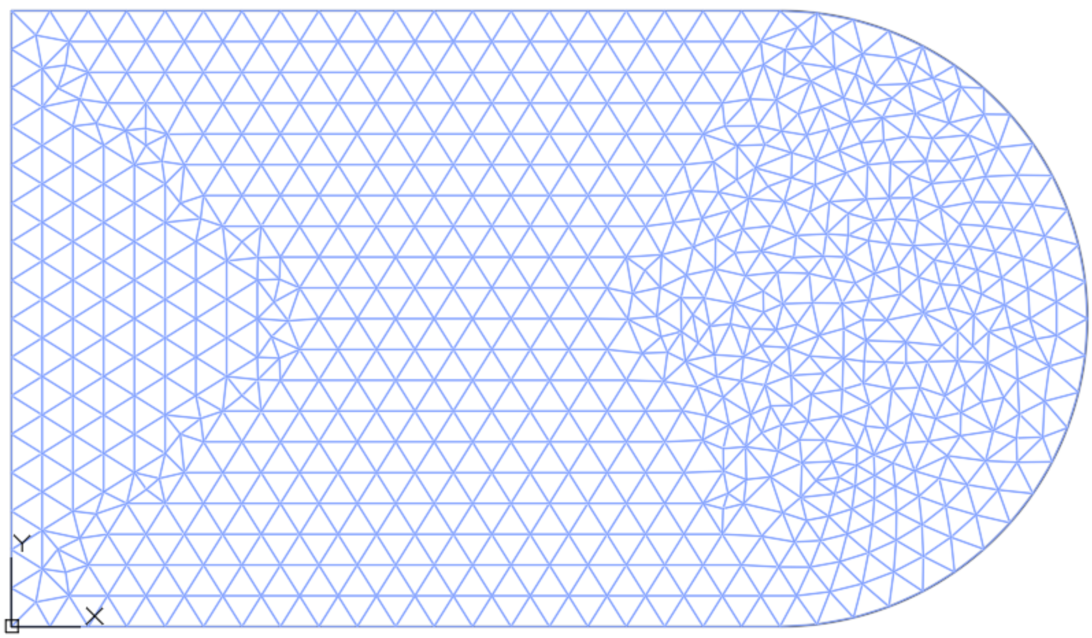


Click on the icon 

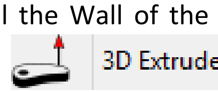
in the ribbon bar and then uncheck the Visible Labels option for Joints, Members and 2D Elements.

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

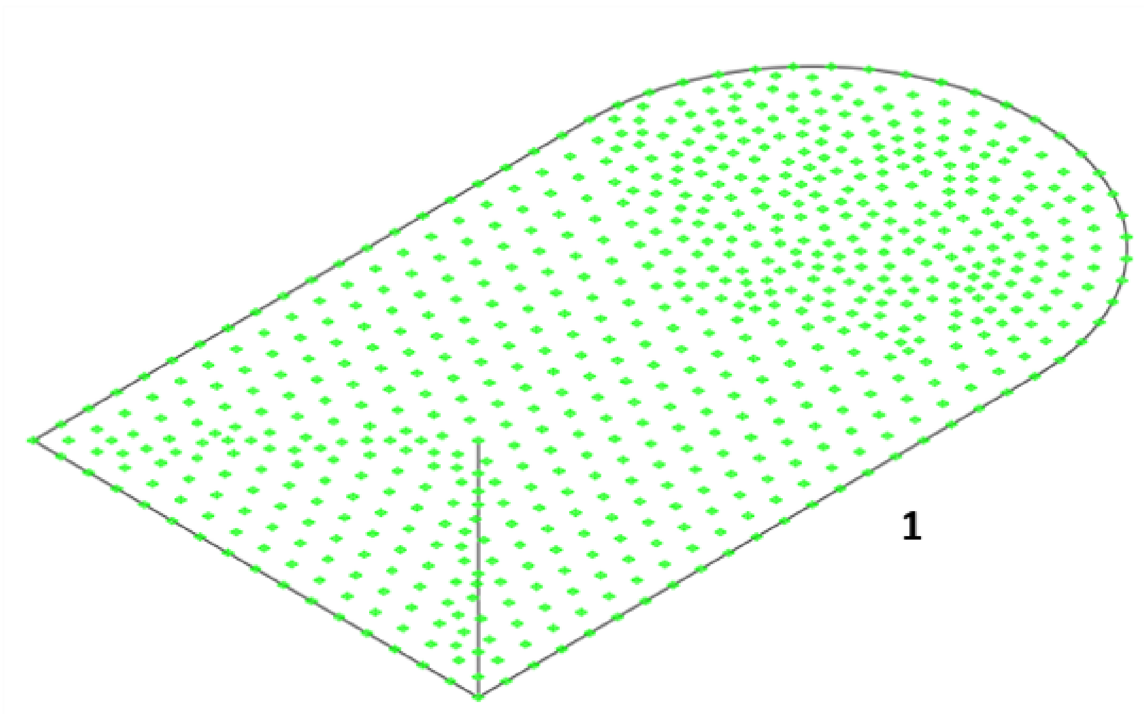
The finite element mesh, without labeling, is shown in the image below.



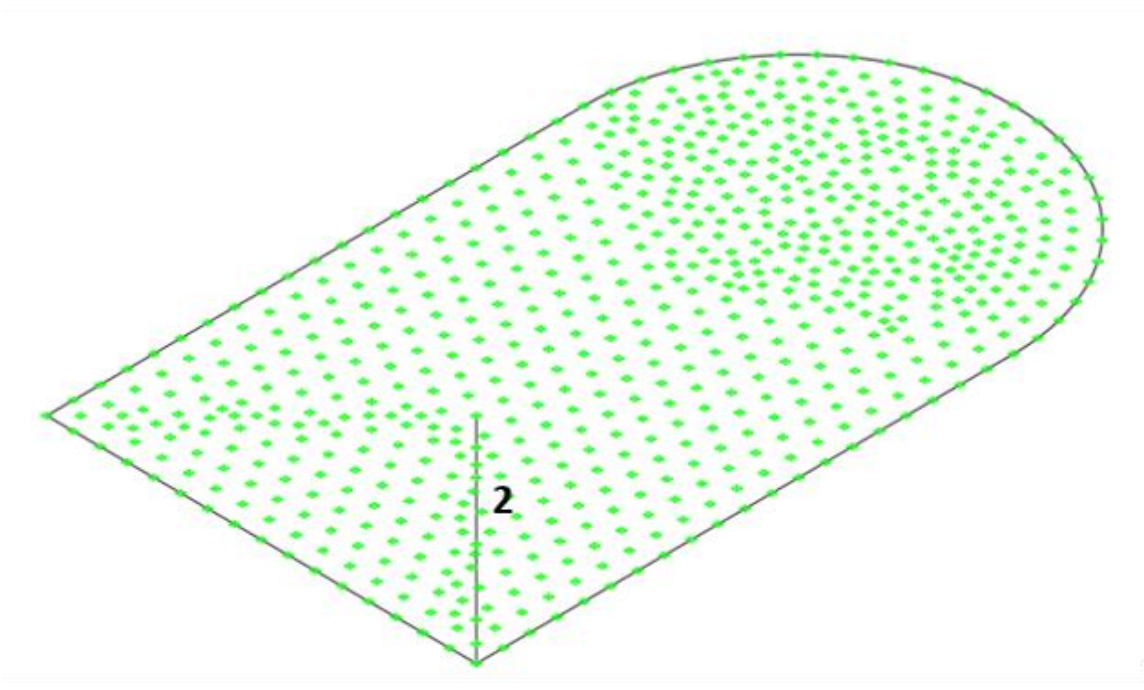
4.5. Create the walls of the tank

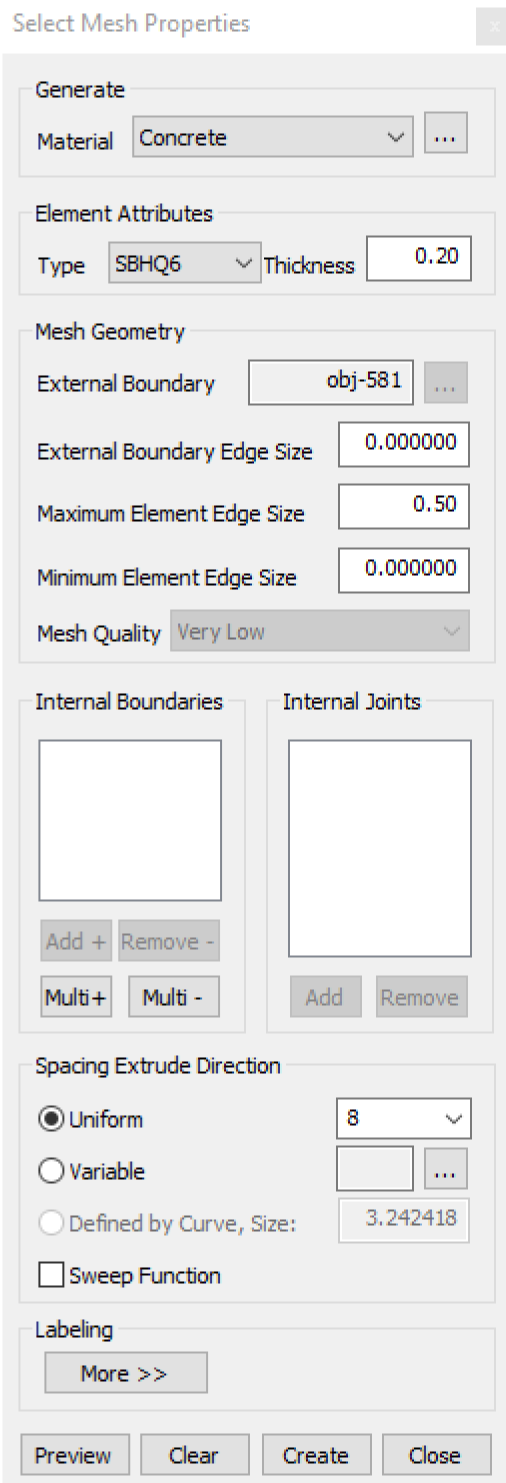
Step #7. Generate the finite elements that will model the Wall of the Tank by extruding the polyline: Switch to Isometric view and click on the icon 

, 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).





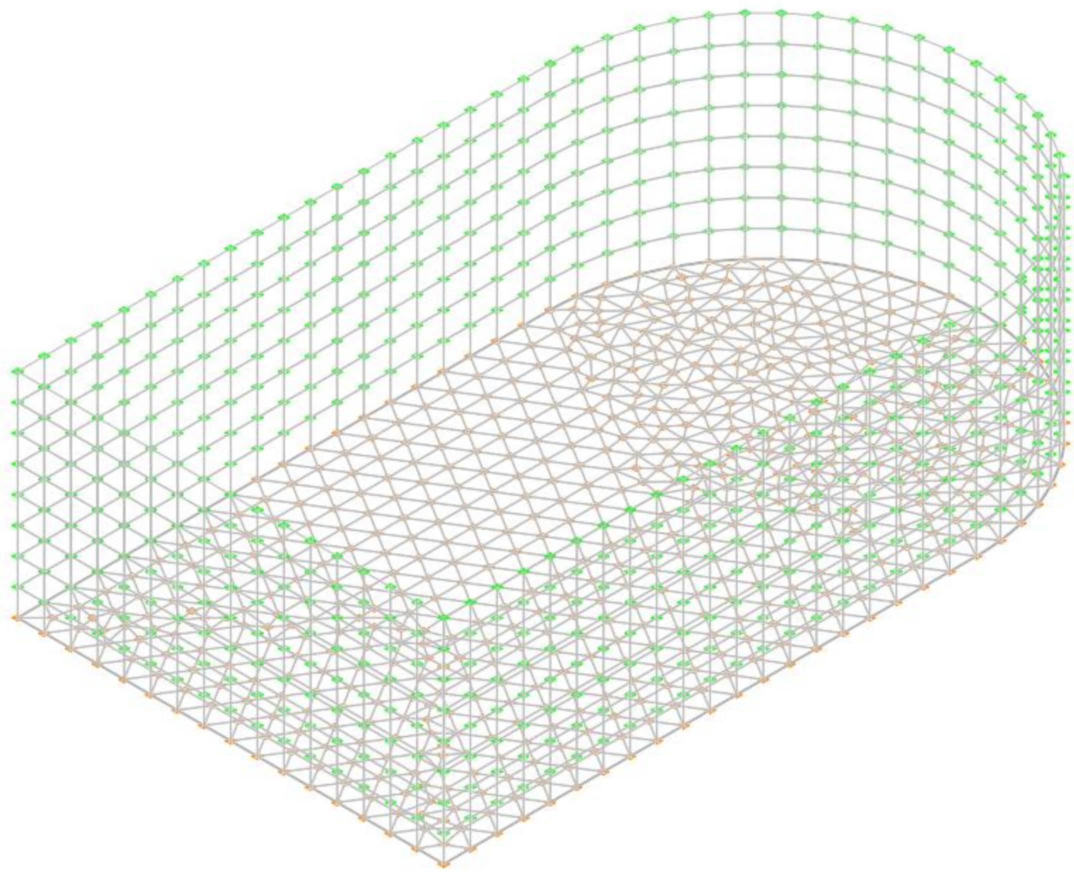
The Select Mesh Properties form appears where you have to enter:

- 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

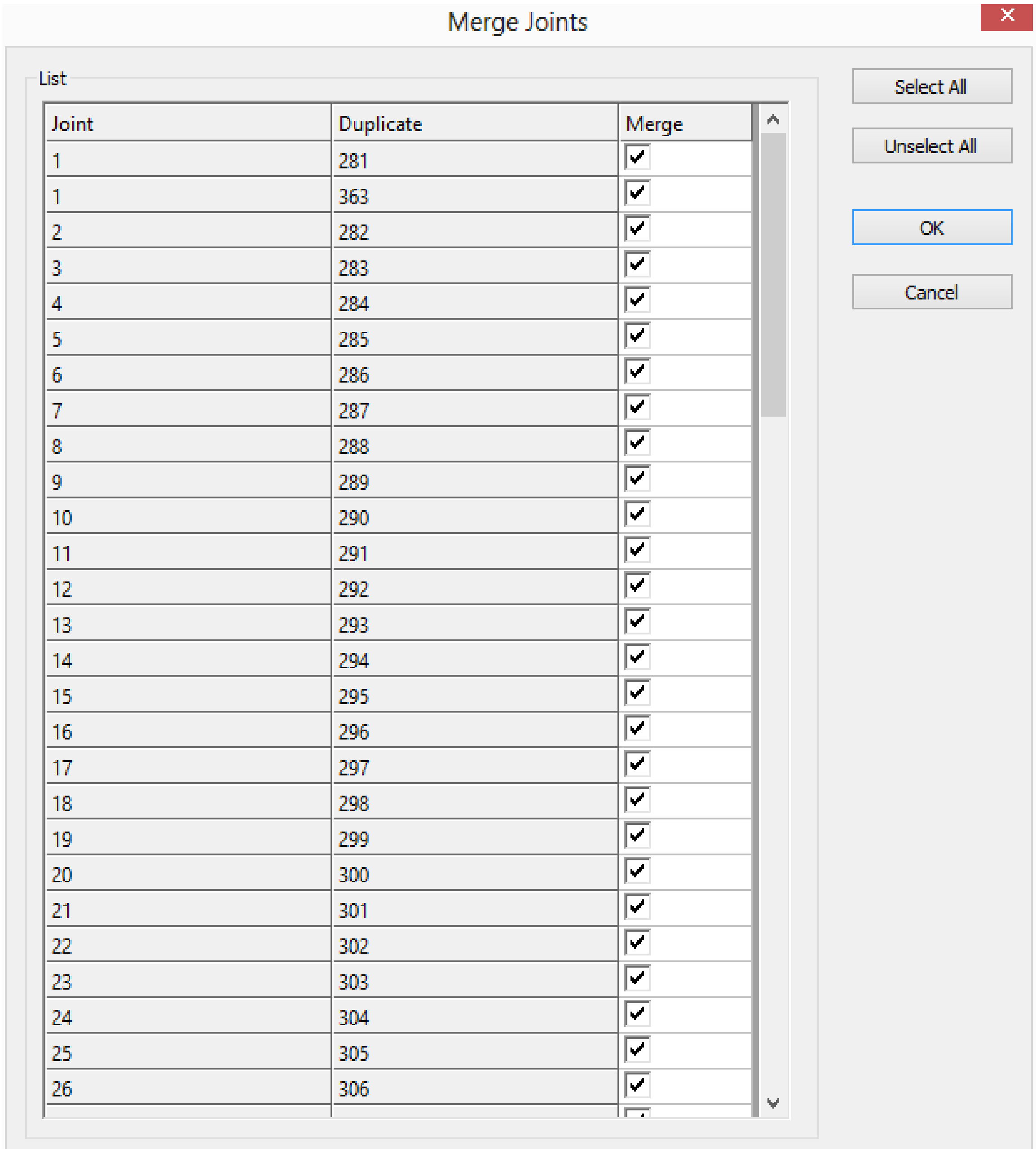
The finite element mesh is presented in the following image.



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.

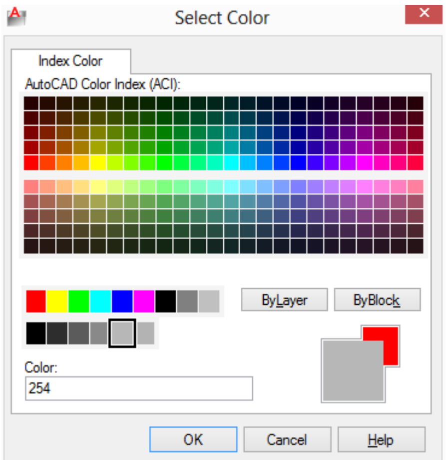
For the *Merge Tolerance <0.001000>*, just press <ENTER> to accept the default value.

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.

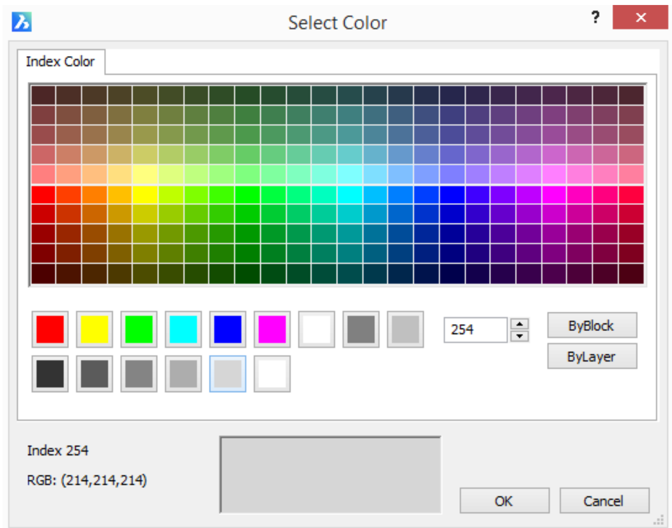


By entering the same command again for the 2nd time, you should get the notification that *0 duplicate joints found* .

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.



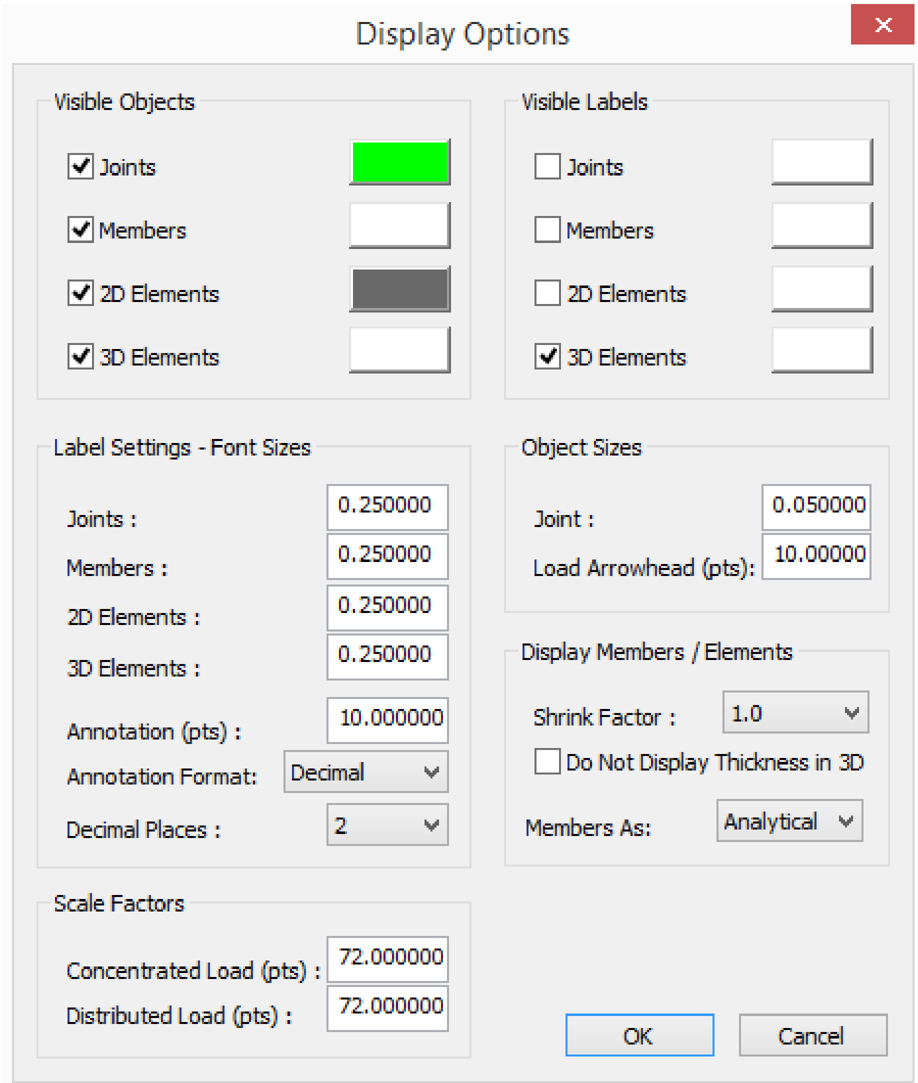
Select Color (AutoCAD)

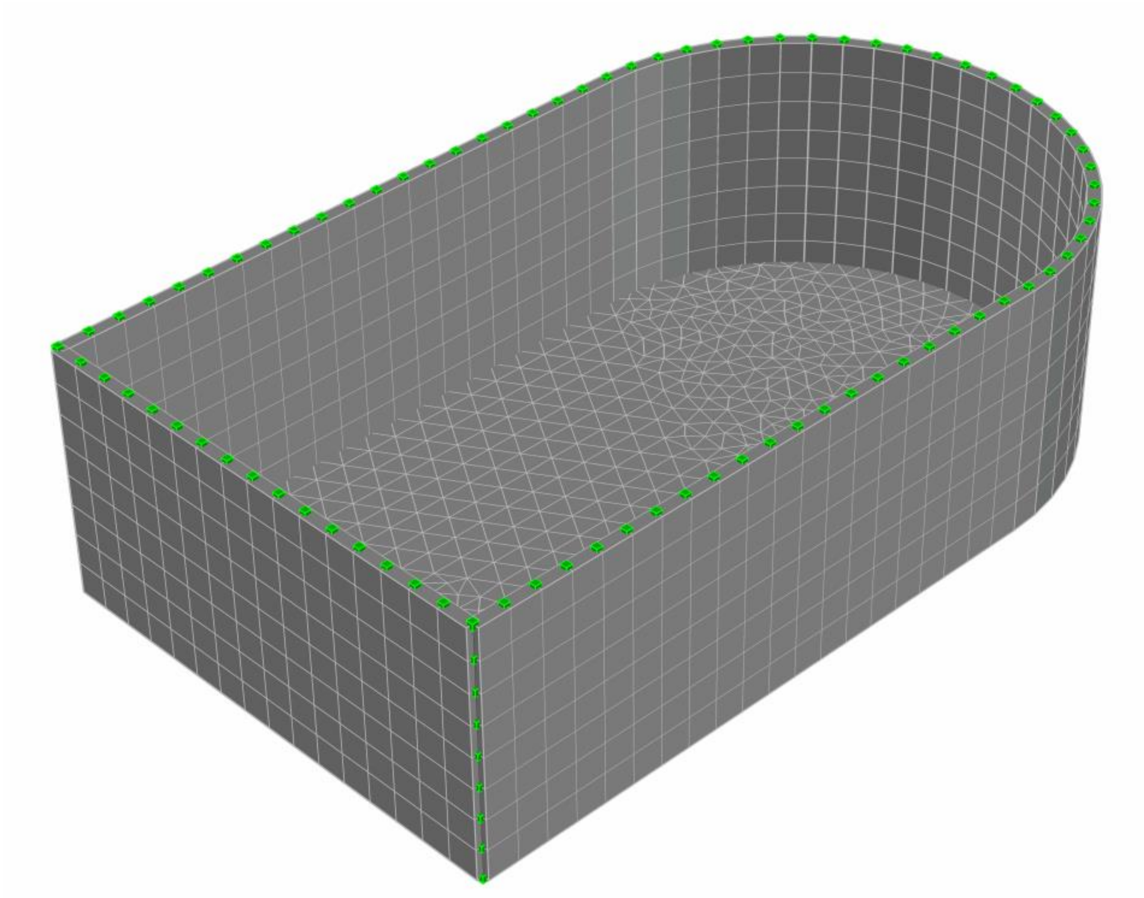


Select Color (BricsCAD)

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.

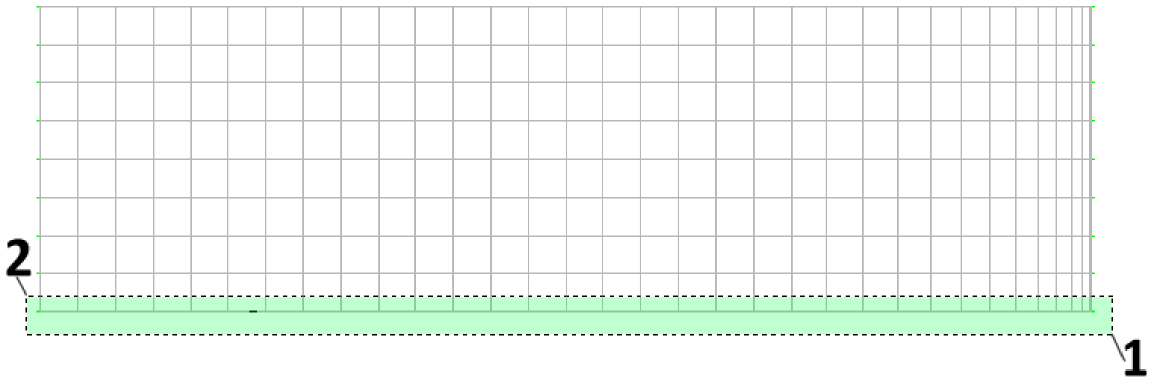
Step #10. Save your Model: In order to save your model, use AutoCAD's/BricsCAD's save command and store the DWG using any filename that you want.

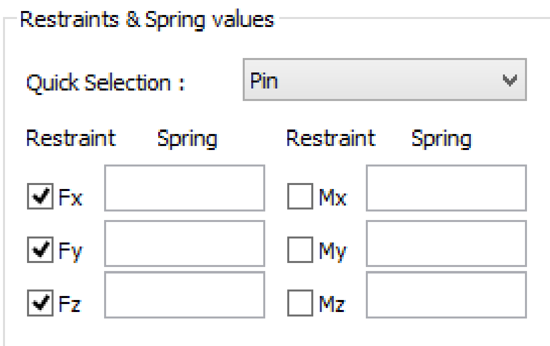
4.6. Create Supports

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.





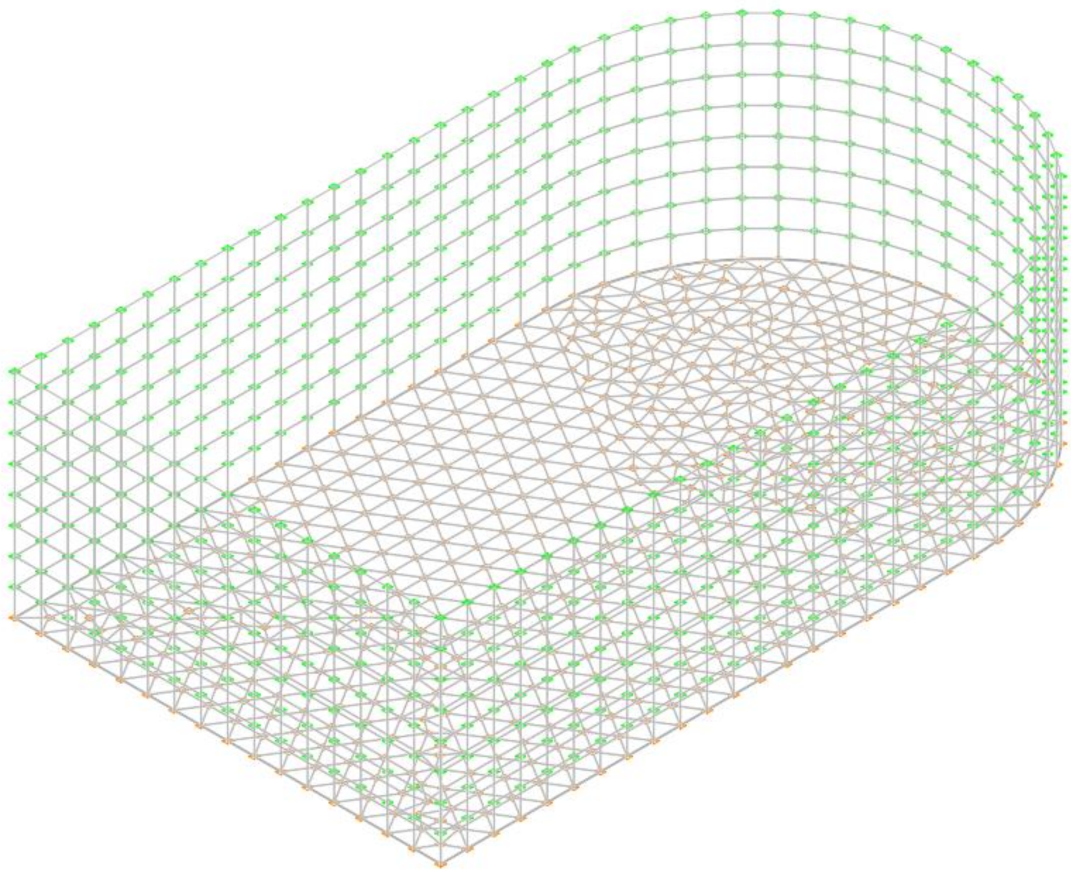
The Joint Properties [Multiple Selection] form appears.

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.

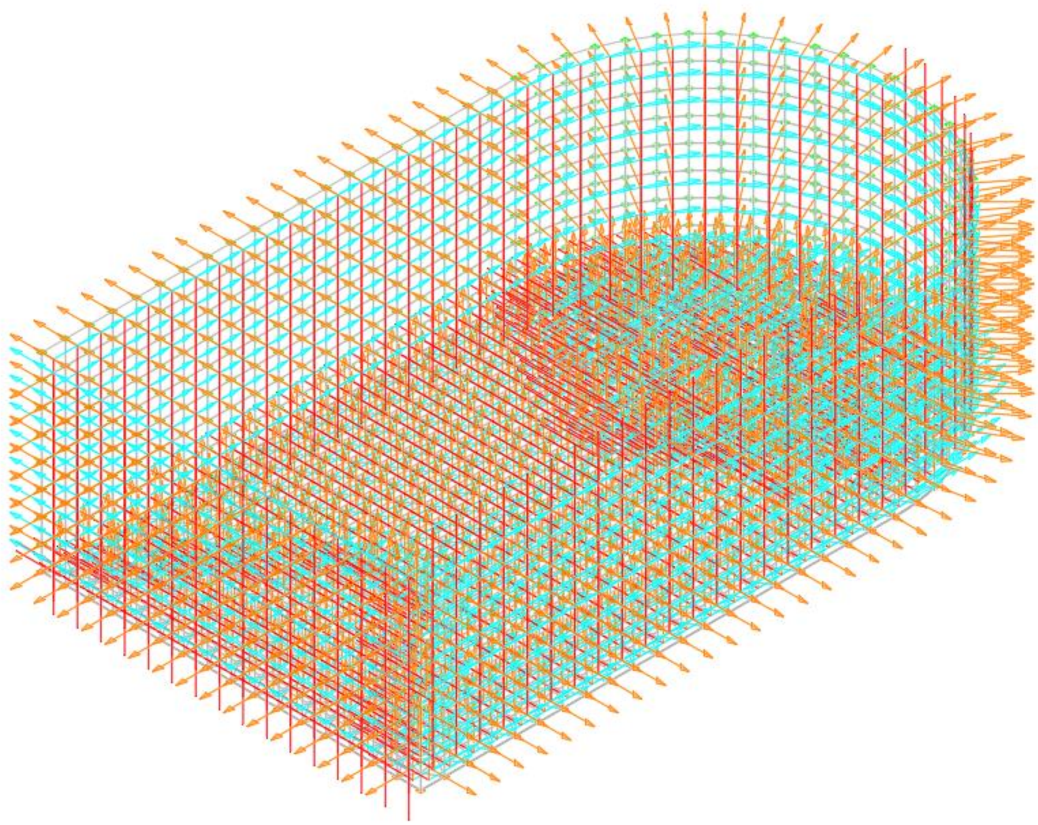
You should get the notification that *0 duplicate joints found* .

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).

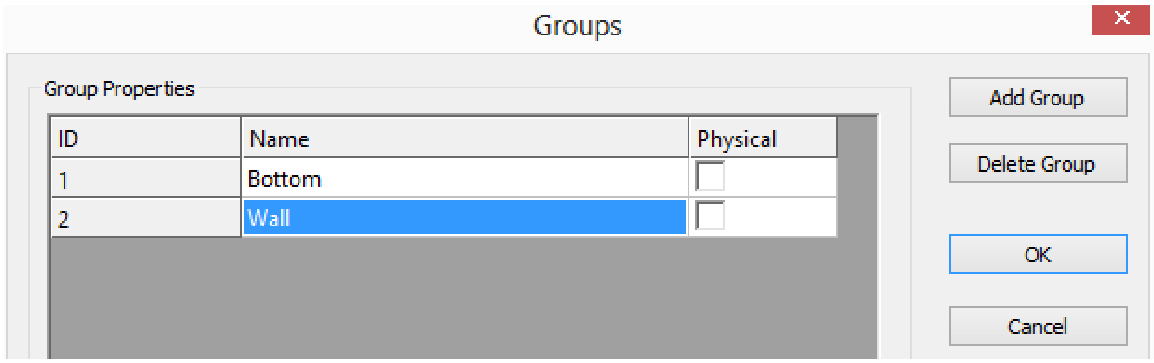


To clear the arrows select Clear from the GTS Display Ribbon Tab.

4.8. Define Groups

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

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



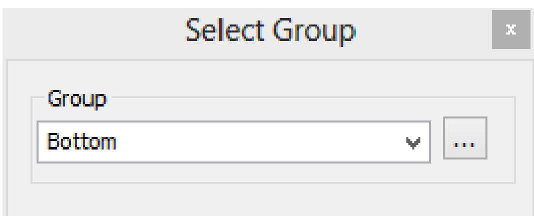
Press the *Add Group* button and enter `Bottom` as *Name* of the group.

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:

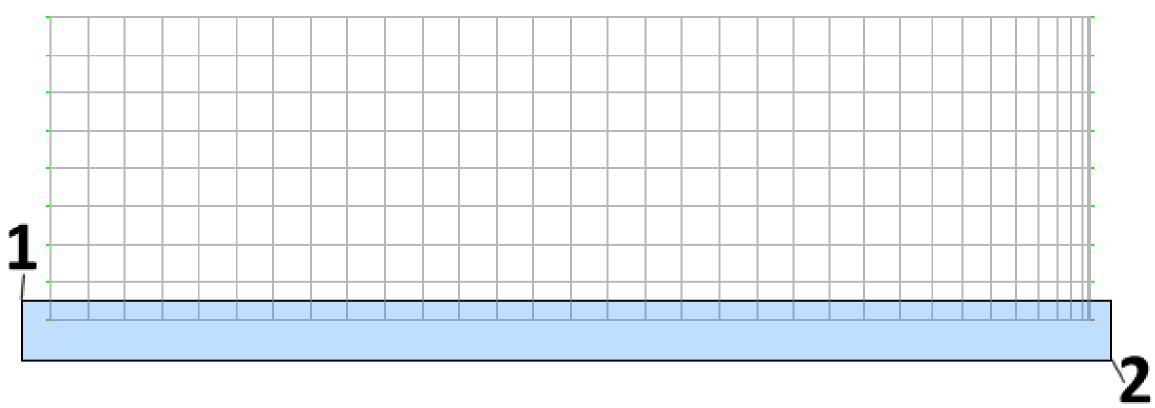
Switch to the FRONT View, by clicking on Front on AutoCAD's cube or BricsCAD's chair.



Click on the icon 

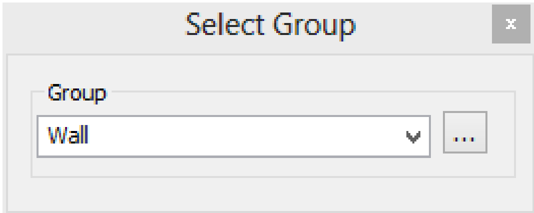
in the Group panel and make sure that the Group Bottom is selected as the active group.

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



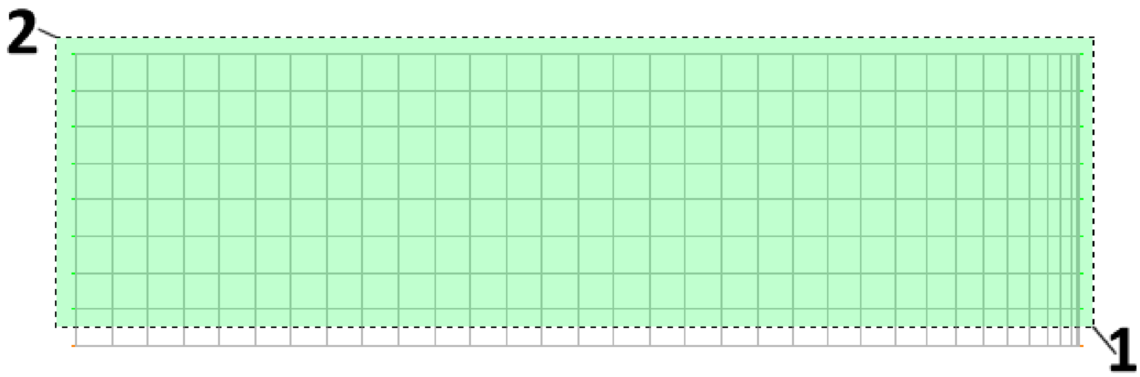
All the shell elements located in the bottom of the tank are now selected and added to group "Bottom".

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

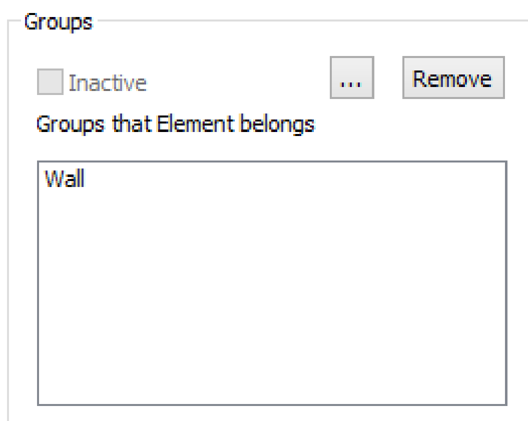


Click on the icon +Shells and make sure that the Group Wall is selected as the active group.

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



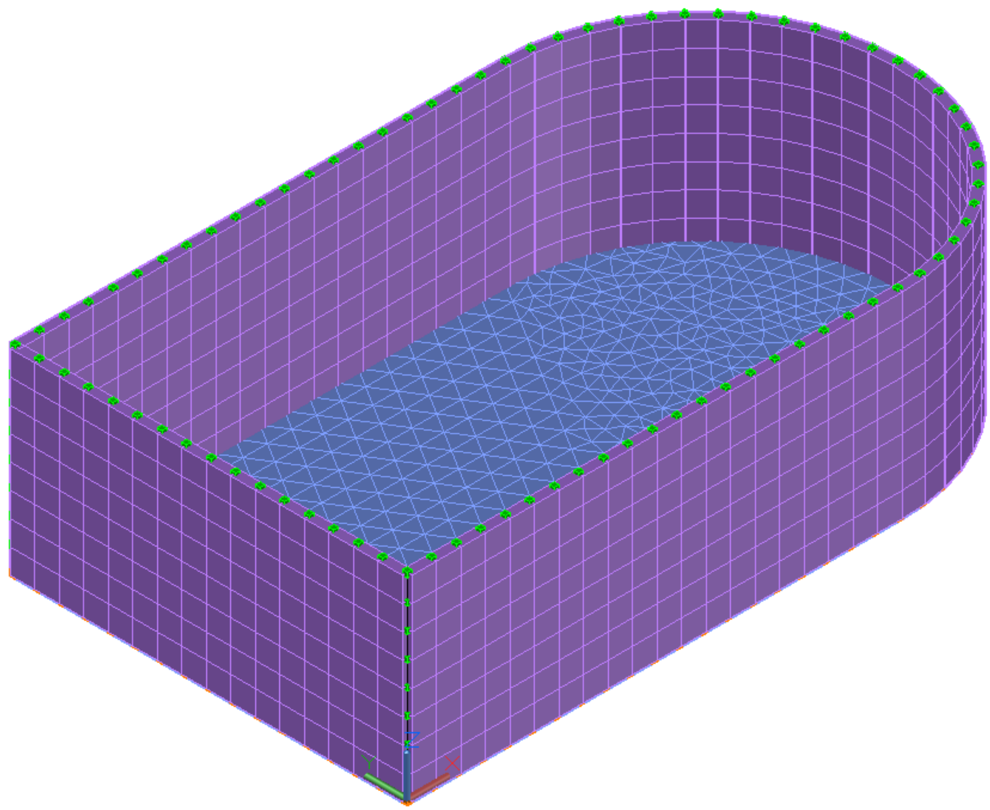
All shell elements located in the wall of the tank are now selected and added to group “Wall”.



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.

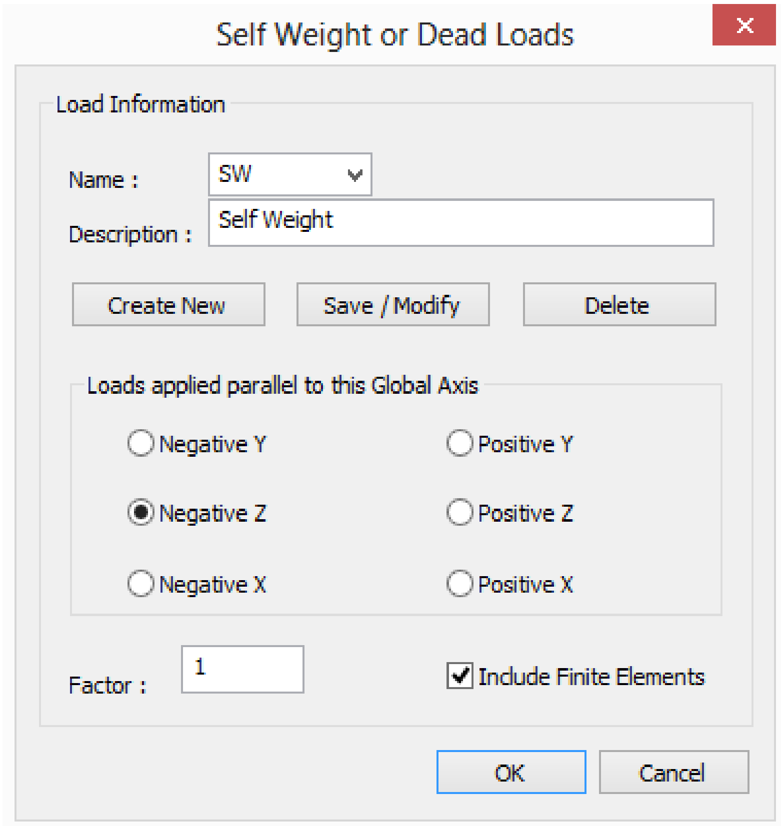


4.9. Define Loads

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.

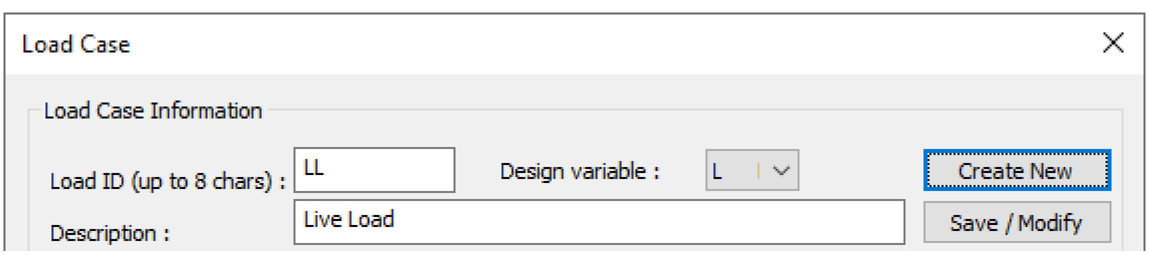


Step #19. Define Load Cases: Click on the icon Load Cases and the Load dialog appears.

Enter:

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

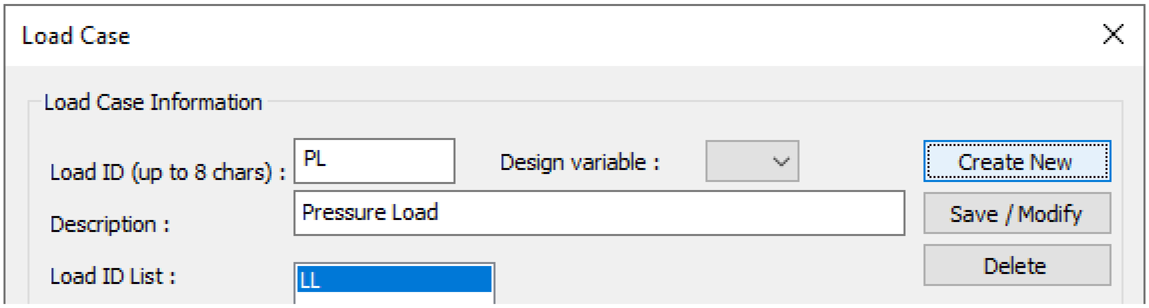
and press Create New.



Enter:

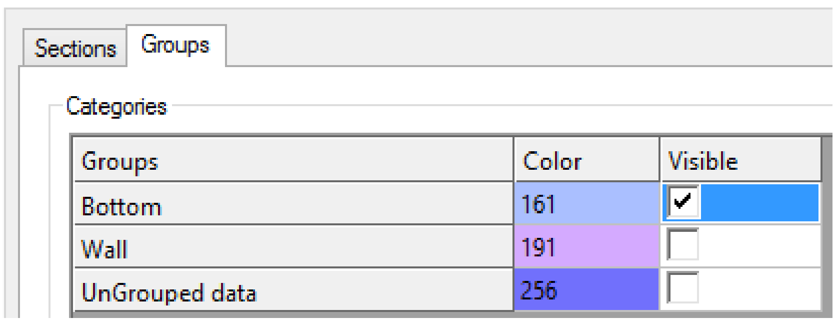
- *PL* as Name
- *Pressure Load* as the Load Description

and press Create New.



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.



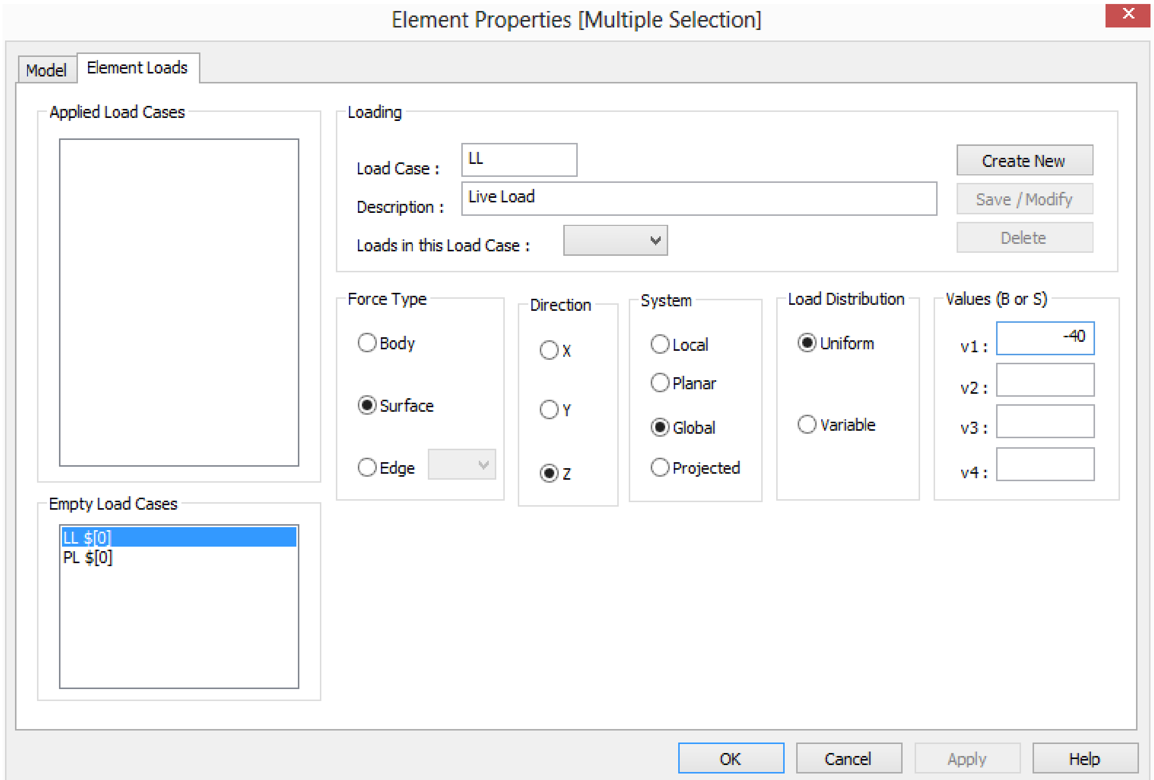
Click on the icon Shell in the Loads Panel. Using a full window, select all entities that appear on screen and press <ENTER> to finish with the selection.

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.

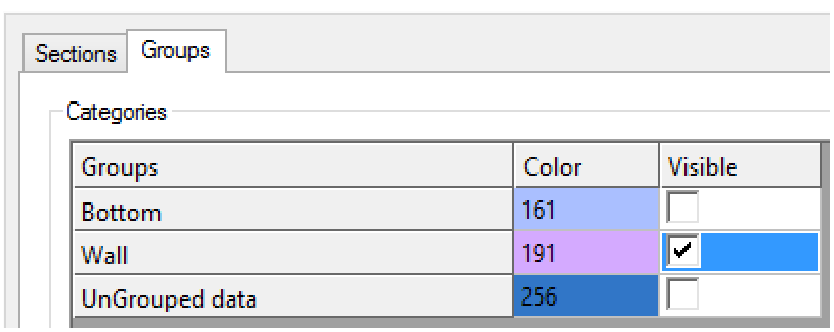


Press OK to close the dialog.

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.



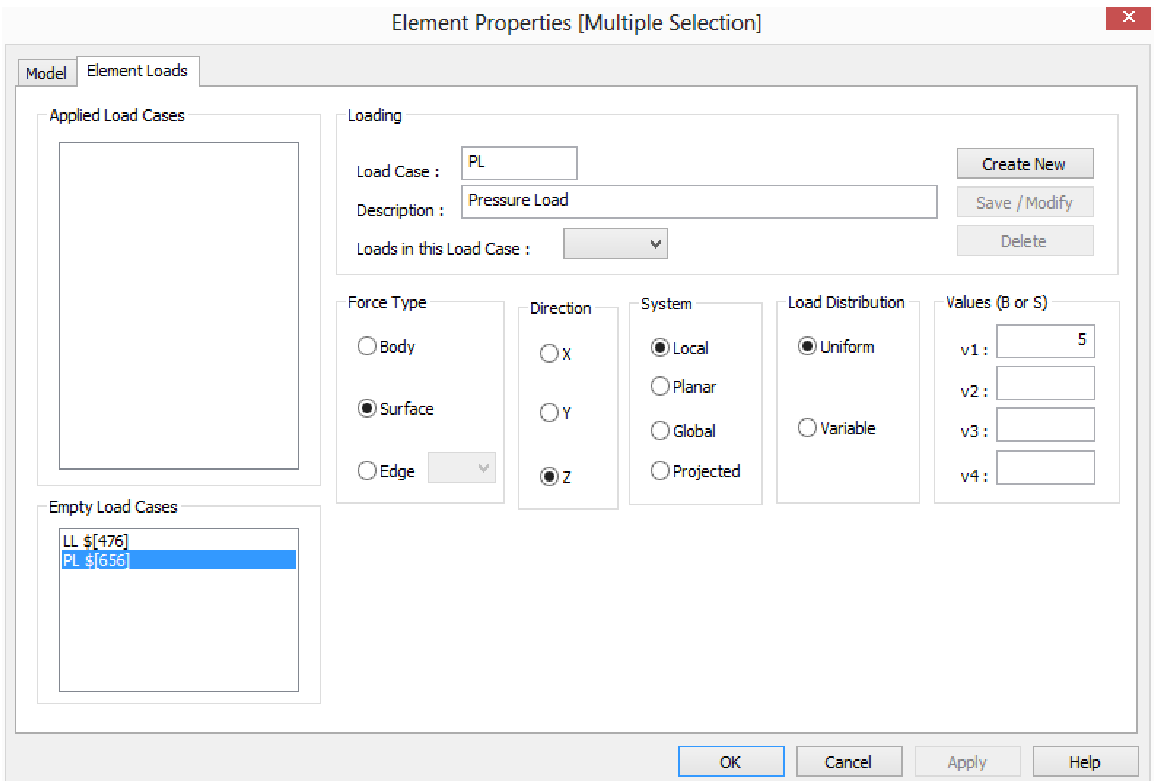
Click on the icon Shell in the Loads Panel. Using a full window, select all entities that appear on screen and press <ENTER> to finish with the selection.

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.



Press OK to close the dialog.

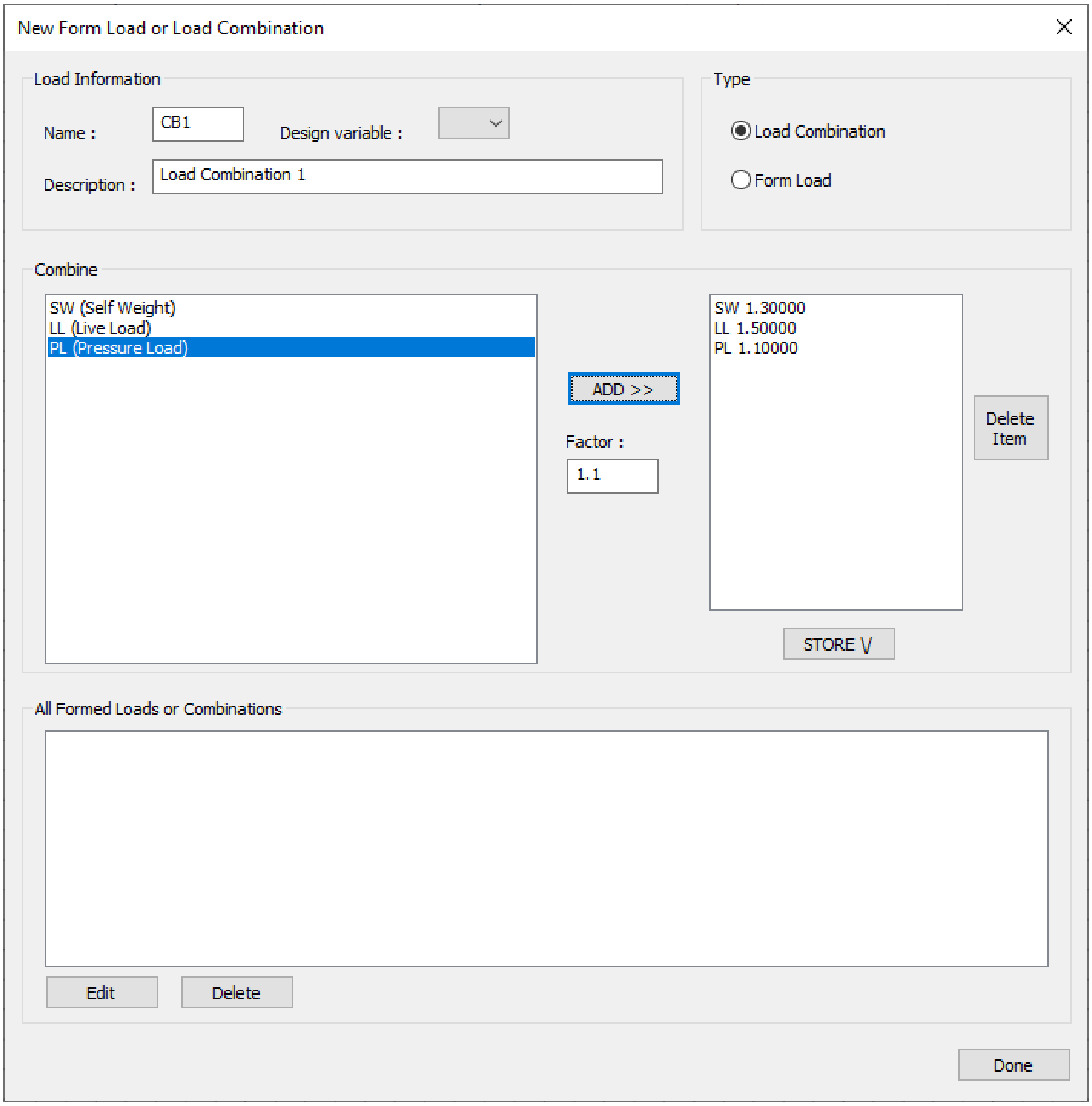
To clear the arrows select Clear from the GTS Display Ribbon Tab.

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:

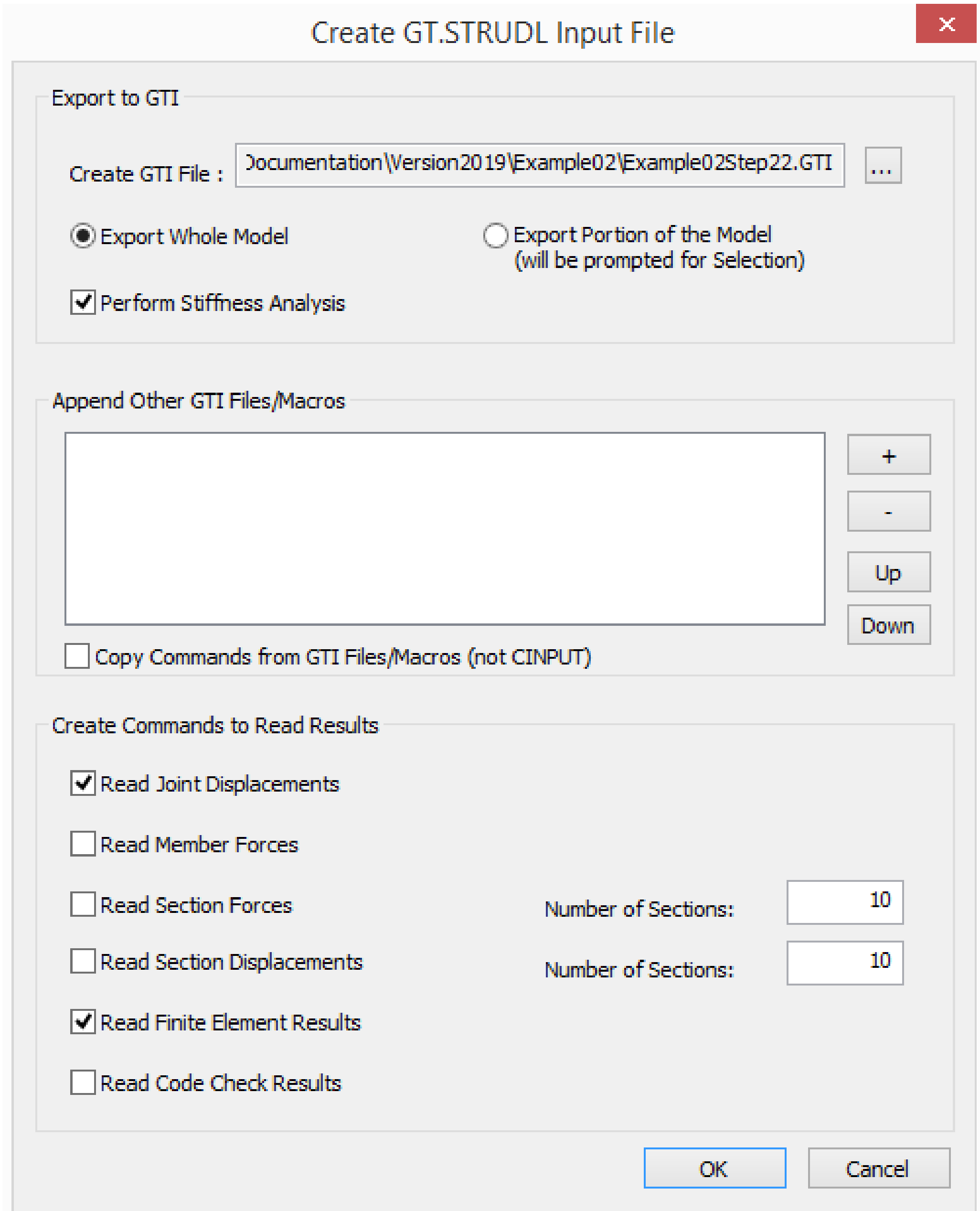
- *CB1* as Name
- *Load Combination 1* for the Description of the Load Combination
- Click on SW, Enter 1.3 as the factor and press ADD>>
- Click on LL, Enter 1.5 as the factor and press ADD>>

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



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.

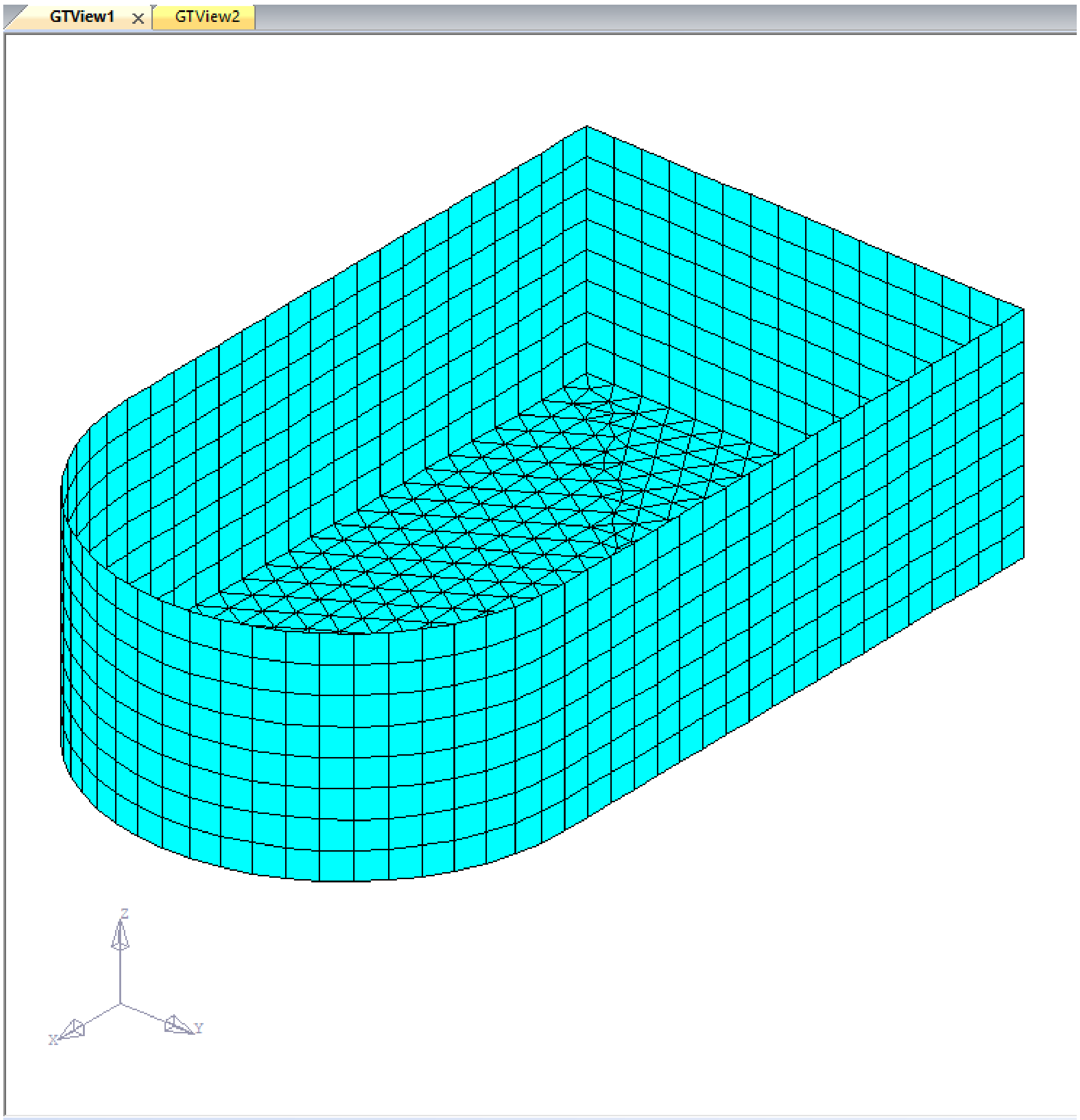


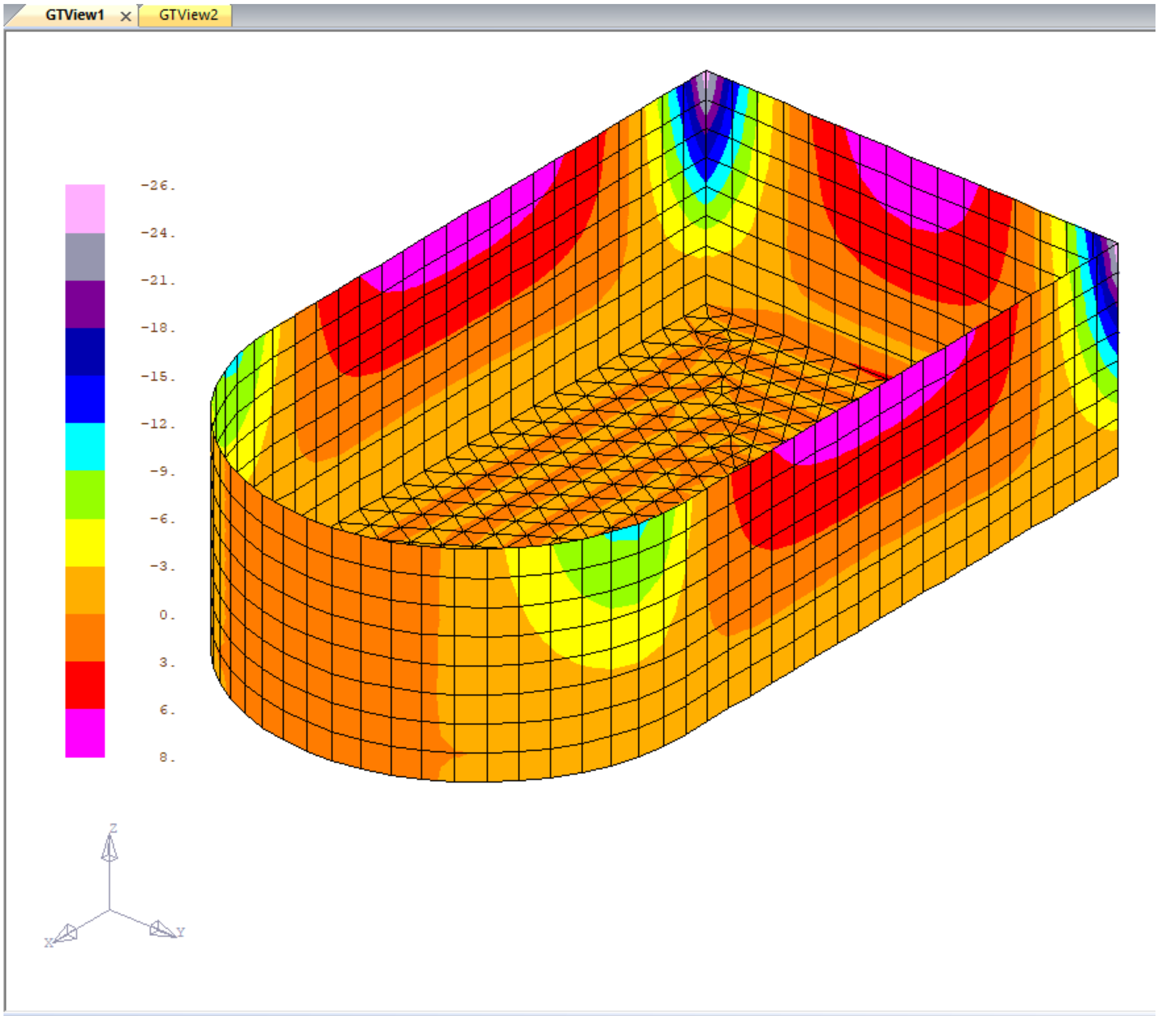
Step #24. View/Edit GTI: Click on the icon Edit GTI and the GTI file created in the previous step will be opened by the system's default text editor.

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

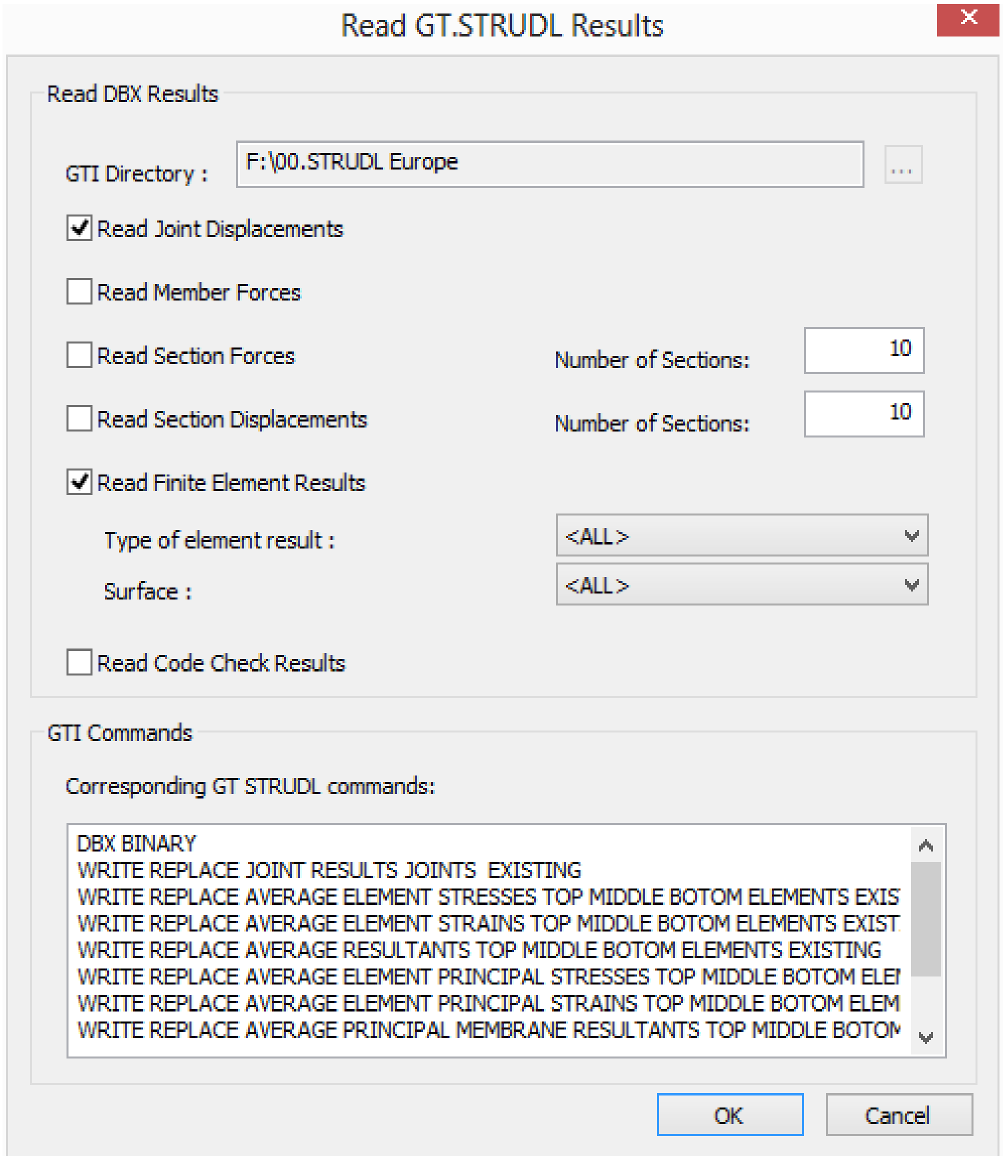
Stiffness analysis is automaticaly performed and DBX result files are automaticaly created.

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.

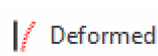


If you get the following error message at the command prompt:

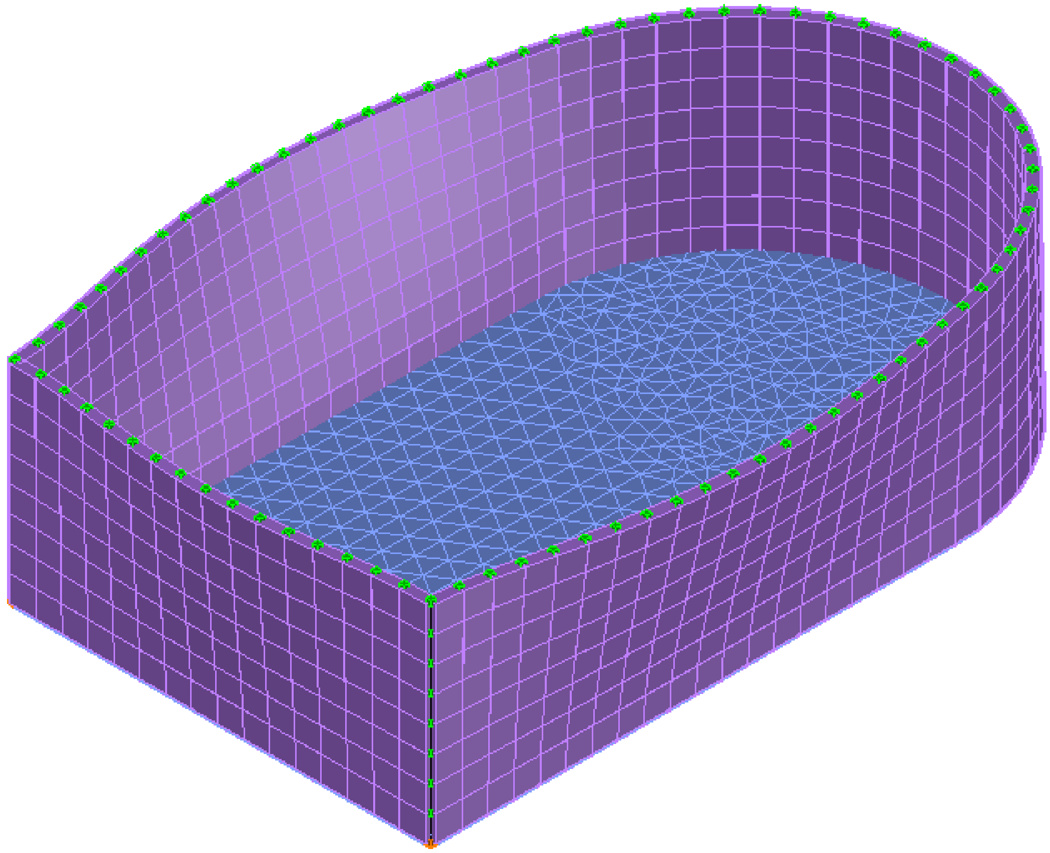
```
ERROR Loading Results: The following DBX files cannot be found:

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 

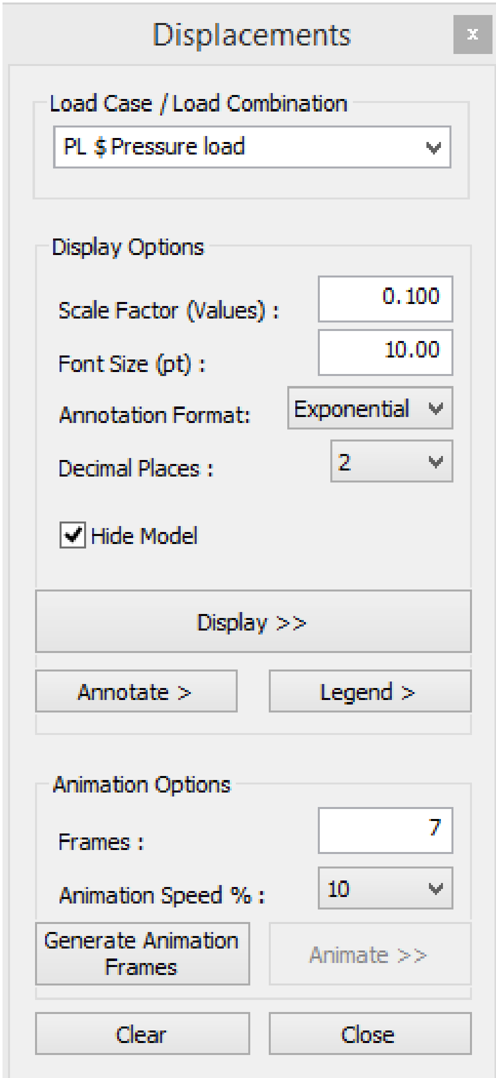
(ribbon tab "GTS Display") and then select PL as the load Case and press ENTER twice. The deformed structure will be drawn as shown below.



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 possition.

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”).



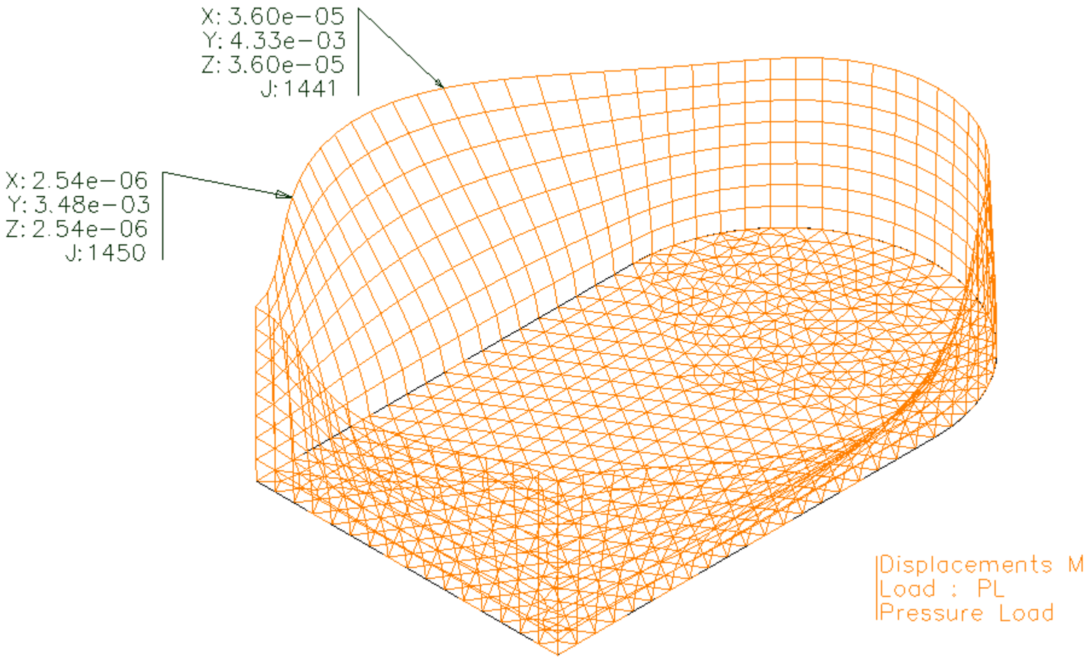
Select:

- *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.

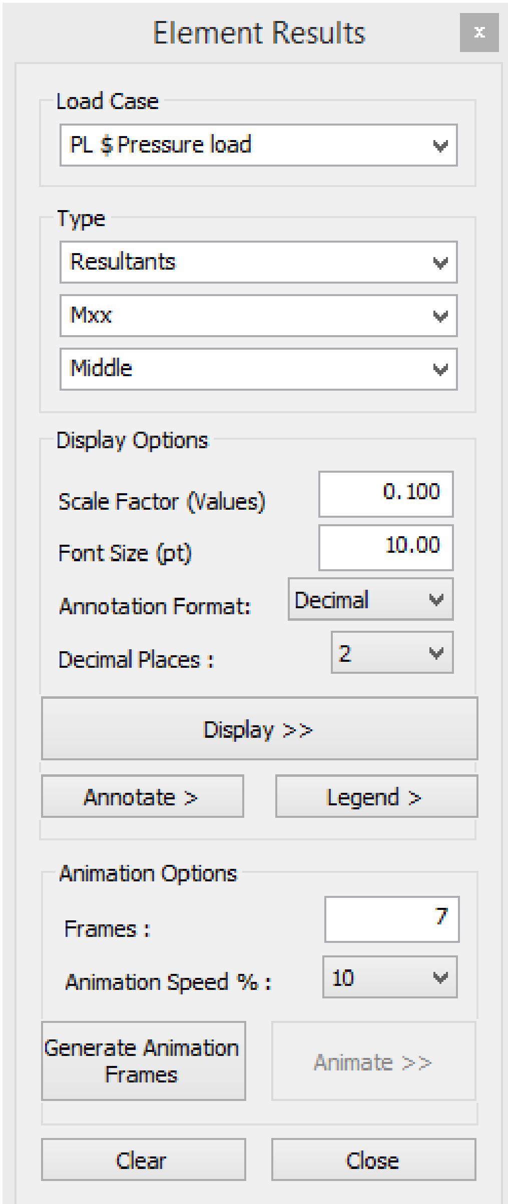
Press on "Legend >" and click at any part of the screen to place the legend of the diagram.



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:

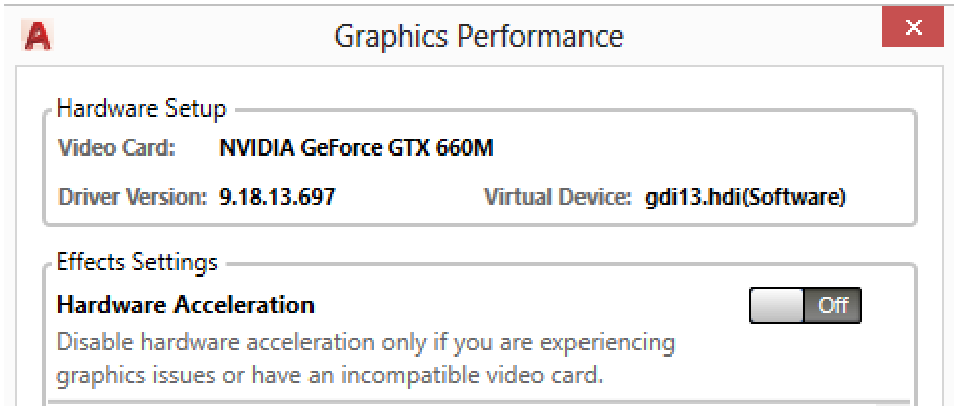
- *PL* as Load Case
- *Resultants* as the Type of element result
- *Mxx* as the Moment Resultant to display
- *Middle* as position (Resultants are only available for the middle surface of a 2D finite element)

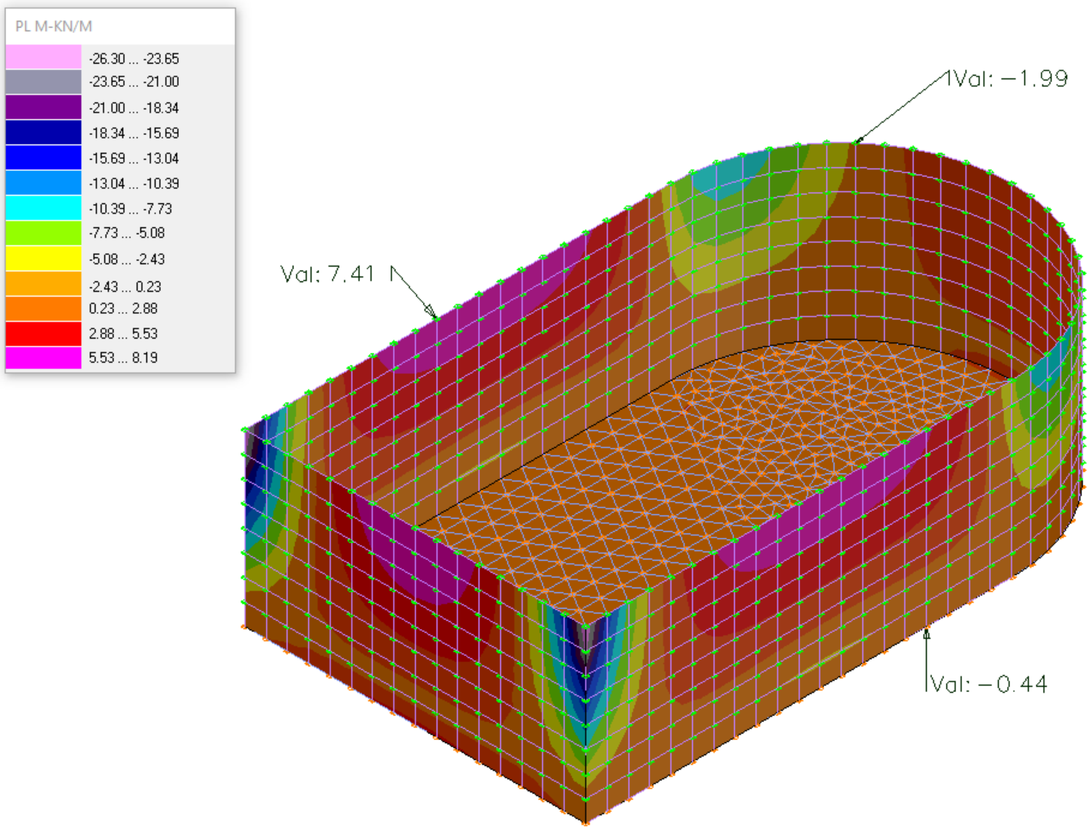
and press "*Display >>*"

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





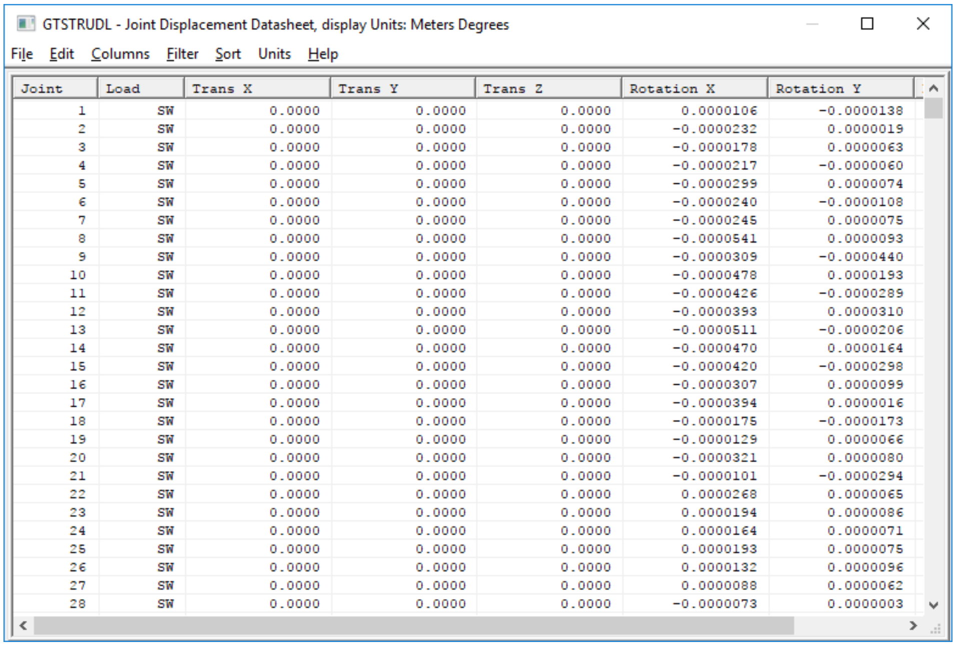
In order to display the animation press “Generate Animation Frames” and then “Animate >>”. To terminate the animation press “Stop” button.

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".

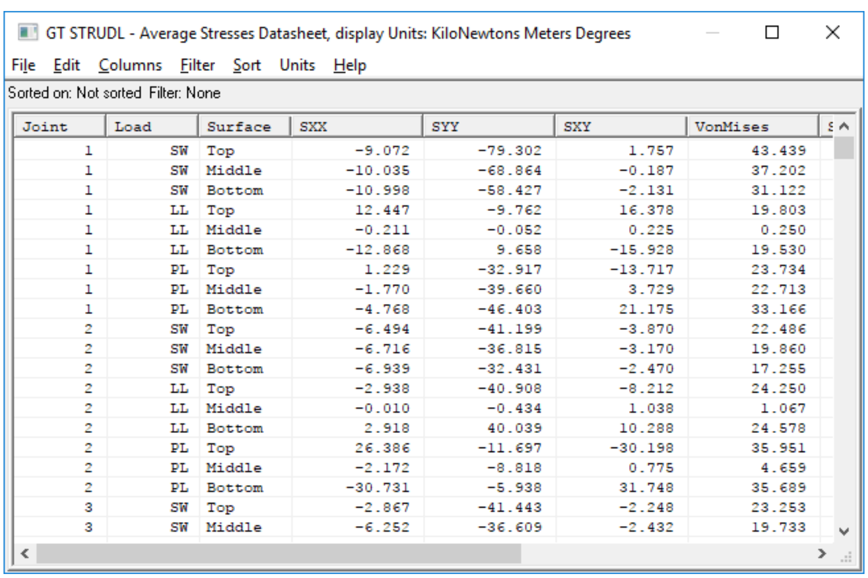
Click on the icon Displacements and the "GTSTRUDL – Joint Displacement Datasheet" dialog appears where you can filter, sort, write results to text file or change results units as shown in figure below;


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

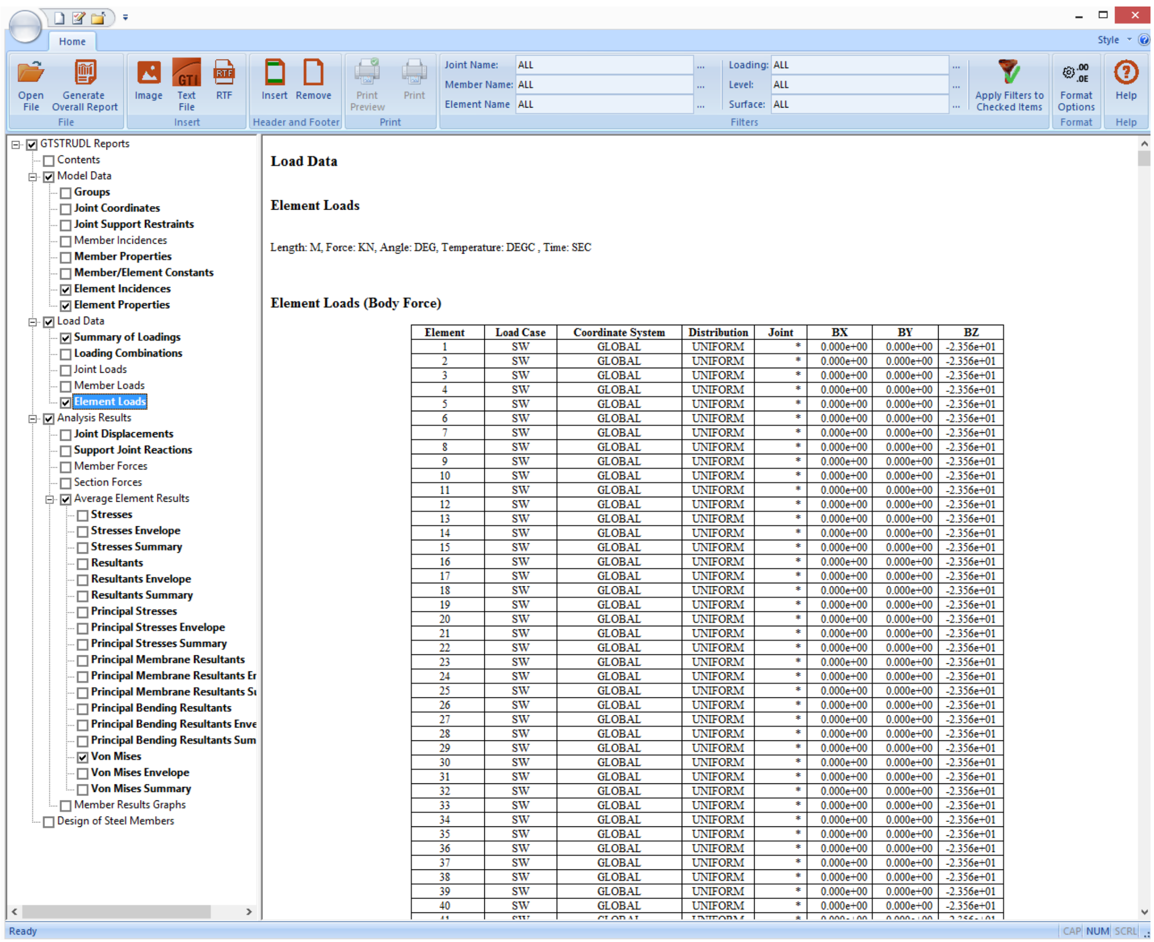
Sorted on: Not sorted Filter: None

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 

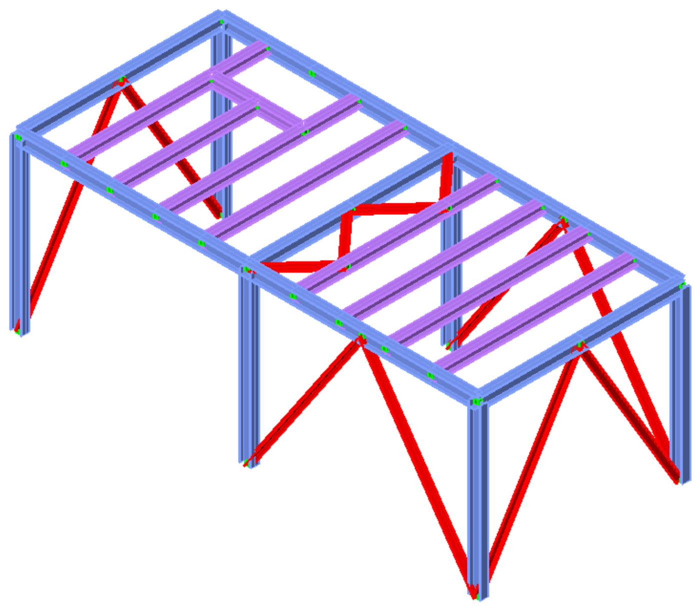
to generate the report of this structure by calling Report Builder. Report Builder is launched in a separate window. You can find more information on how creating your report in the GT STRUDL® Report Builder Getting Started Guide (Help Icon on the right top corner on Report Builders' window).



5. Tutorial Example #3

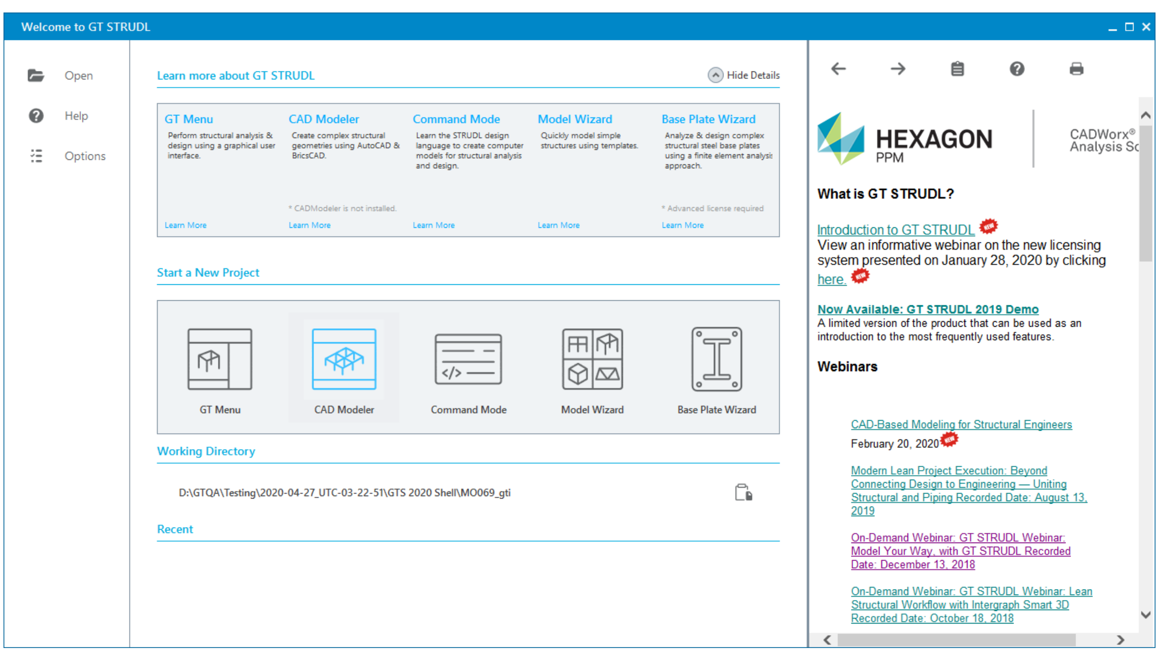
5.1. Introduction

The modeling of a frame structure using CAD Modeler shown below is demonstrated in a step-by-step process.

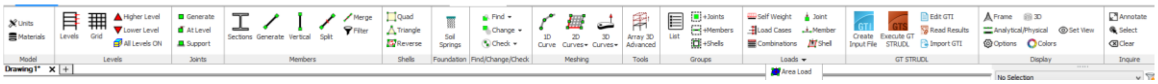


5.2. Open CAD Modeler and start working

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>`.

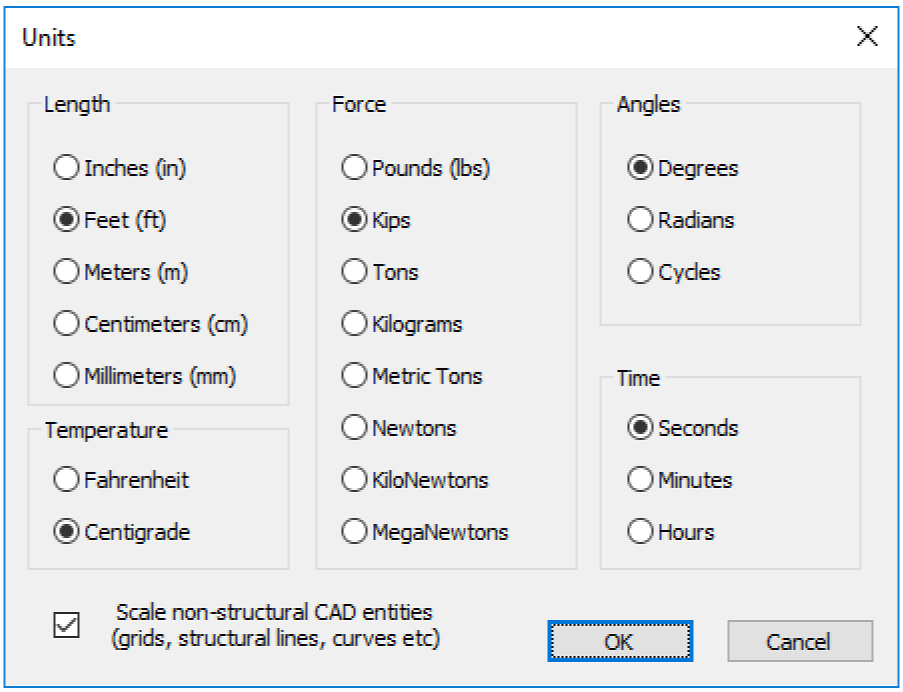
If AutoCAD's/BricsCAD's ribbon area is not visible, type `RIBBON` and press `<ENTER>`.

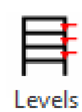
Ribbon commands will be used in this tutorial example. However, since all ribbon commands appear in the menu area, you can use the menu bar as well.

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*.

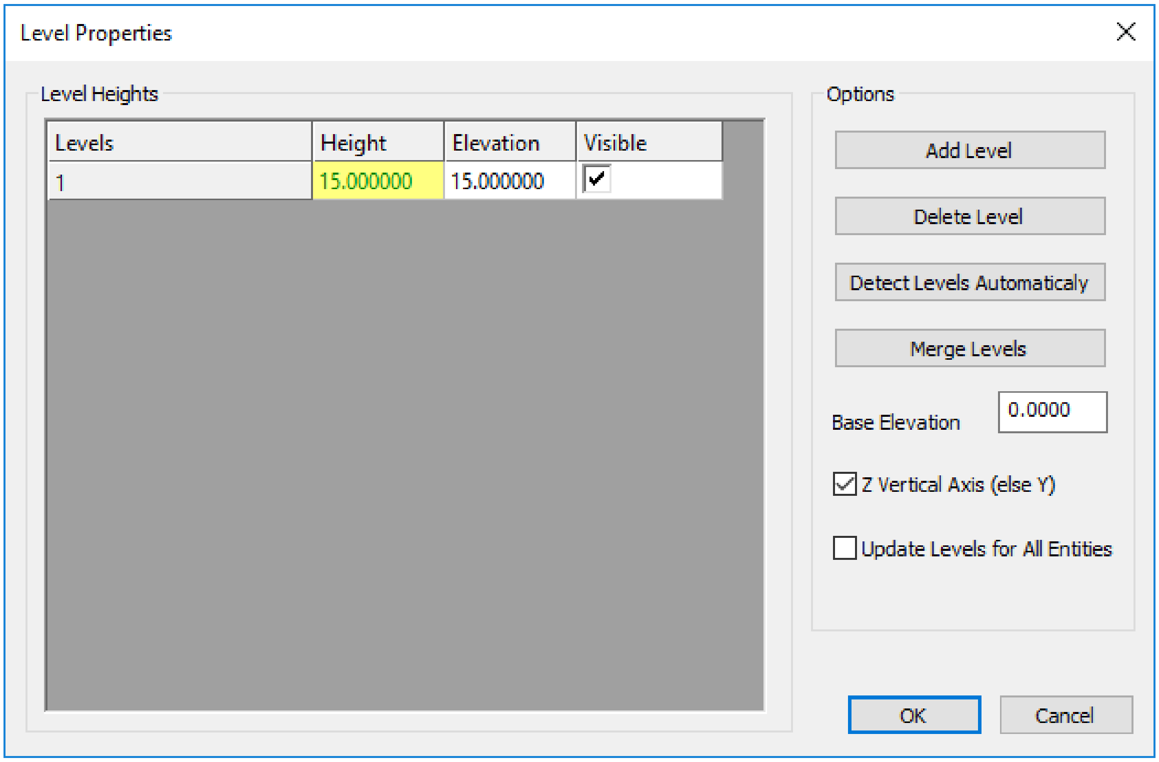


Step #4. Define the one level of the model by pressing the icon 

. 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`.

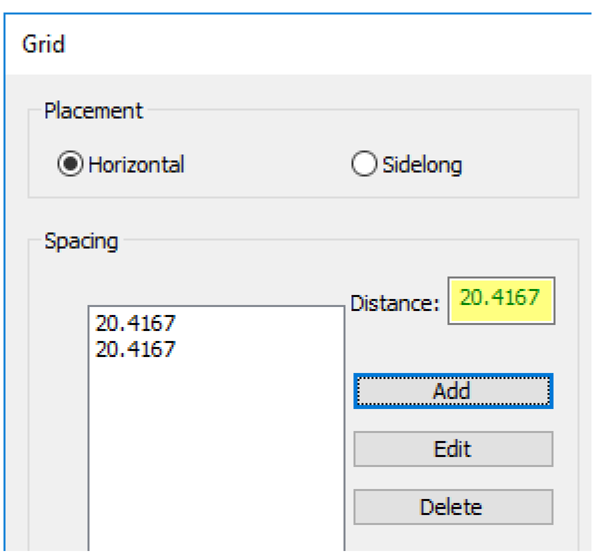
Note: Some Edit Boxes appear in yellow background and green fonts, like the one at the Level Heights. You can use mixed units in the yellow edit boxes. For more information about Mixed Units and the valid syntax please read GT STRUDL GT Menu Guide

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



Step #5. Enter a Grid that will help you enter the columns quickly by clicking on the icon 

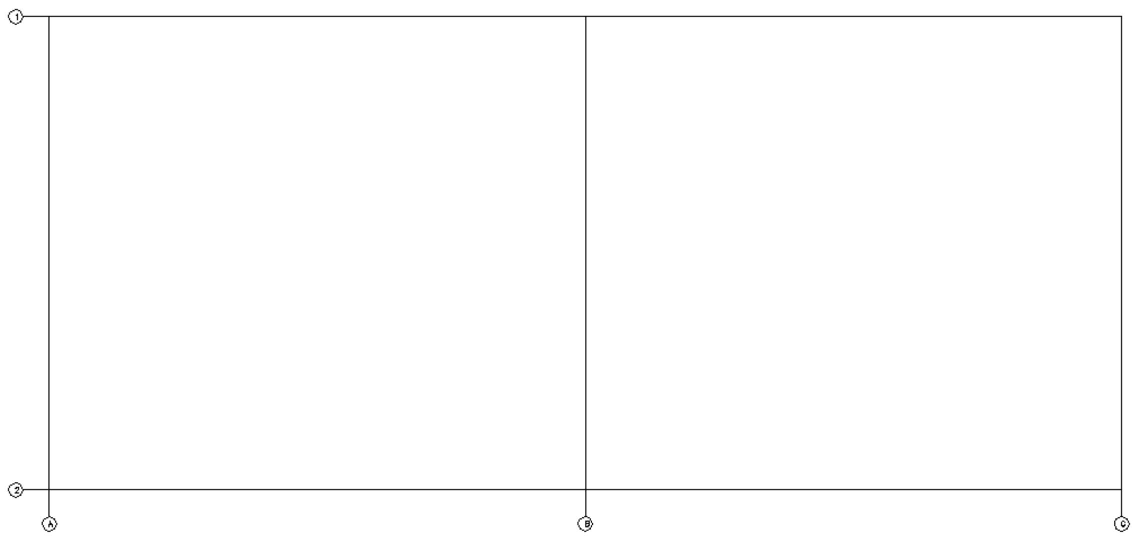
. The grid is going to have 2 (20.4167 ft) spaces in the horizontal direction (X) and 1 space (18 ft) in the sidelong direction (Y). Enter `20ft-5in` in the *Distance* text box and press the *Add* button 2 times. The program will automatically convert the distance of 20ft -5in to 20.4167 ft.



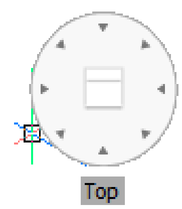
Then, click on *Sidelong* in the *Placement* area to enter the spacing in the Y direction, and enter `18` and *Add*.

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

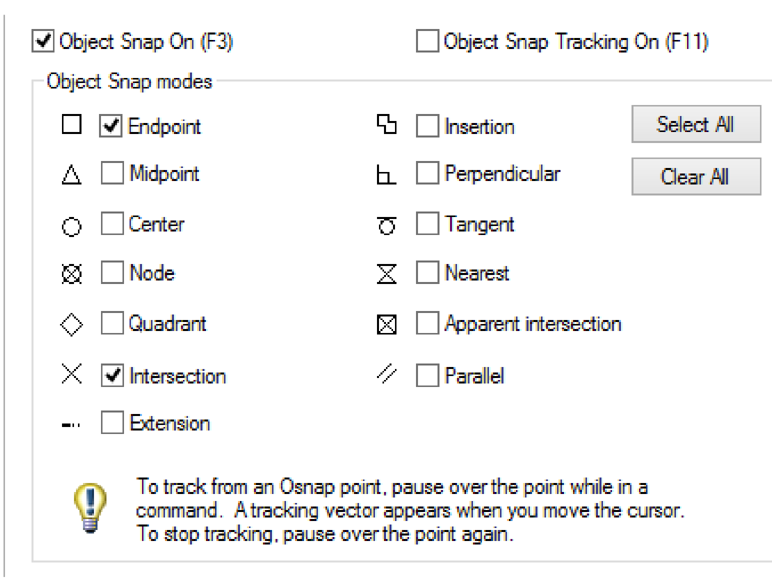
The grid is created, having its upper left corner A-1 at the point 0,0,0.



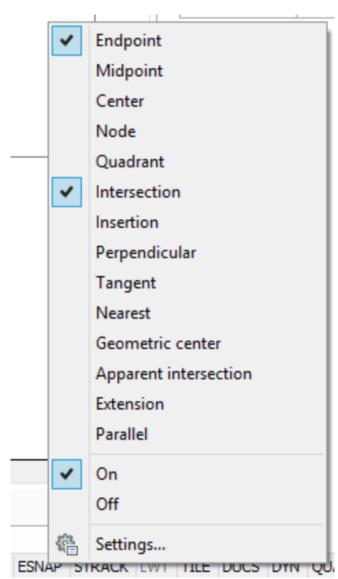
You will be able to view the grid by pressing the Top Icon in AutoCAD's Cube or BricsCAD's Chair, or preferably by typing `Z` (for Zoom), `E` (for Extents) and press `<ENTER>`.



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.

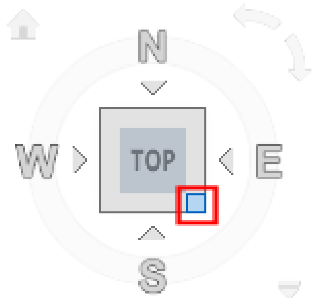


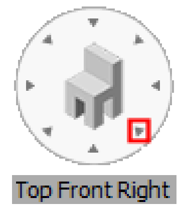
AutoCAD's Snap Setting



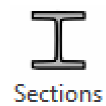
BricsCAD's Snap Settings

Step #6. Click at *Top Front Right* of the View Cube (AutoCAD) or Chair (BricsCAD) to change the view of the model.

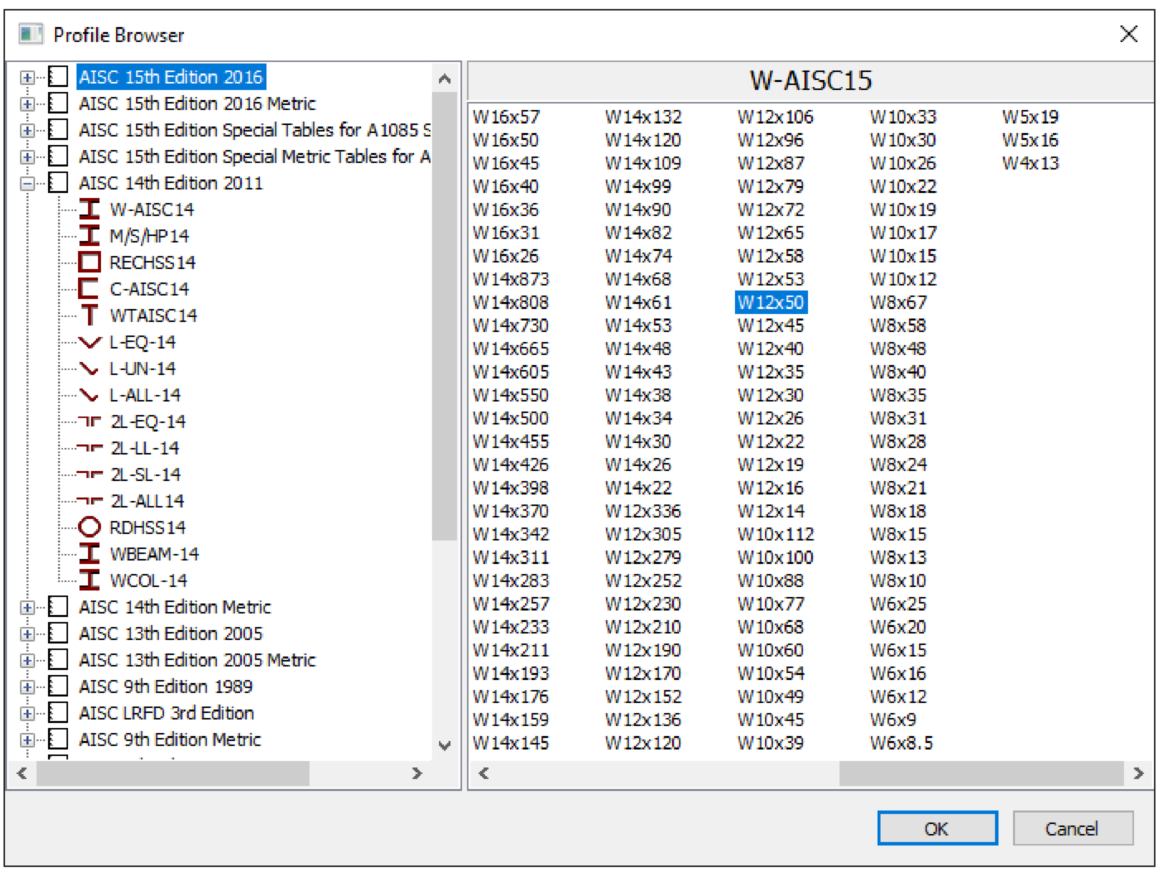


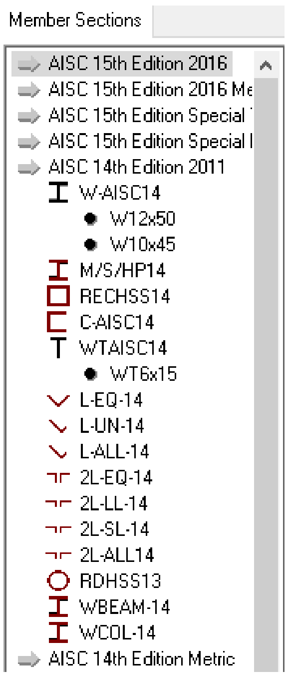


AutoCAD's View Cube for Isometric View BricsCAD's View Cube for Isometric View

Enter the cross-section profiles that will be used at the model by pressing the icon 

. 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.



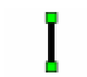


The profile is added to the project and it appears in the left list-box having a black dot in front of it.

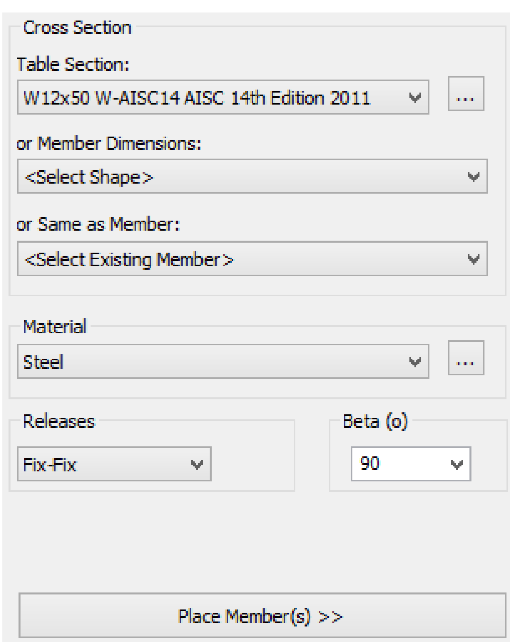
Using the same procedure, add 2 additional profiles: *W10x45*, for beams and *WT6x15* for the bracing from the table *WTAISC14*. Press OK to close the form.

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

Step #7. Start entering the columns by clicking on the icon 

. The dialog *Place Member* appears that helps you to quickly select properties for the members that are going to be entered.



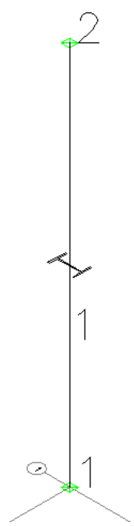
Select *W12x50* as the cross section for the columns. Make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *90*. There is no need to close this dialog manually.

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.

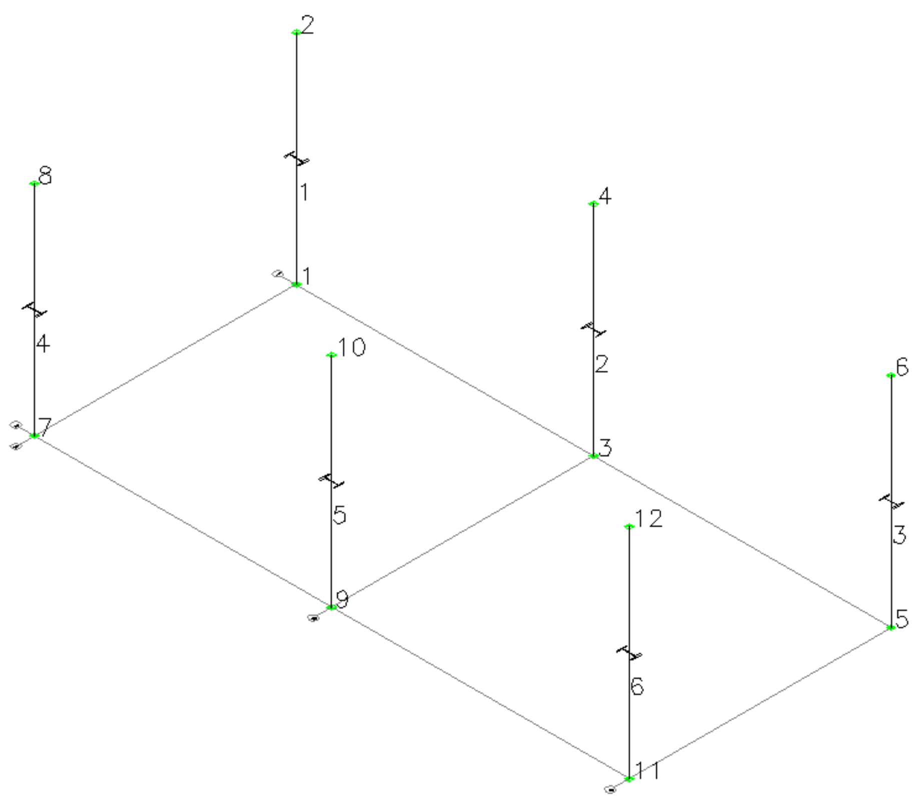
Repeat the same procedure by clicking at the points B-1, C-1, A-2, B-2 and C-2.

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).

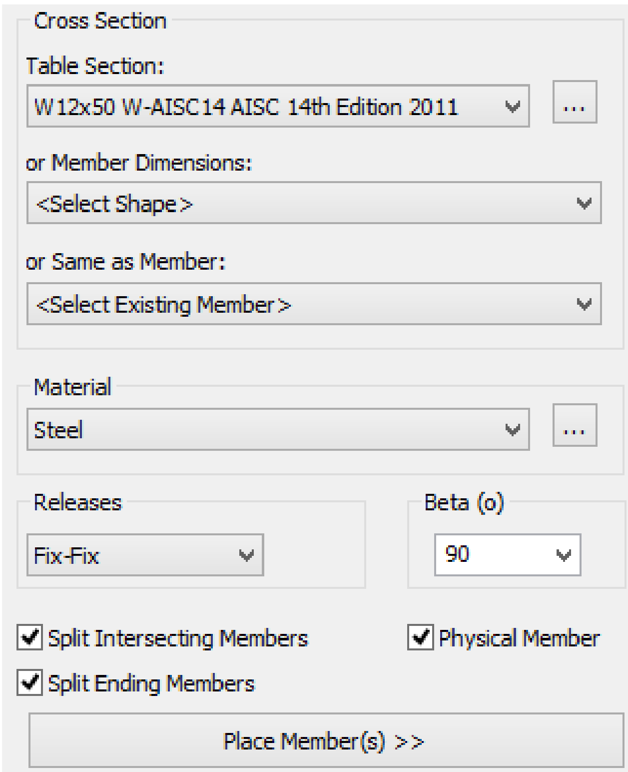
Step #8. As you can see in the isometric view below, column members 1 to 6 were created together with joints 1 to 12 at their ends. Each column is 15 ft long, as defined in Level Properties (height of the first floor).



5.5. Create beams and girders

Step #9. Start entering the beams, along X axis, by clicking on the icon 

. The *Place Member* form appears.

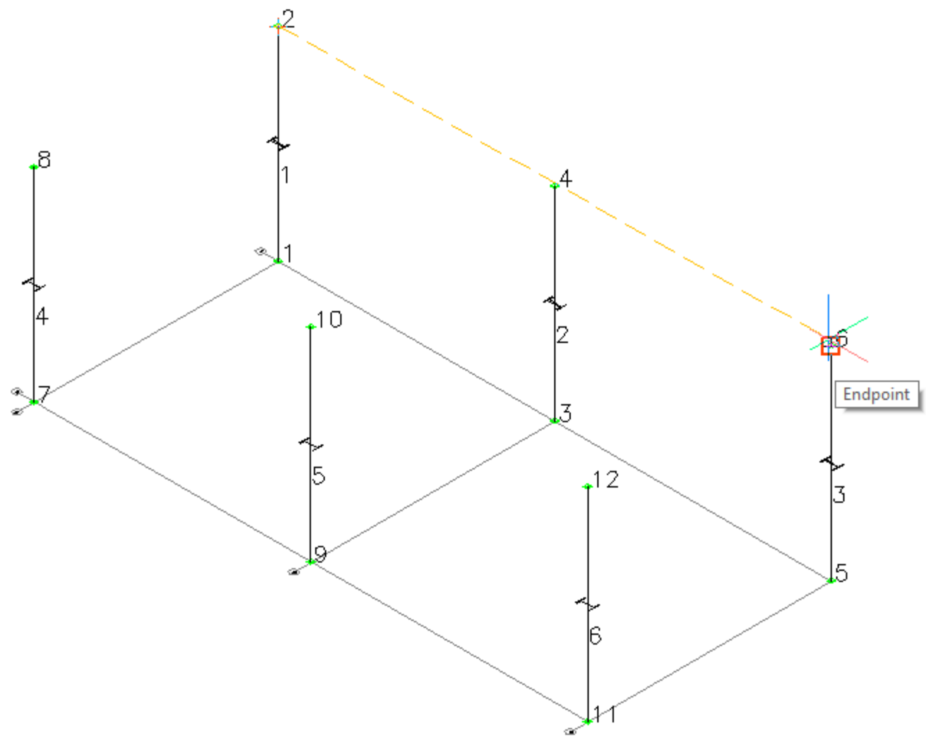


Select *W12x50* as the cross section and make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *90*. You have to set the Beta angle equal to 90 degrees in order to orientate the local Y axis of the W12x50 cross section along the Z global axis. Make sure that the option *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.

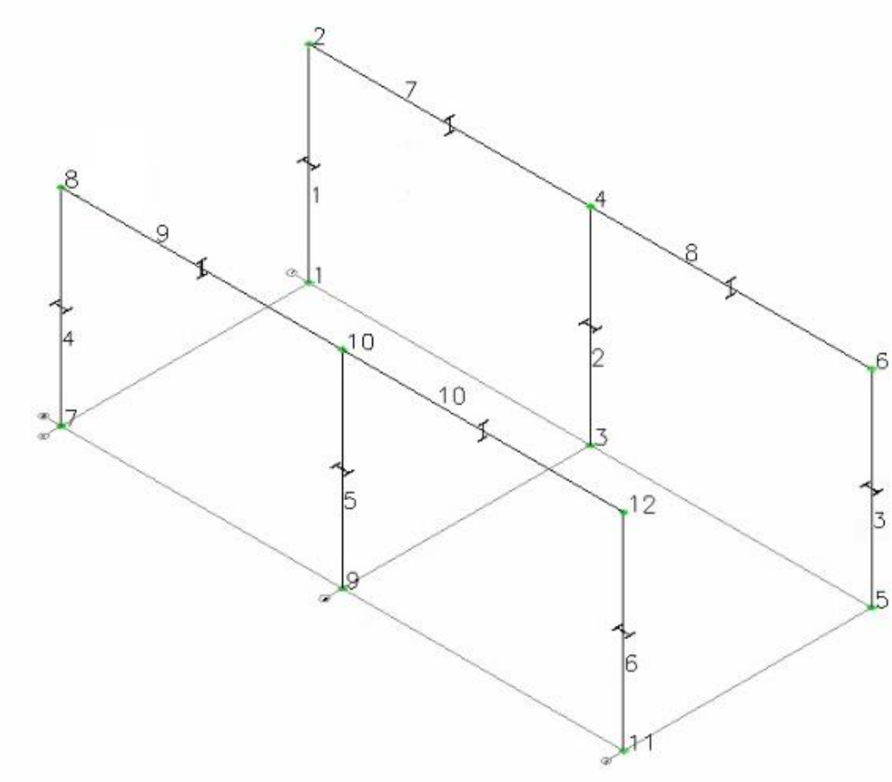
Then you have to define the *Ending Point (x,y,z)* so click on Joint 6, as shown in the picture below.



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.

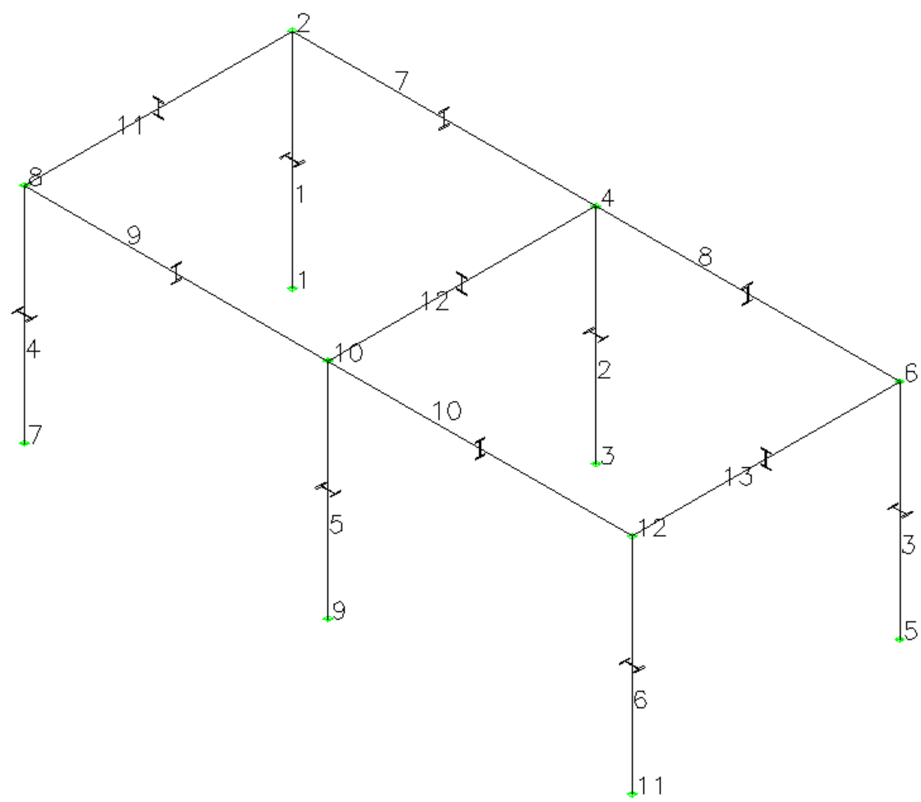
Since the command is still active, you are prompted to enter the Starting Point (x,y,z), repeat the same procedure by clicking on Joint 8 (top of column at position A-2), and then click at Joint 12 (top of column at position C-2). Members 9 and 10 will be created.



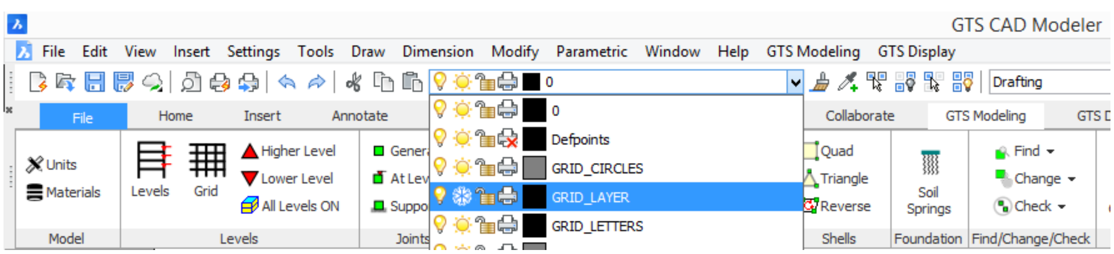
Step #10. Enter the beams, along Y axis. The command Generate Beams should be still active, else you can call it again by clicking again on the icon . Keep the same settings at the Place Member Form, as in the previous step, regarding the cross section and Beta angle, but do NOT click on *Split Intersecting 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.

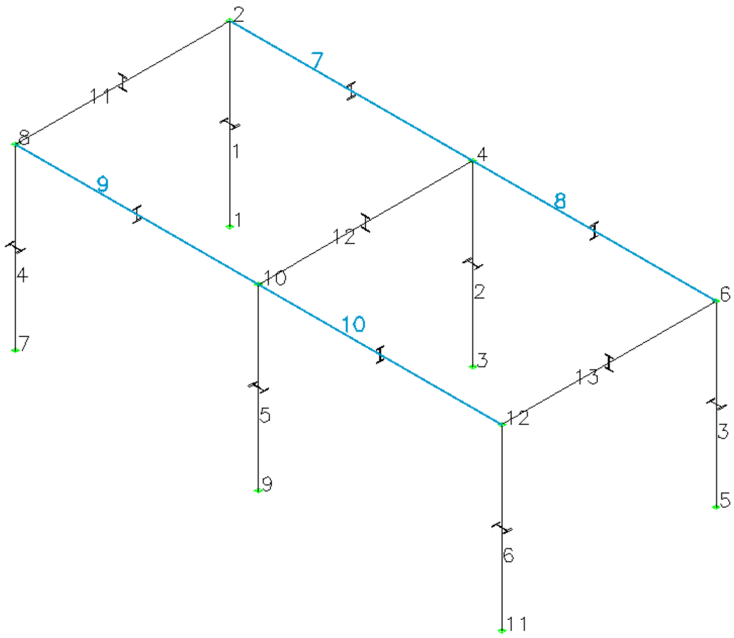
Repeat the same procedure by clicking on the Joints 10 and 4 to generate Member 12, Joints 12 and 6 to generate Member 13. Then, press ESC to terminate the command.



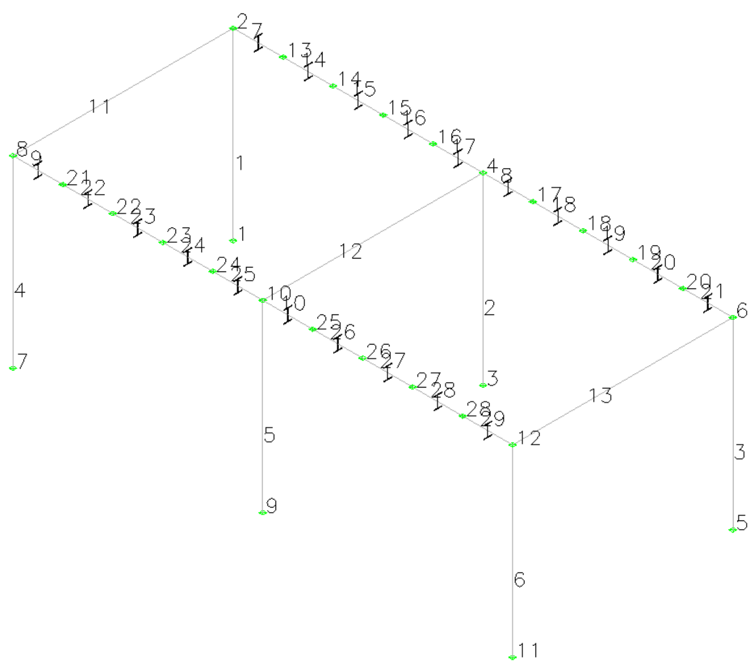
Step #11. Hide Grid: Since the Grid is no longer needed it can be hidden using AutoCAD/BricsCAD commands to freeze the layer GRID_LAYER. This can be done by clicking on the Home Tab and then selecting the GRID_LAYER from the drop down list of layers and clicking on its freeze icon.



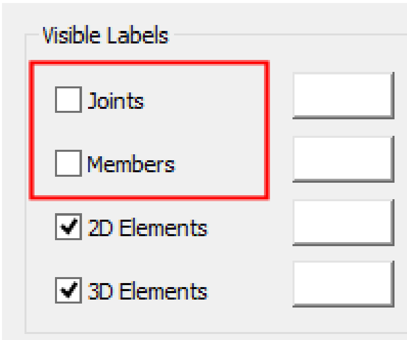
Step #12. Split horizontal members: Click on the icon  and click on the members 7, 8, 9 and 10, and then press <ENTER>.



In order to define the Distance for spliting the member or the number of parts (negative number), enter −5 , so that the beams will be split into 5 equal parts.

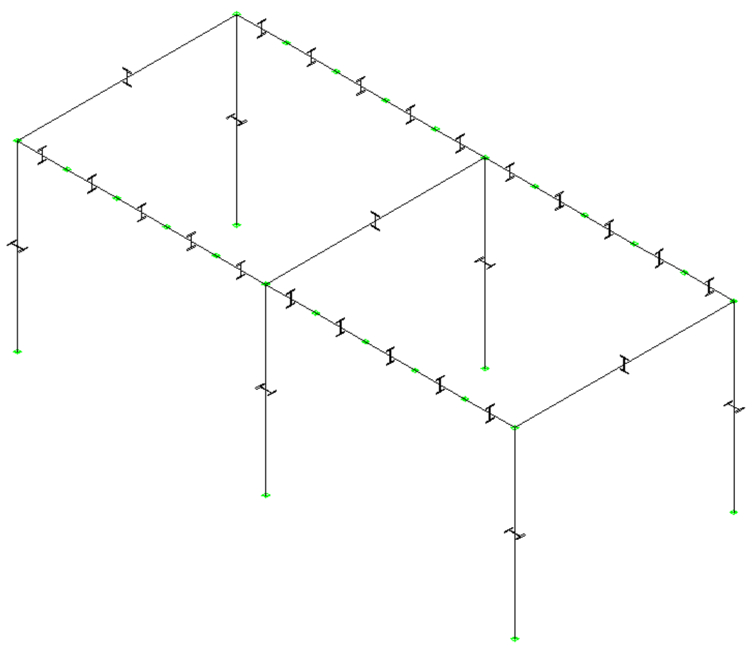


Step #13. Turn OFF labeling:

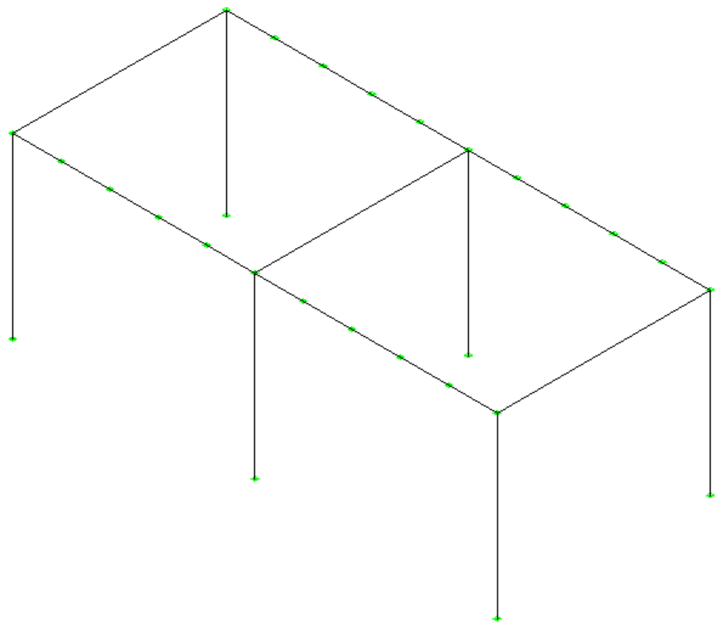


Click on the icon Options in the ribbon bar and then uncheck the Visible Labels option for Joints and Members.

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



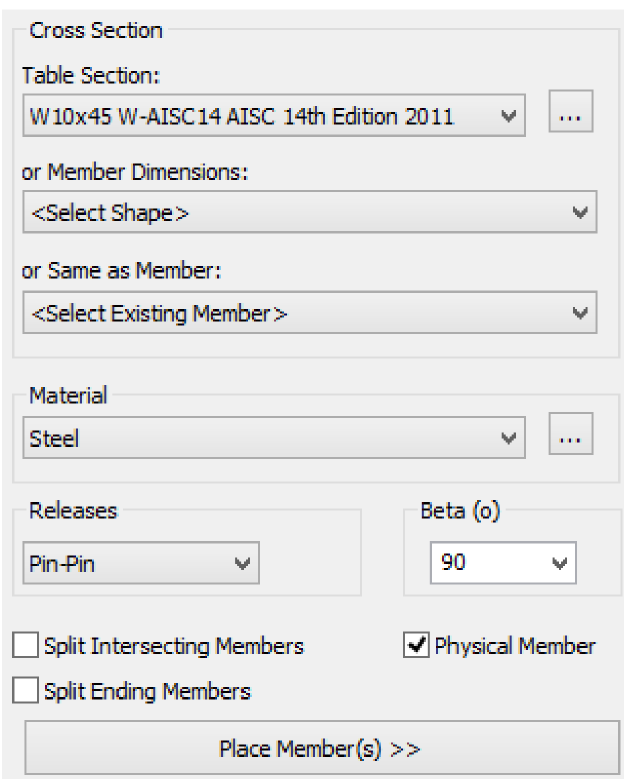
Press the icon Clear to remove the informative cross section shapes in the middle of each member:



5.6. Create girders

Step #14. Place girder members along Y axis: Click on the icon 

and Place Member form appears.



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

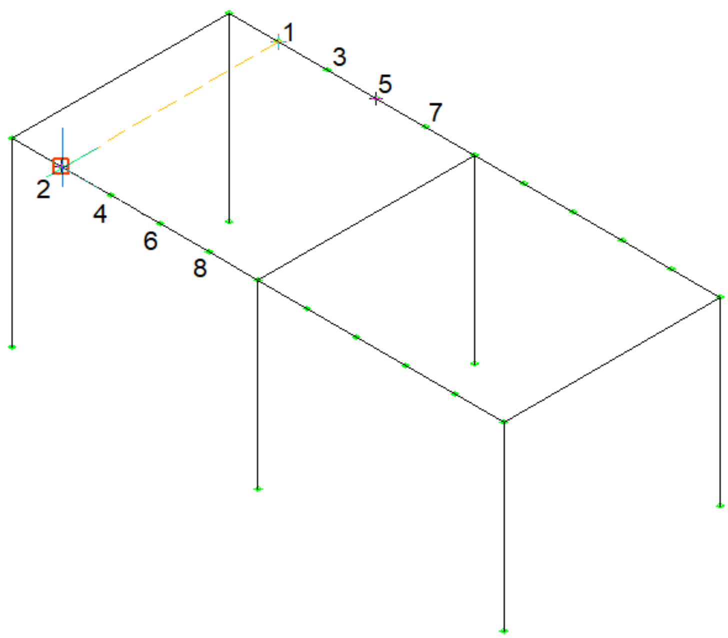
Press the "Place Member >>" button.

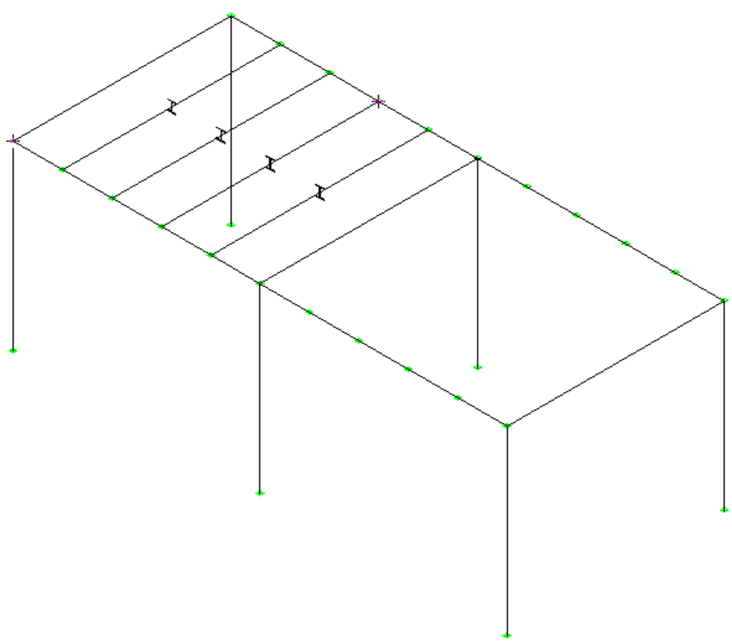
Click on the Joint located at the point 1 of the following image. Click on the joint at point 2 and the girder member is generated.

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.



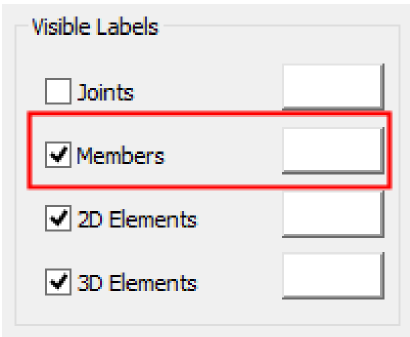


Press *ESC* to terminate the Generate Beam Command.

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.

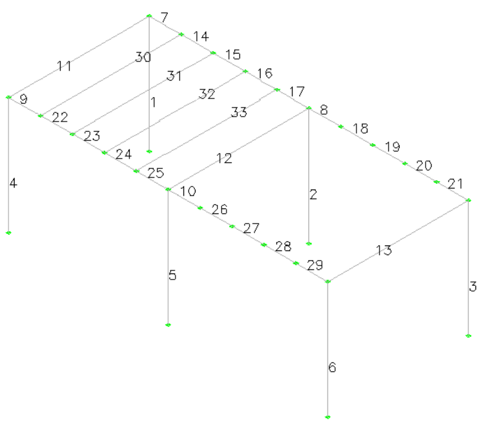
Turn ON labeling:



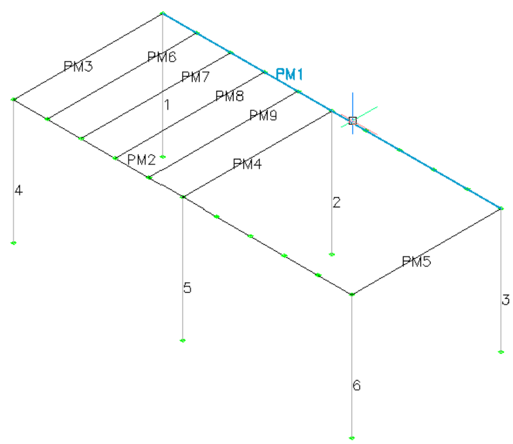
Click on the icon Options in the ribbon bar and then check the Visible Labels option for Members and press OK

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.

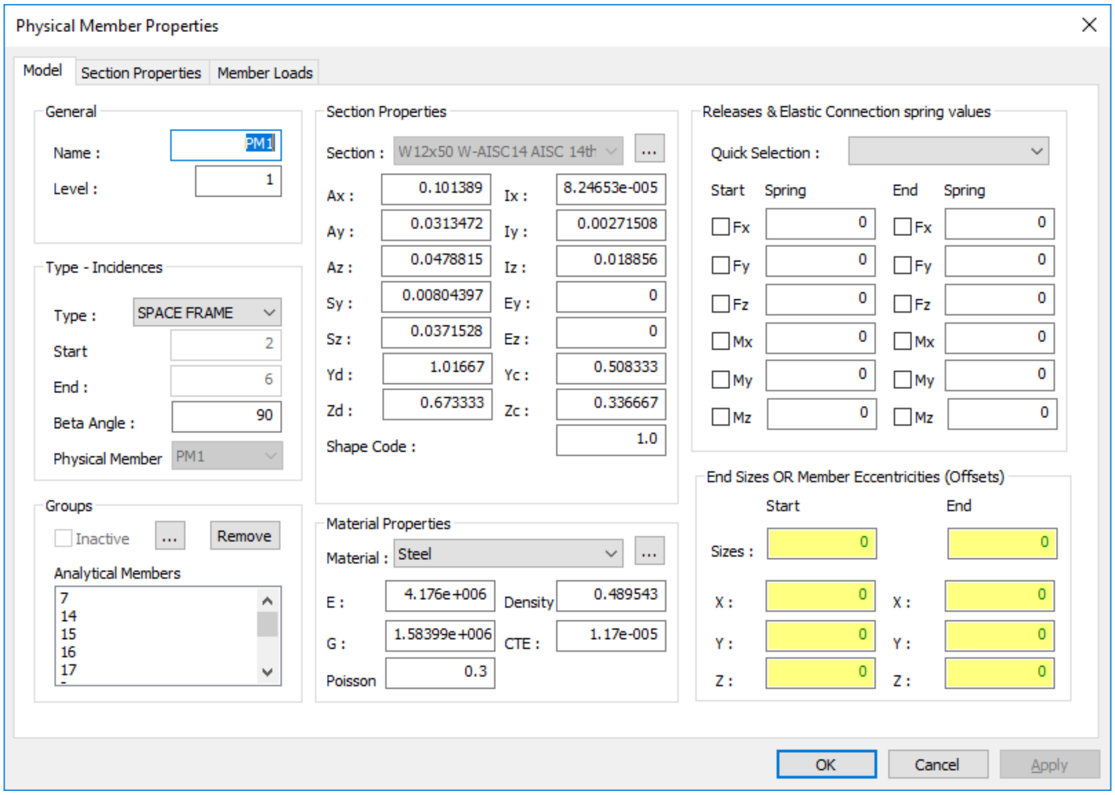


Analytical View



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.



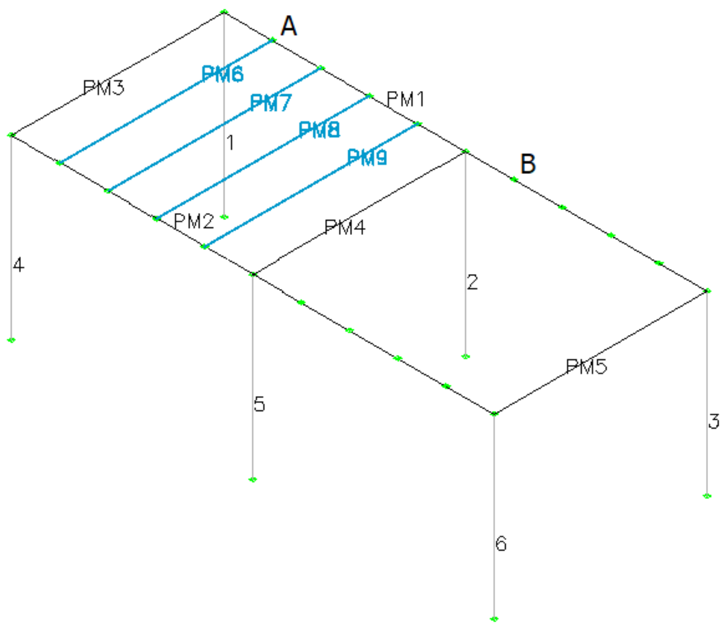
Step #16. Copy Physical Members

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.

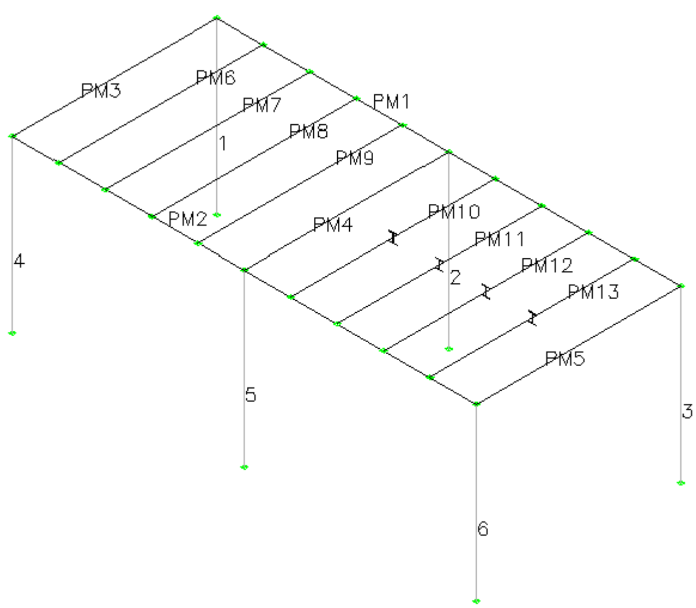
Type AutoCAD/BricsCAD `COPY` command and when asked to *Select objects/entites:* click on the members PM6, PM7, PM8 and PM9. You will get a notification about the selected entities and press `<ENTER>`.

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.

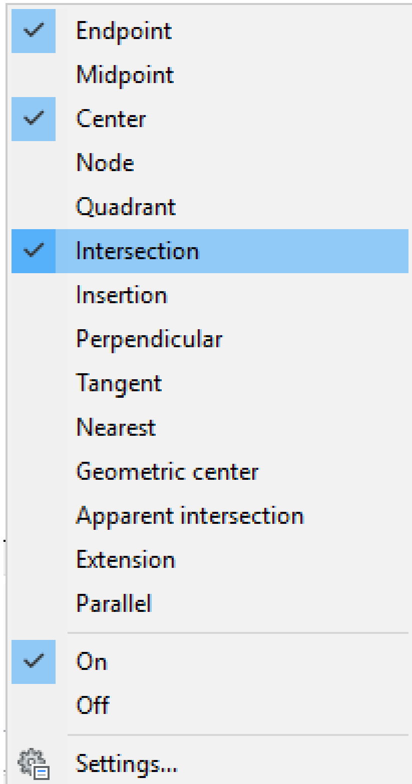


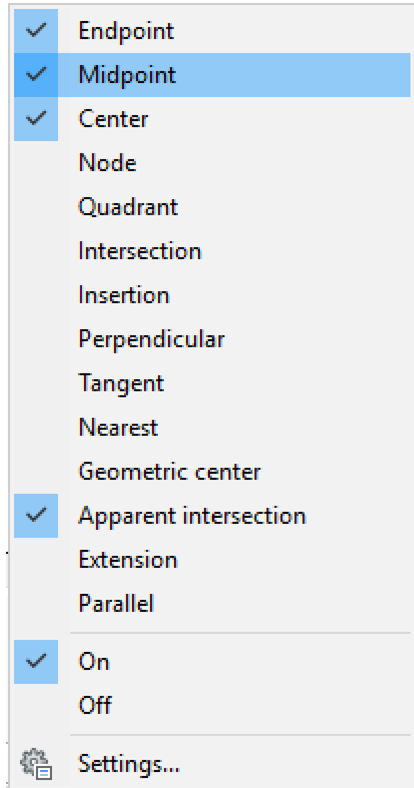
Switch back to analytical view by pressing the icon Analytical/Physical and turn OFF member labeling (as explained in Step #13).

Press the icon Clear to remove the informative cross section shapes in the middle of each member.

Step #17. Place bracing members.

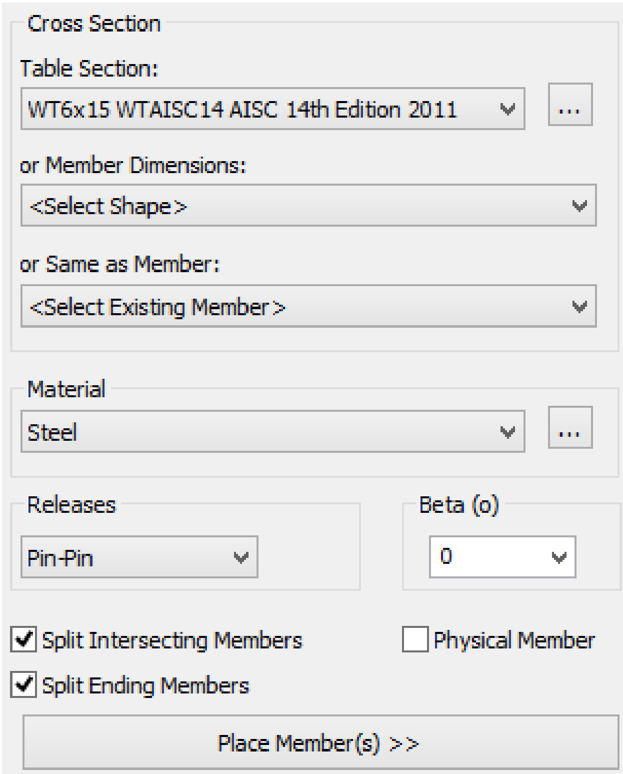
Set ESNAP to Midpoint and not Intersection: Right-click at ESNAP at the bottom of the screen and deselect “Intersection” and select “Midpoint” option.





Click on the icon 

and Place Member form appears.



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.

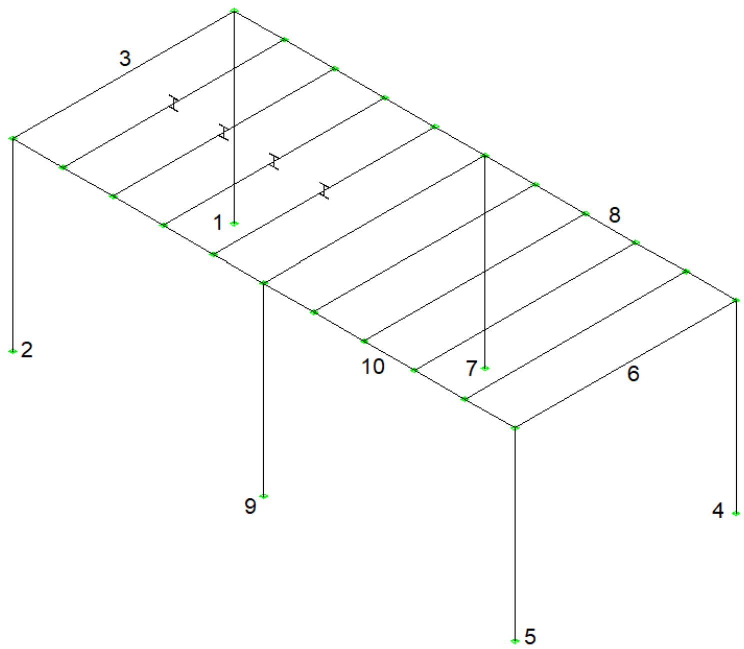
Continue by clicking on Joints at points 5 and 6 and another member is generated.

Continue by clicking on Joint at point 7 and midpoint 8 and another member is generated.

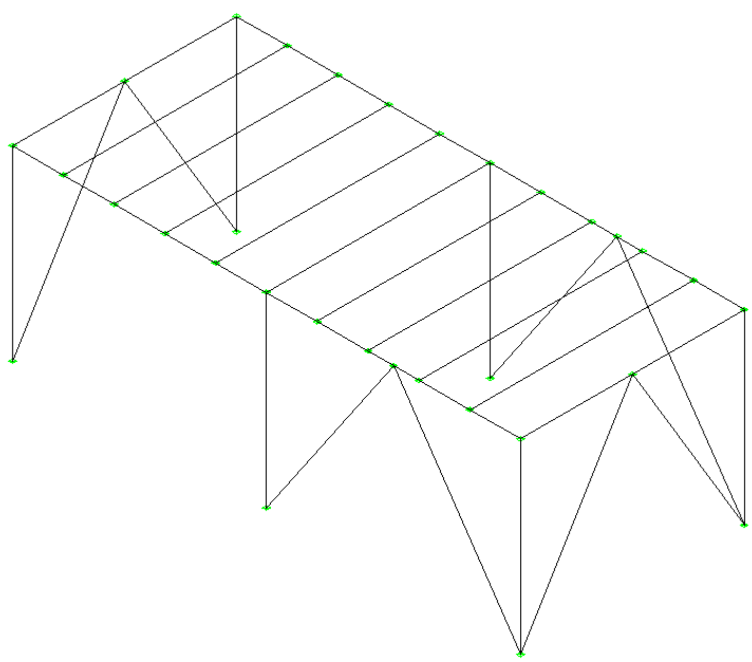
Continue by clicking on Joints at points 4 and 8 and another member is generated.

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.

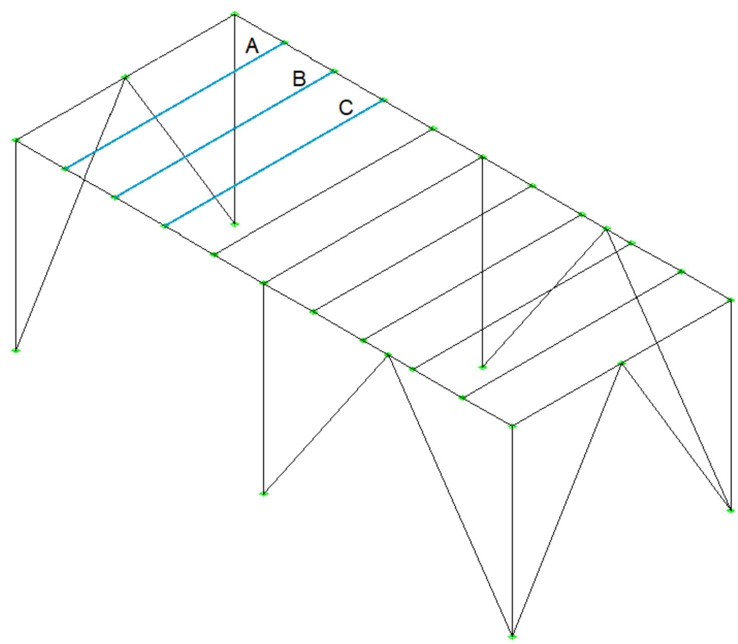


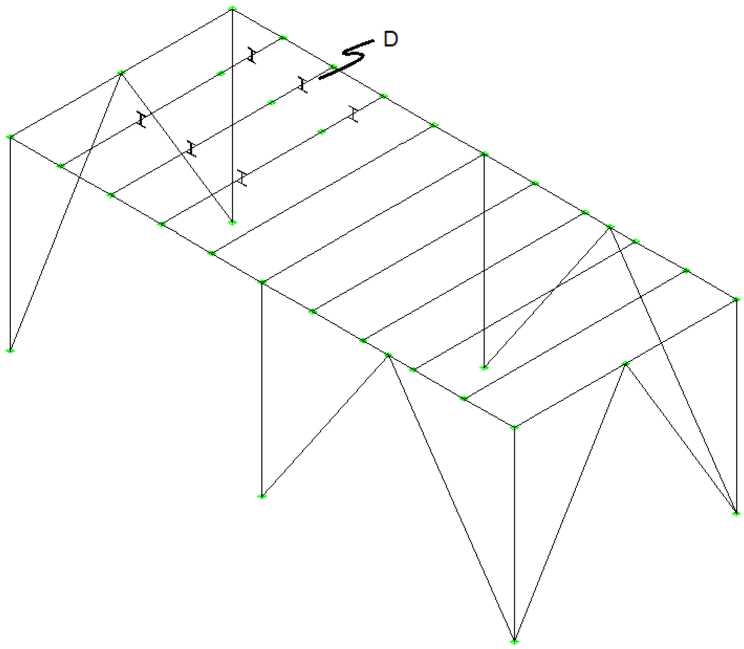
Step #18. Split Girders

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 spliting 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: Spit command can be used in two different ways. If you enter a negative value (eg. -5), the member is spit 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 spit at two parts, at the point that is 5ft (current length uints) from the start of the member. The value that you entered is used to set the distance from start.

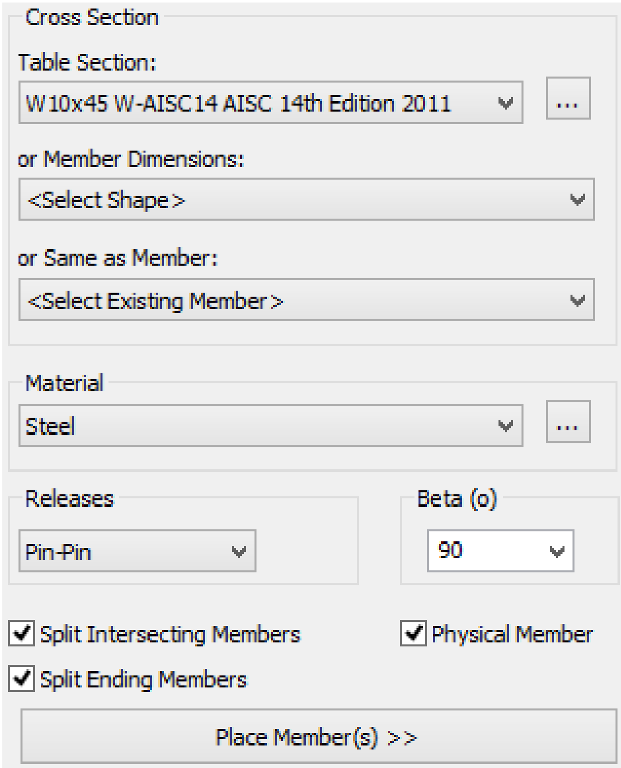




Select member marked in D and delete it, by pressing DEL key and then and click on the icon  to remove the informative cross section shapes in the middle of each member.

Step #19. Add additional members along X

Click on the icon  and Place Member form appears.



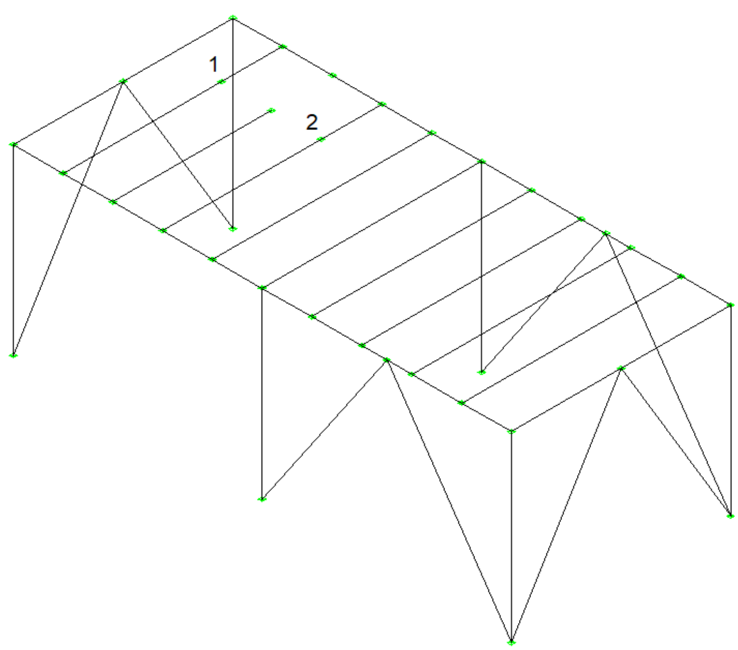
Select *W10x45 W-AISC14 14th Edition 2011* as the cross section and make sure that Material is set to *Steel*, Releases to *Pin-Pin* and Beta angle is *90*.

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.

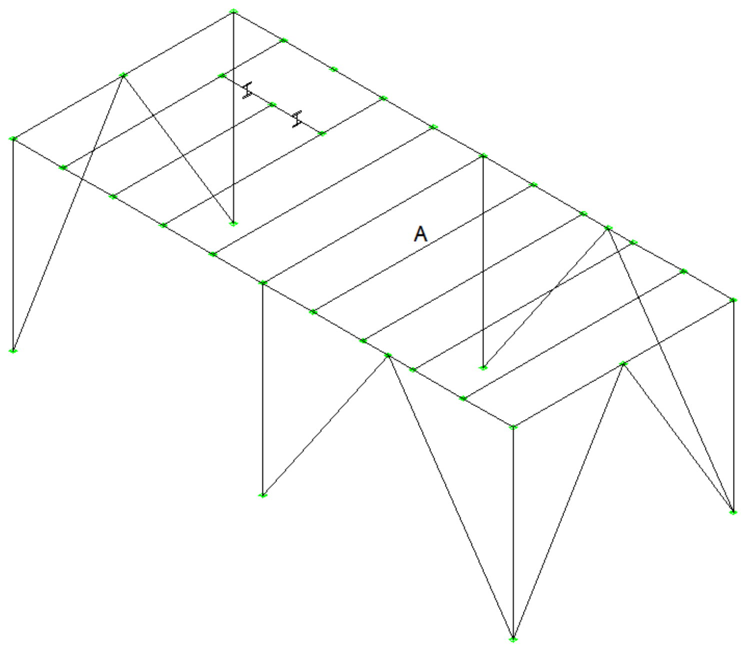
Click on the Joint located at the point 1 of the following image. Click on the Joint at point 2 and two members are generated along X axis.

Press *ESC* to terminate the Generate Beam Command.

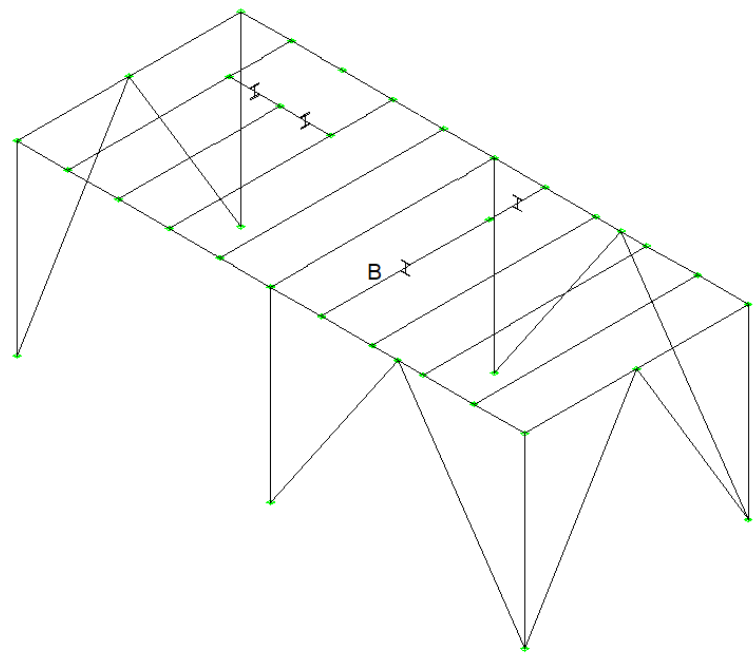


Step #20. Split Girders: Click on the icon 

and click on the member marked in A (of the following image) and then press <ENTER>.



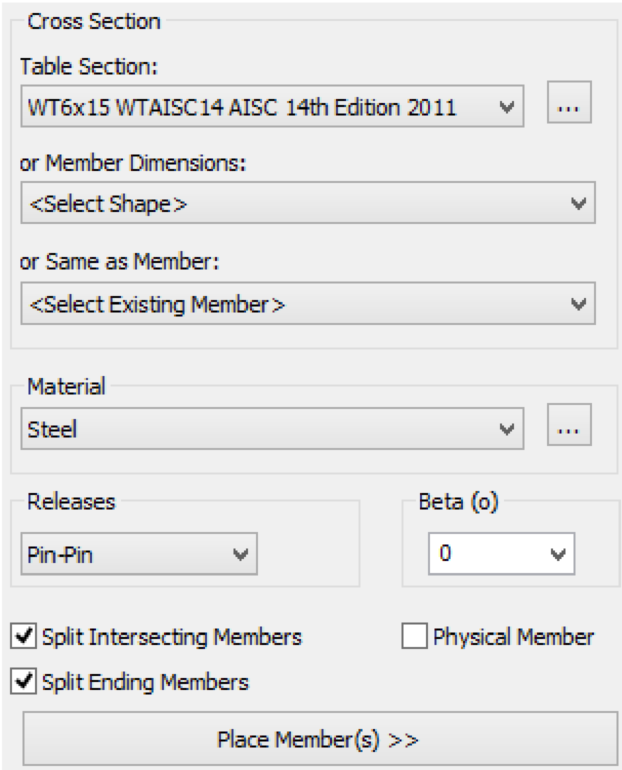
In order to define the *Distance for spliting the member or the number of parts (negative number)*, enter `4.5`, so that the beam will be split at the location 4.5ft from the start of the member (see Notes of Step #18 for more information about the split command).



Again, click on the icon  and click on the member B (of the previous image) and then press <ENTER>. In order to define the *Distance for spliting 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).

Click on the icon  to remove the informative cross section shapes in the middle of each member.

Step #21. Create additional braces: Click on the icon  and Place Member form appears.



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*.

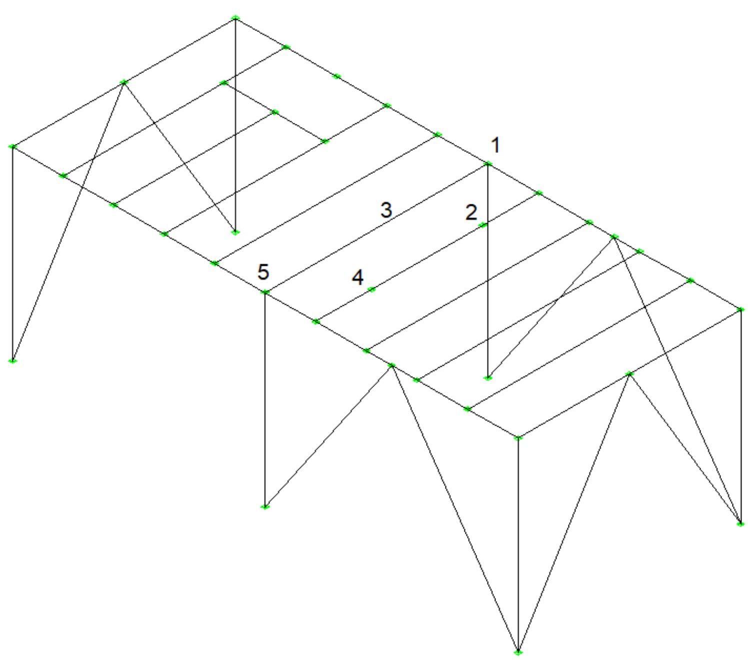
Make sure that the option “Split Intersecting Members” is ON and “Split Ending Members is also ON. Uncheck Physical Member option.

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.

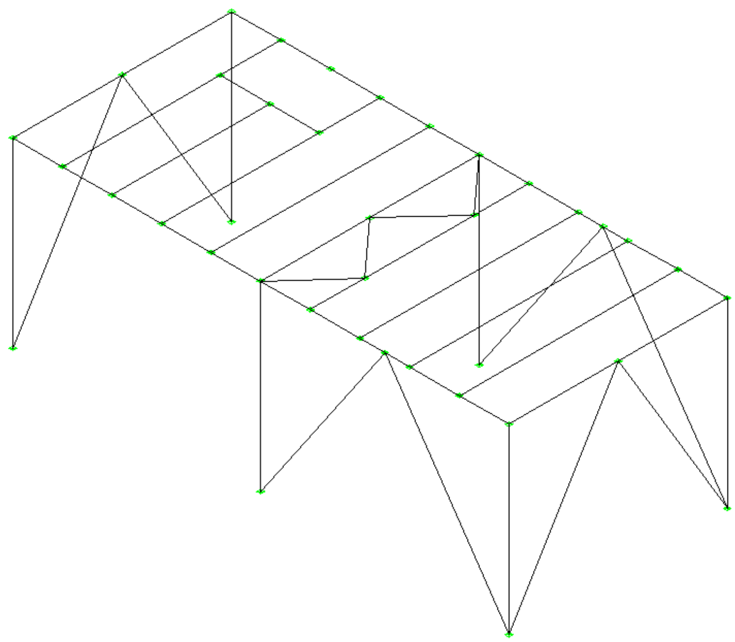
Having the command still active, click on the Joint at point 2 and midpoint 3 and another brace member is generated.

Repeat the same procedure to generate two more members from points 3 to 4 and 4 to 5.



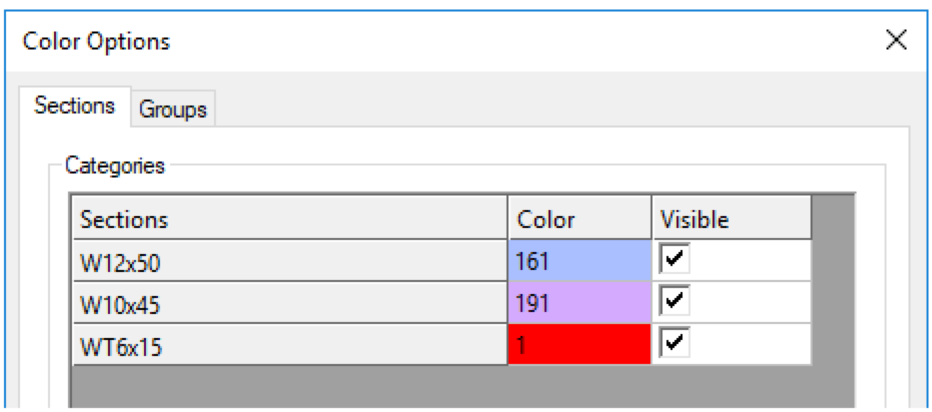
Press *ESC* to terminate the Generate Beam Command and click on the icon 

to remove the informative cross section shapes in the middle of each member. The created model is displayed in the following image.

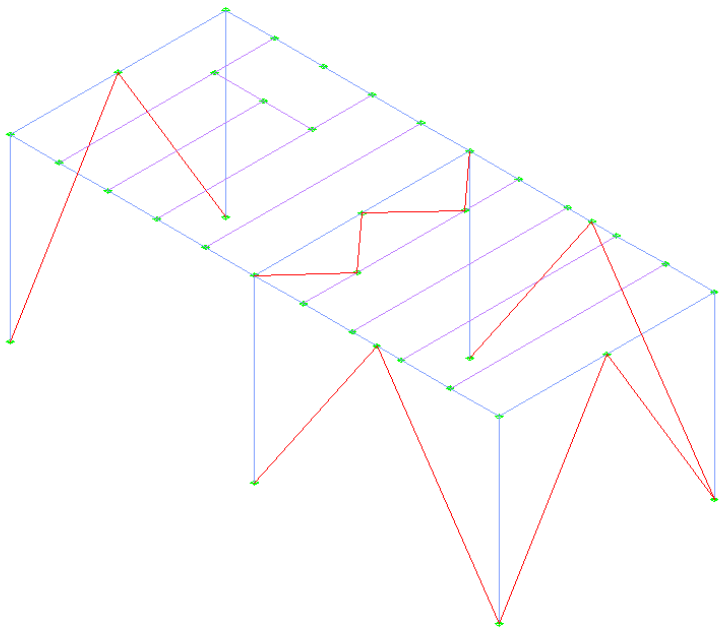


Step #22. Switch to 3D View

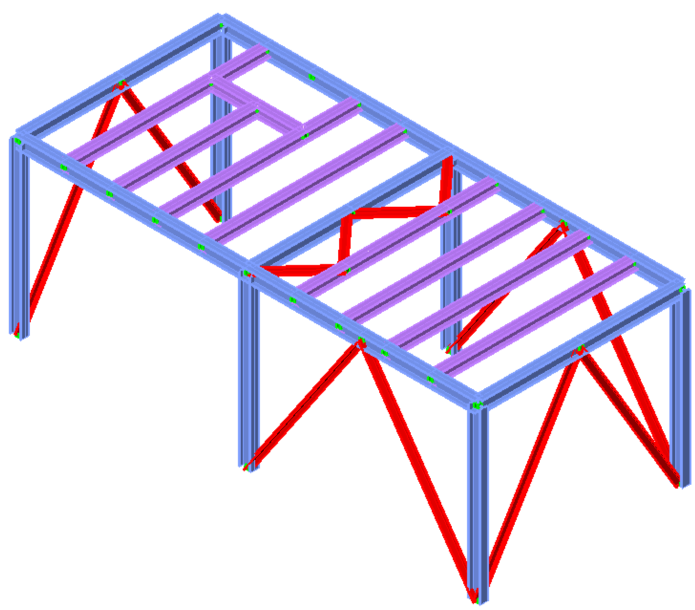
Click on the icon Colors to set different colors for each profile.



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



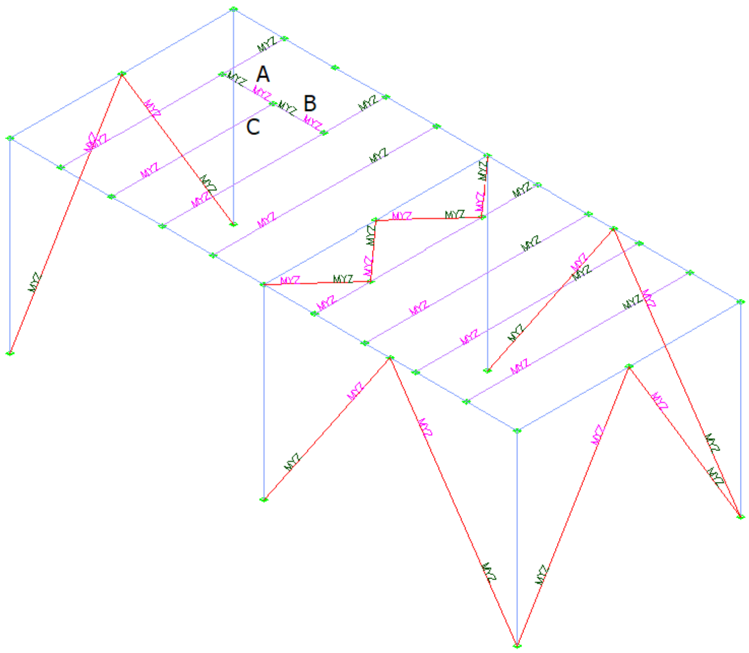
Press the icon to display the 3D solid view of the model, replacing the wireframe view:



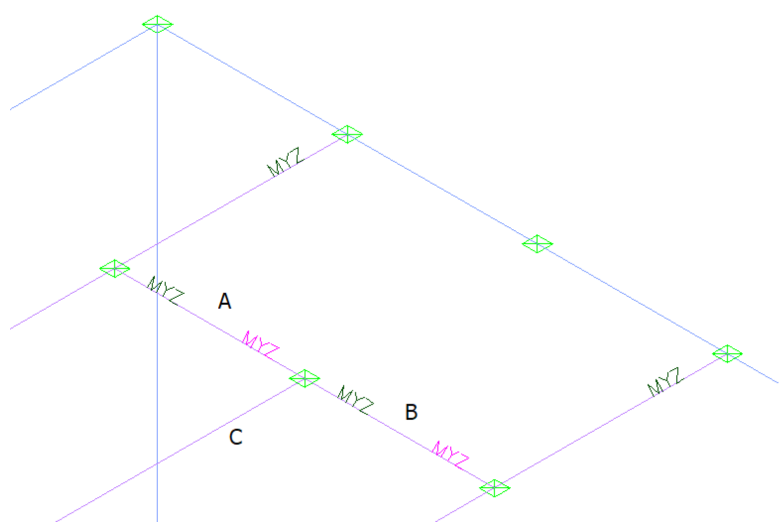
Press the icon Frame to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

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.

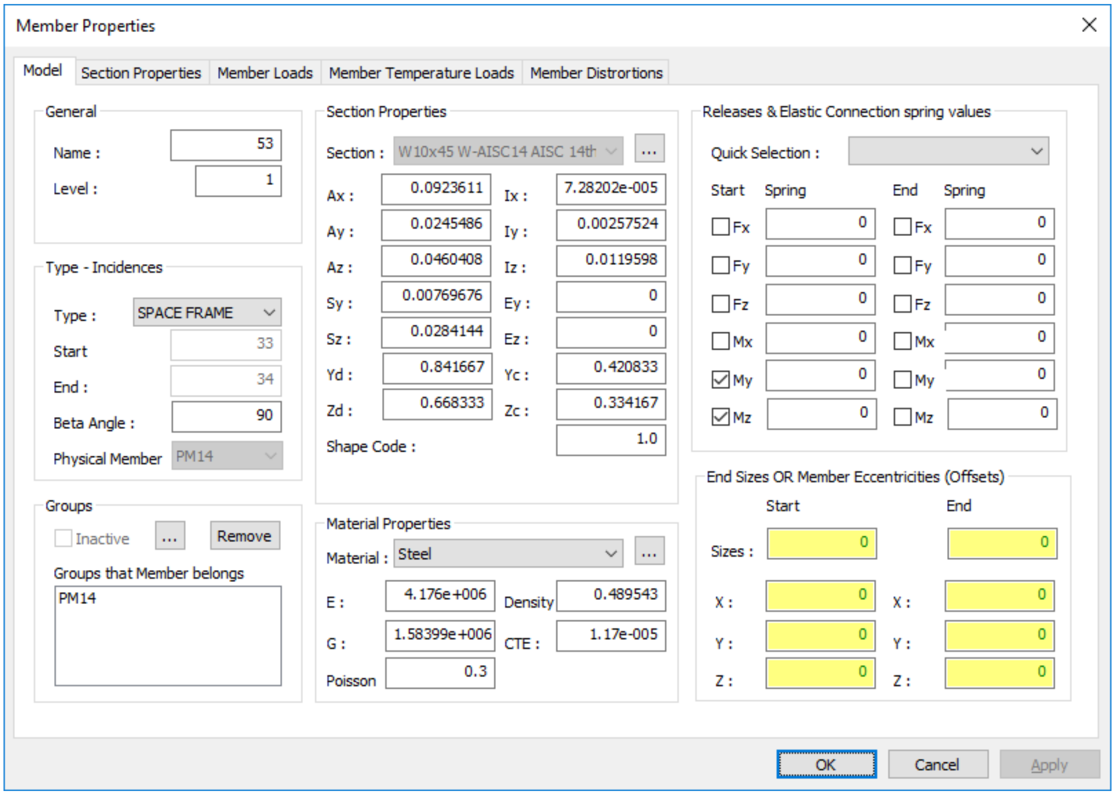
Step #24. Change member releases: Press the Releases icon (in GTS Display Tab of the ribbon) to show releases for each member.



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.

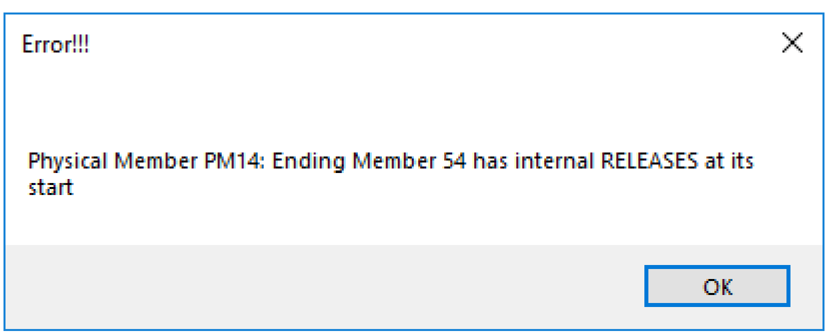


Select **member marked in A** by double clicking on it and uncheck the My and Mz Releases of the member at its End. Note that this member belongs to Physical Member PM14.

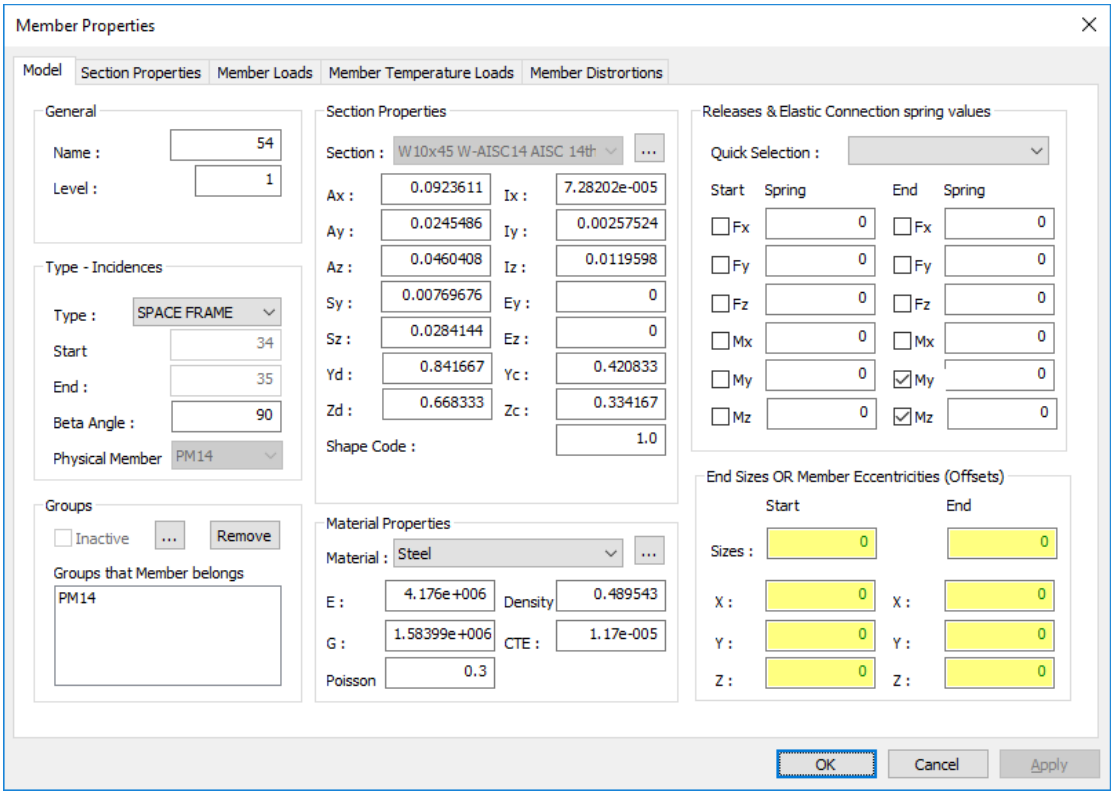


By pressing OK, you get an error message, saying that the Physical Member PM14, has its analytical member 54 (is the member marked in B in the image above) with internal releases at its start. Physical members are now allowed to have internal releases at their internal joints but only at their starting 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.

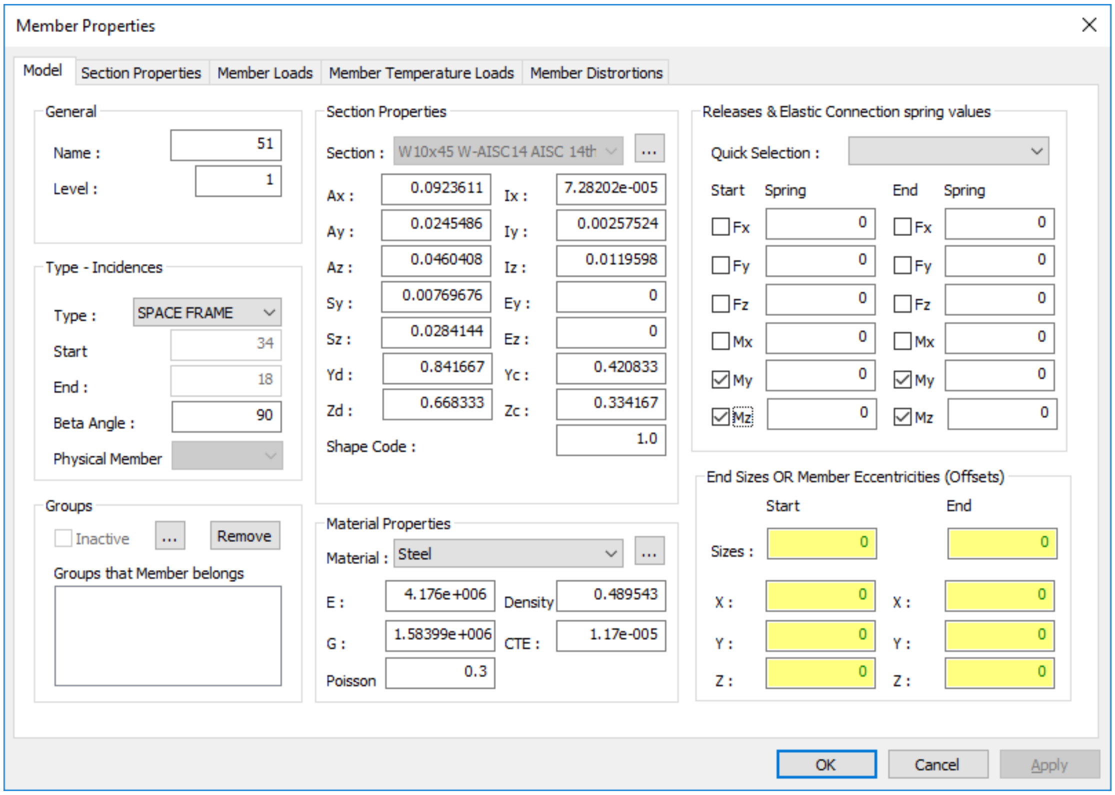


In order to fix the physical member PM14, select **member marked in B** by double clicking on it and uncheck the My and Mz Releases of the member at its Start and press OK.

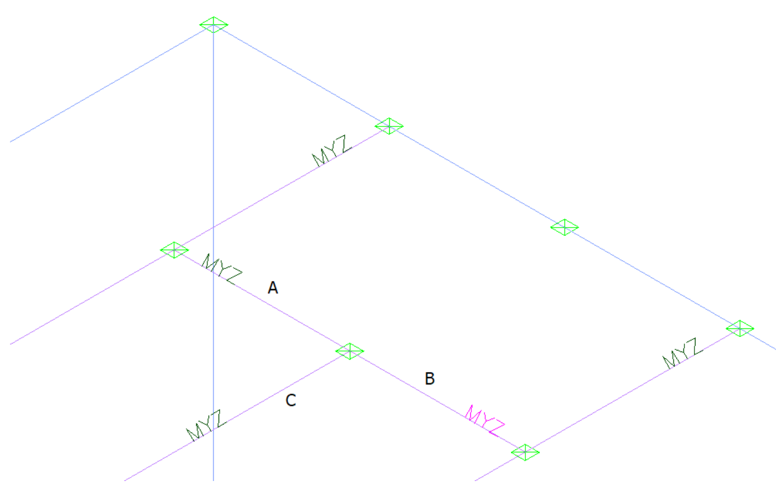


No errors are reported now.

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.



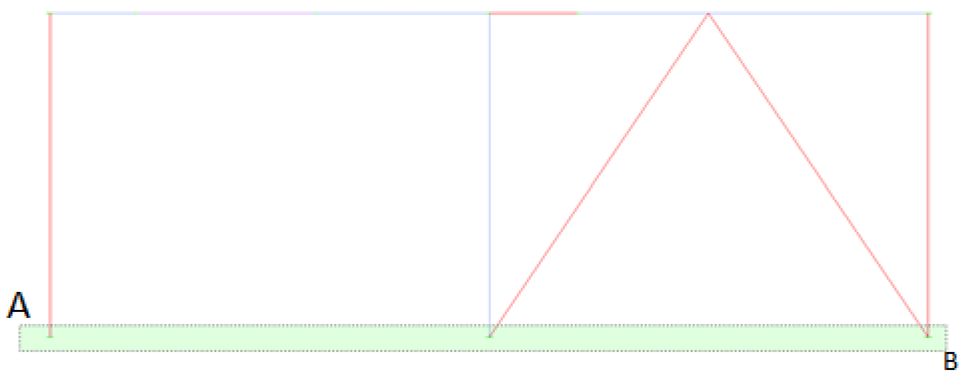
Press the Releases icon to show releases for each member.



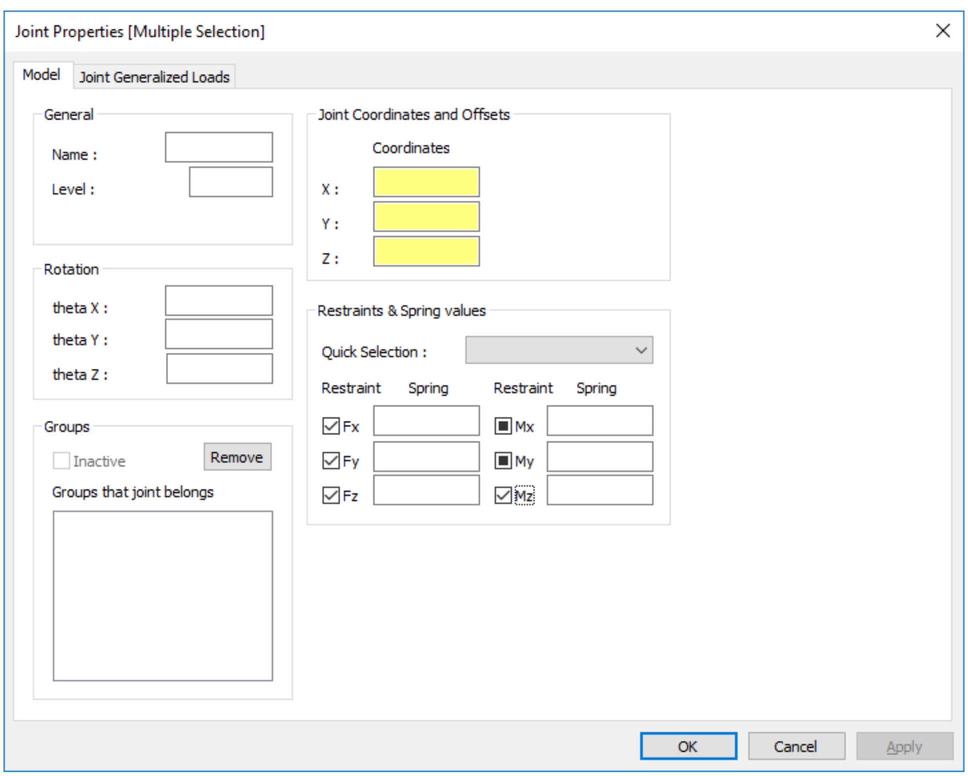
Click on the icon Clear to clear annotations.

5.7. Define supports

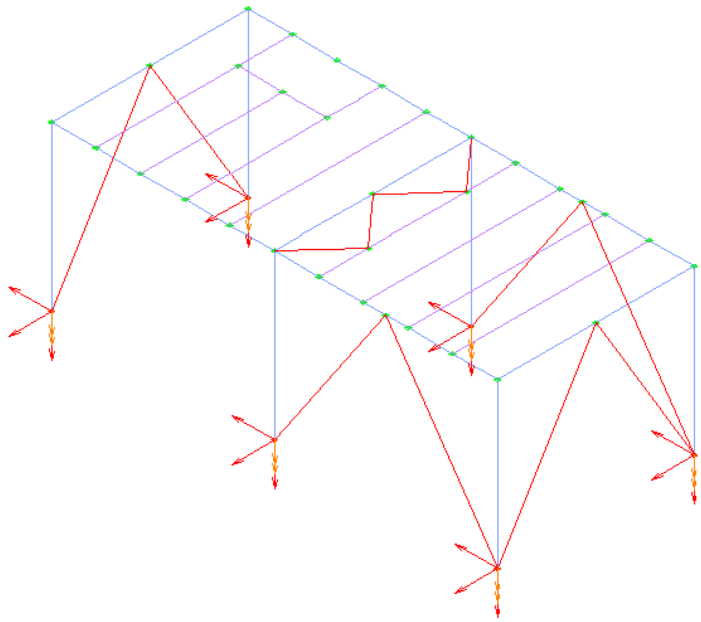
Step #25. Create supports: Use AutoCAD's Cube or BricsCAD's' Chair to change the view to Front. Press the icon Support (in GTS Modeling Tab of the ribbon), select all Joints at the bottom, by clicking at points A and B of the following image, and press ENTER.



At the Joint Properties [Multiple Selection] dialog box select Fx, Fy, Fz and Mz as supports for the selected joints and press OK.



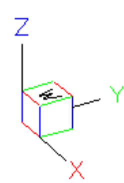
Switch the view to Top Front Right (as in Step #6), and press the icon Joint Supports (in GTS Display Tab of the ribbon) to view Joints supports.



Click on the icon Clear to clear annotations and symbols.

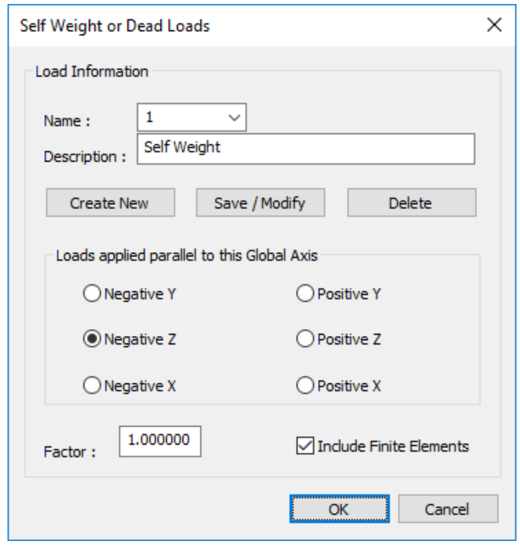
5.8. Define Loads

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.



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.

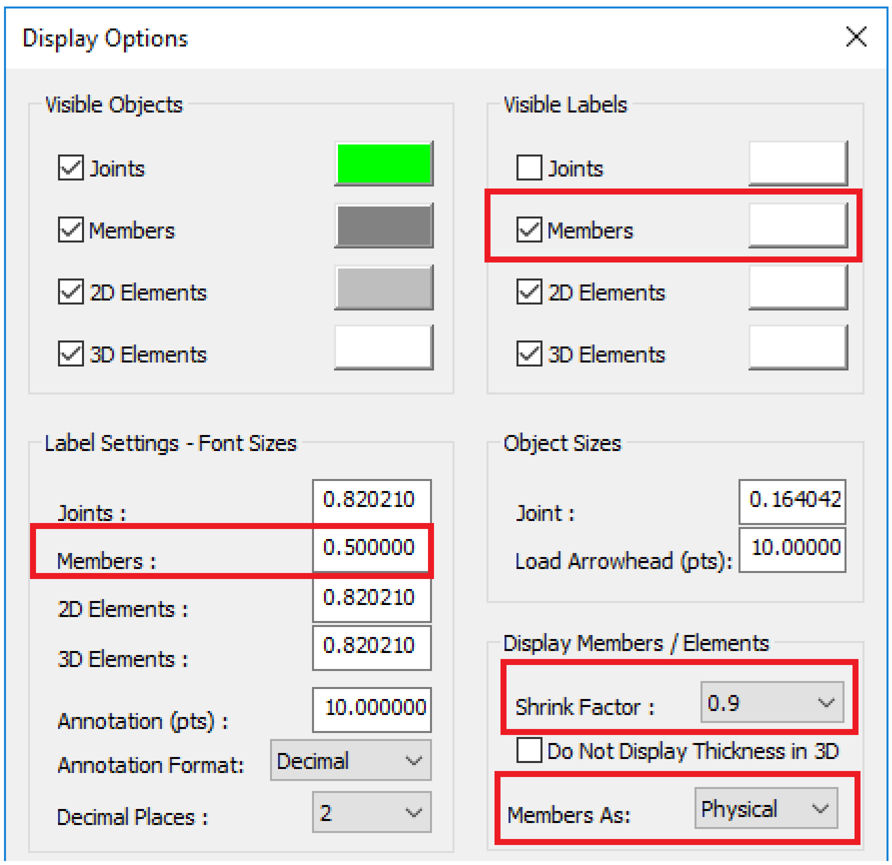
```
Physical Member PM1: OK
Physical Member PM2: OK
Physical Member PM3: OK
Physical Member PM5: OK
Physical Member PM6: OK
Physical Member PM8: OK
Physical Member PM10: OK
```

```
Physical Member PM14: OK
0 Errors Found
```

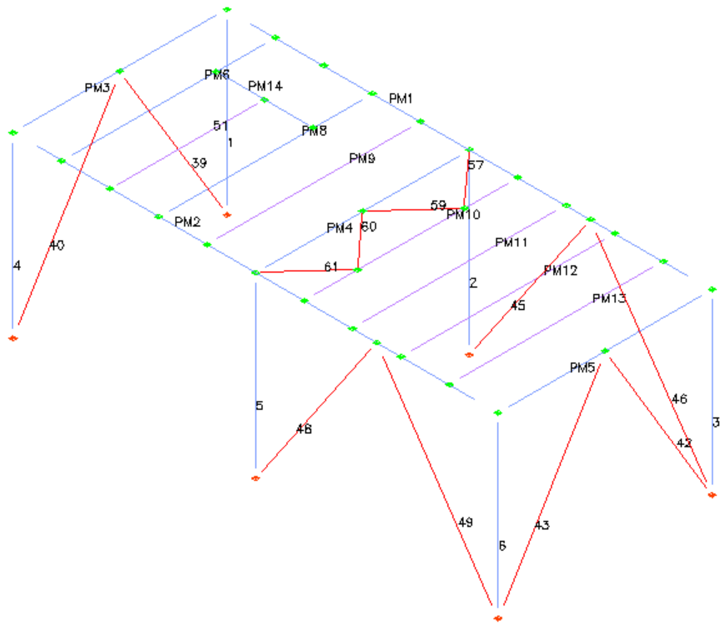
Note: You can also run all other checks of the same drop list, to check for Interference Joints, Duplicate Members, Zero Length Members, Duplicate Names and Database Integrity. You should not get any errors or warnings.

View physical members: Click on the icon 

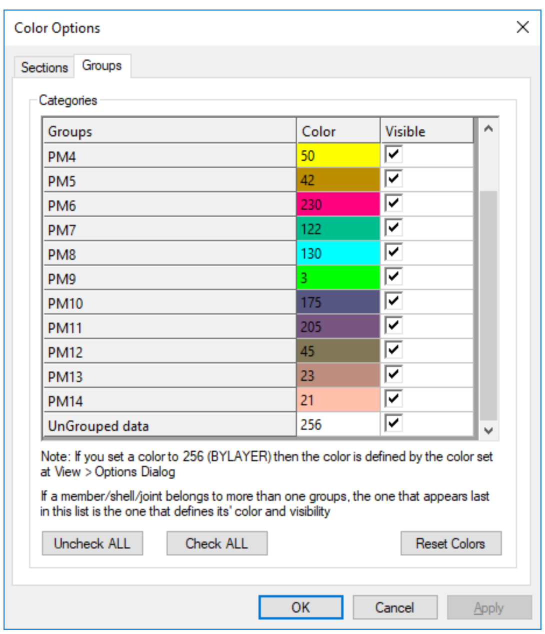
in the ribbon bar and in the Display Options Form check Visible Labels for Members, set Font Size for Members equal to 0.5, Set Shrink Factor to 0.9 and display members as "Physical".

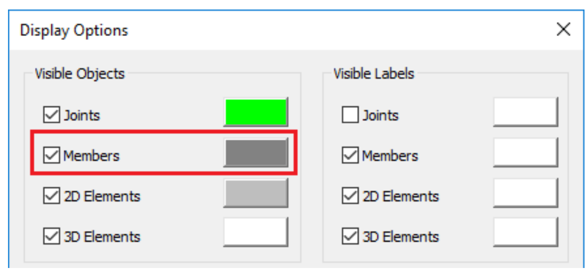


Press OK to close the form and to see the physical members in wireframe view.

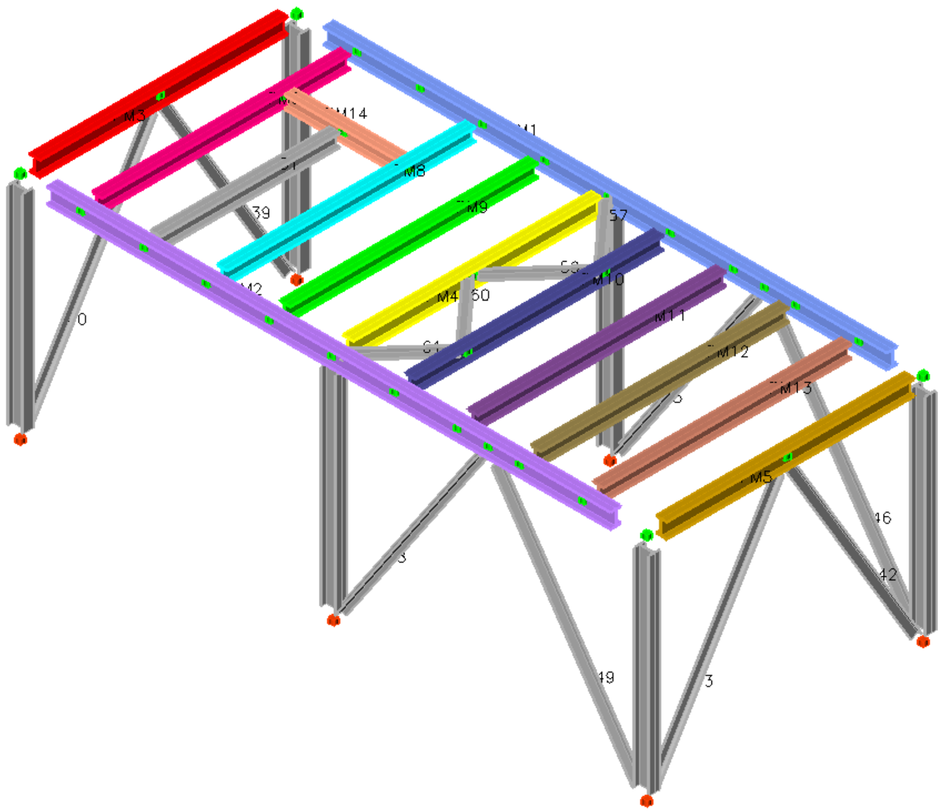


Click on the icon Colors and select the 2nd Tab in order to colorize members by their group. Member do not belonging to any Physical Member are in the "Ungrouped data". Their default color is BYLAYER (256), meaning use the color from Display Options Form, but you can set any other color. Press OK to close the Color Options Form.

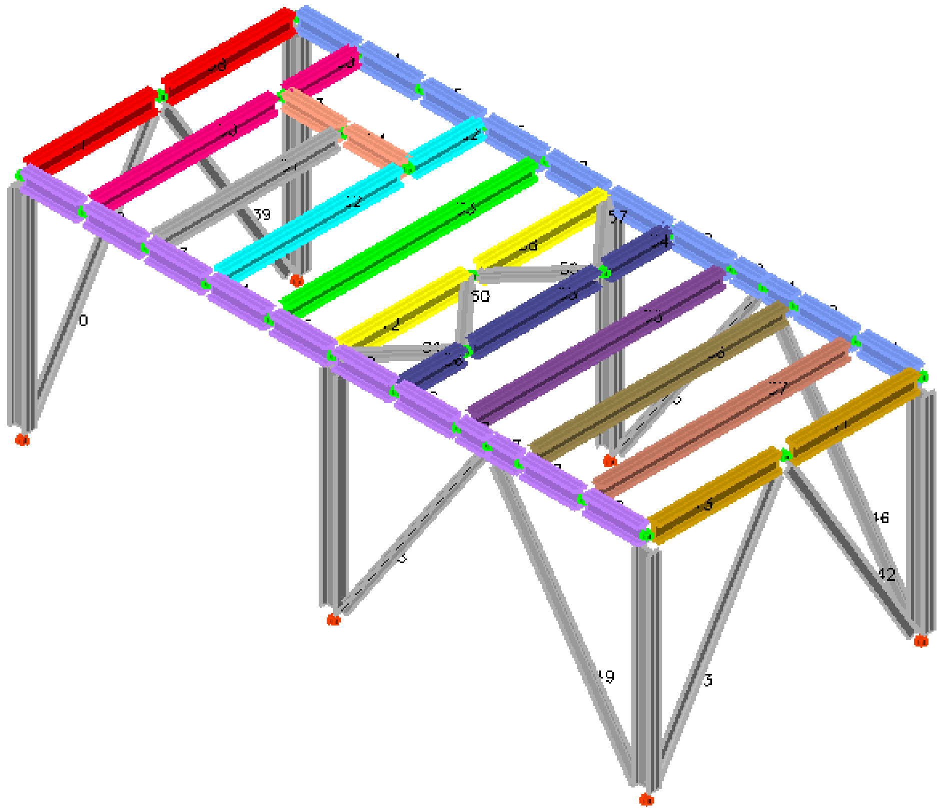




Now each physical member has a different color and you can also press icon to display the 3D solid view, as in the image bellow.



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.



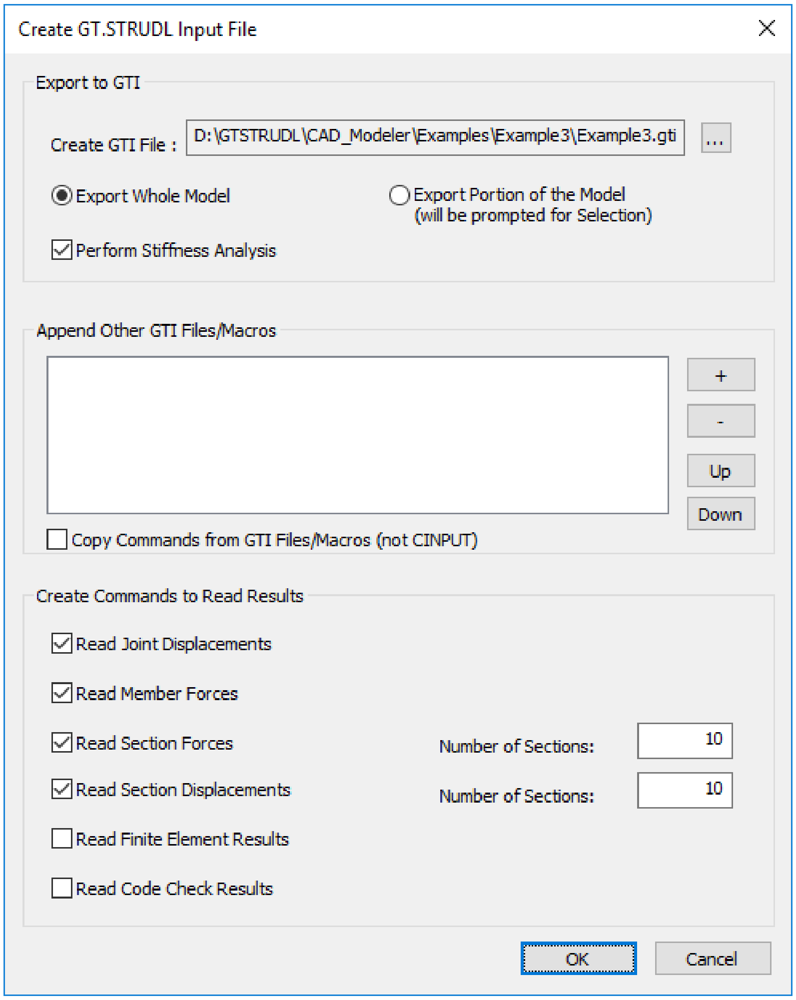
Click on the icon Options and turn off labels for members, set shrink factor back to 1.0, display members as “Analytical” and press OK.

Also cick on the icon Colors and select the 1st Tab in order to colorize members by their section and press OK.

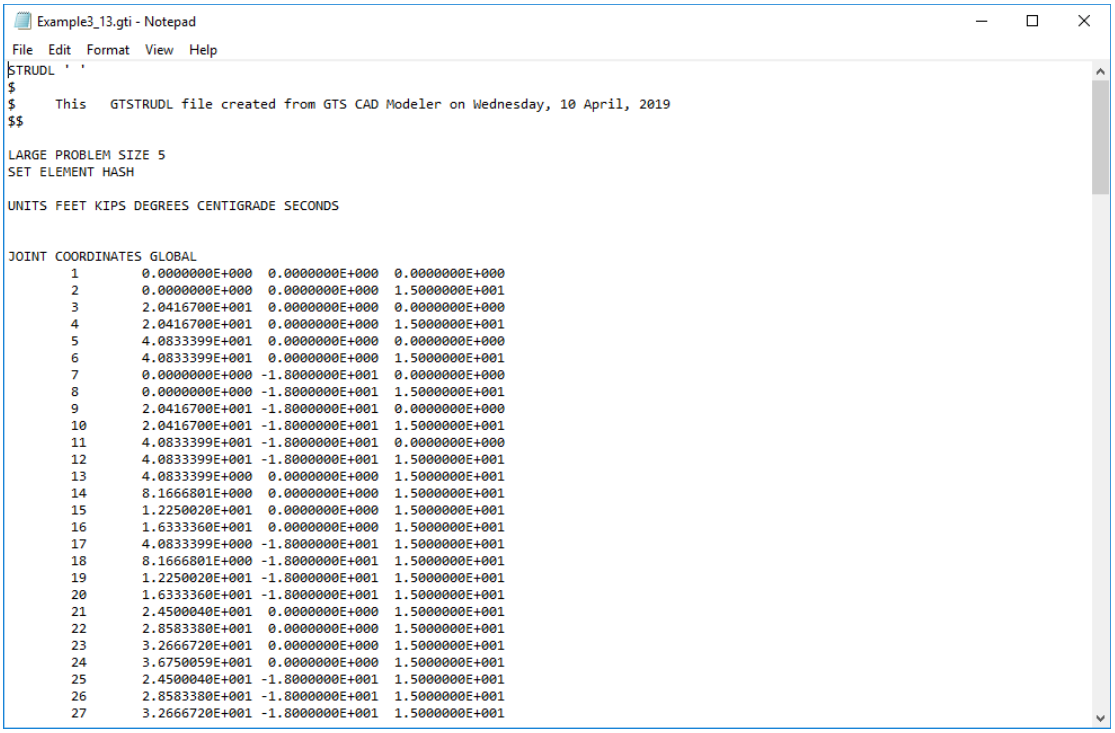
Press the icon Frame to switch back to wireframe view.

Step #29. Create input file: Click on the icon GTI 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

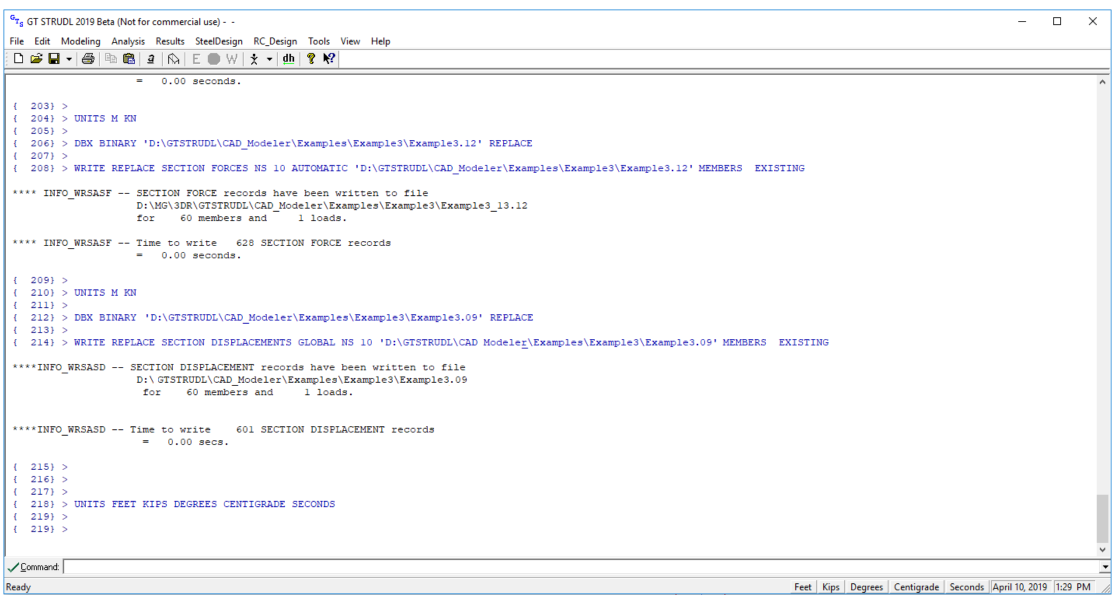


Step #30. View/edit input file: Click on the icon  Edit GTI to view/edit GTI input file.



```
Example3_13.gti - Notepad
File  Edit  Format  View  Help
STRUDL ' '
$
$     This   GTSTRUDL file created from GTS CAD Modeler on Wednesday, 10 April, 2019
$$

LARGE PROBLEM SIZE 5
SET ELEMENT HASH

UNITS FEET KIPS DEGREES CENTIGRADE SECONDS


JOINT COORDINATES GLOBAL
        1        0.0000000E+000  0.0000000E+000  0.0000000E+000
        2        0.0000000E+000  0.0000000E+000  1.5000000E+001
        3        2.0416700E+001  0.0000000E+000  0.0000000E+000
        4        2.0416700E+001  0.0000000E+000  1.5000000E+001
        5        4.0833399E+001  0.0000000E+000  0.0000000E+000
        6        4.0833399E+001  0.0000000E+000  1.5000000E+001
        7        0.0000000E+000 -1.8000000E+001  0.0000000E+000
        8        0.0000000E+000 -1.8000000E+001  1.5000000E+001
        9        2.0416700E+001 -1.8000000E+001  0.0000000E+000
        10       2.0416700E+001 -1.8000000E+001  1.5000000E+001
        11       4.0833399E+001 -1.8000000E+001  0.0000000E+000
        12       4.0833399E+001 -1.8000000E+001  1.5000000E+001
        13       4.0833399E+000  0.0000000E+000  1.5000000E+001
        14       8.1666801E+000  0.0000000E+000  1.5000000E+001
        15       1.2250020E+001  0.0000000E+000  1.5000000E+001
        16       1.6333360E+001  0.0000000E+000  1.5000000E+001
        17       4.0833399E+000 -1.8000000E+001  1.5000000E+001
        18       8.1666801E+000 -1.8000000E+001  1.5000000E+001
        19       1.2250020E+001 -1.8000000E+001  1.5000000E+001
        20       1.6333360E+001 -1.8000000E+001  1.5000000E+001
        21       2.4500040E+001  0.0000000E+000  1.5000000E+001
        22       2.8583380E+001  0.0000000E+000  1.5000000E+001
        23       3.2666720E+001  0.0000000E+000  1.5000000E+001
        24       3.6750059E+001  0.0000000E+000  1.5000000E+001
        25       2.4500040E+001 -1.8000000E+001  1.5000000E+001
        26       2.8583380E+001 -1.8000000E+001  1.5000000E+001
        27       3.2666720E+001 -1.8000000E+001  1.5000000E+001
```

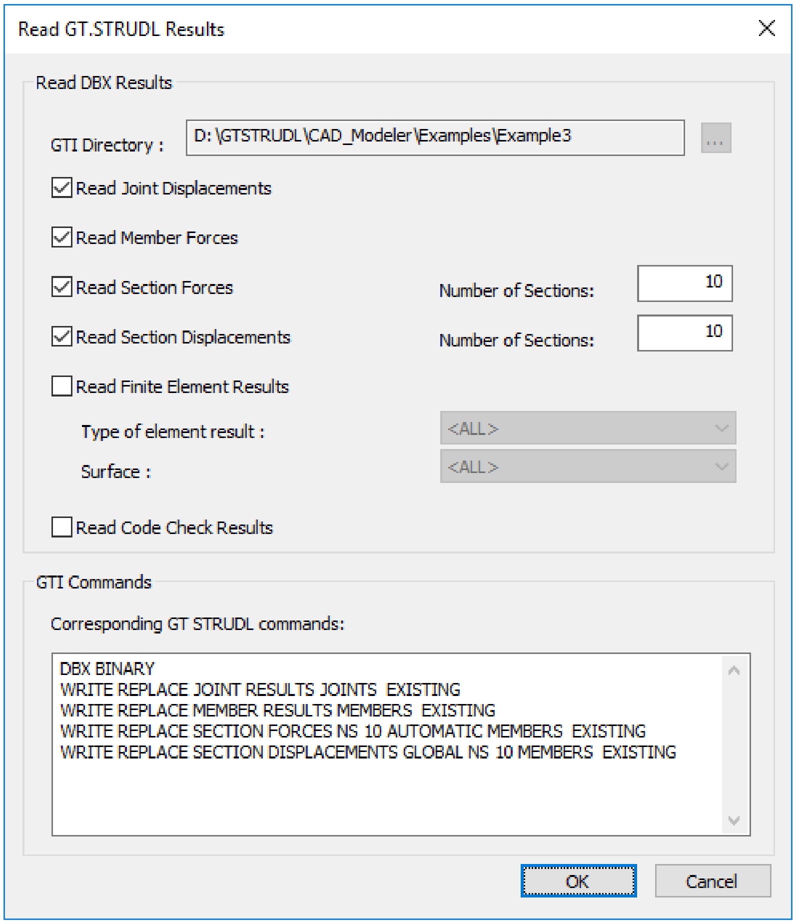
Step #31. Execute GT.STRUDL: Click on the icon to perform the stiffness analysis. GT.STRUDL window will appear with information about analysis.

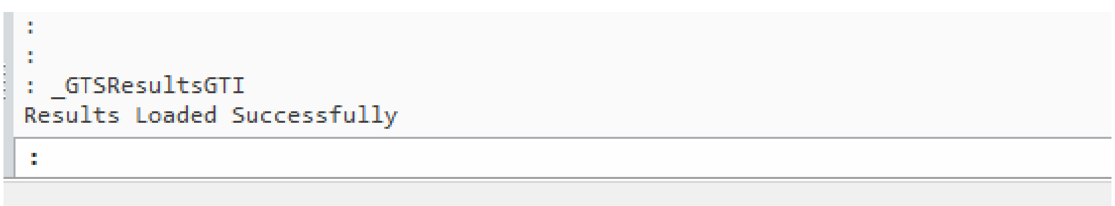


```
GT STRUDL 2019 Beta (Not for commercial use) -  -
File  Edit  Modeling  Analysis  Results  SteelDesign  RC_Design  Tools  View  Help
                   =   0.00 seconds.

{  203} >
{  204} > UNITS M KN
{  205} >
{  206} > DBX BINARY 'D:\GTSTRUDL\CAD_Modeler\Examples\Example3\Example3.12' REPLACE
{  207} >
{  208} > WRITE REPLACE SECTION FORCES NS 10 AUTOMATIC 'D:\GTSTRUDL\CAD_Modeler\Examples\Example3\Example3.12' MEMBERS  EXISTING

**** INFO_WRSASF -- SECTION FORCE records have been written to file
                   D:\MG\3DR\GTSTRUDL\CAD_Modeler\Examples\Example3\Example3_13.12
                   for    60 members and     1 loads.

**** INFO_WRSASF -- Time to write   628 SECTION FORCE records
                   =   0.00 seconds.

{  209} >
{  210} > UNITS M KN
{  211} >
{  212} > DBX BINARY 'D:\GTSTRUDL\CAD_Modeler\Examples\Example3\Example3.09' REPLACE
{  213} >
{  214} > WRITE REPLACE SECTION DISPLACEMENTS GLOBAL NS 10 'D:\GTSTRUDL\CAD Modeler\Examples\Example3\Example3.09' MEMBERS  EXISTING

****INFO_WRSASD -- SECTION DISPLACEMENT records have been written to file
                   D:\ GTSTRUDL\CAD_Modeler\Examples\Example3\Example3.09
                    for    60 members and     1 loads.


****INFO_WRSASD -- Time to write    601 SECTION DISPLACEMENT records
                    =   0.00 secs.

{  215} >
{  216} >
{  217} >
{  218} > UNITS FEET KIPS DEGREES CENTIGRADE SECONDS
{  219} >
{  219} >
Command:
Ready                         Feet | Kips | Degrees | Centigrade | Seconds | April 10, 2019 | 1:29 PM
```

5.10. Read analysis results

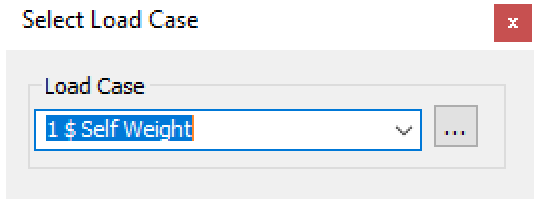
Step #32. Read GT.STRUDL results: Click on the icon Read Results to read GT.STRUDL results.

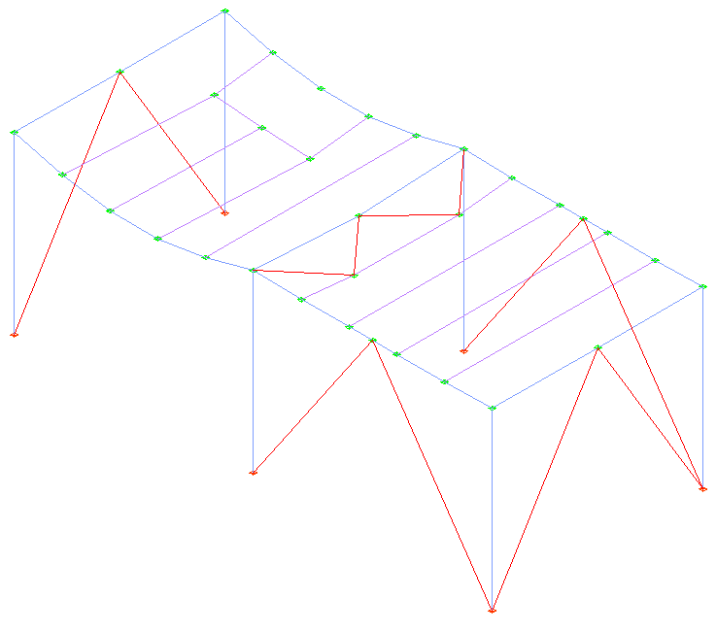




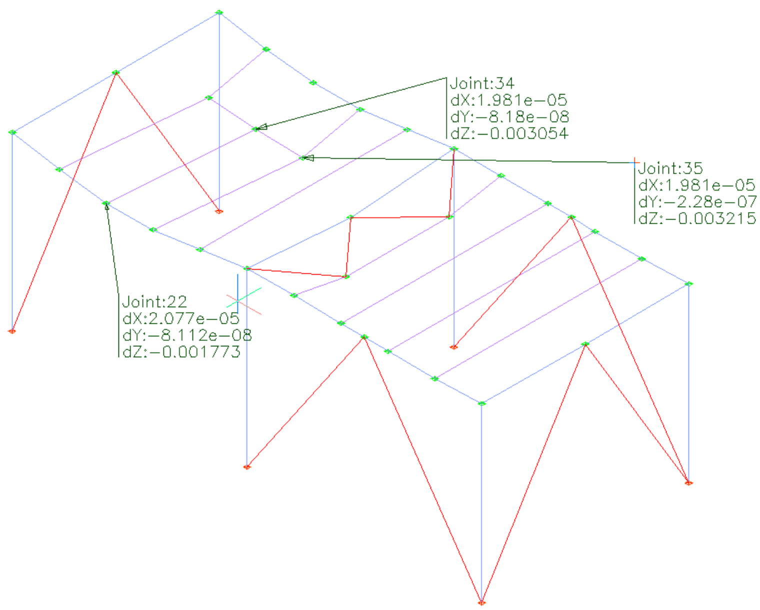
5.11. Display analysis results

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.



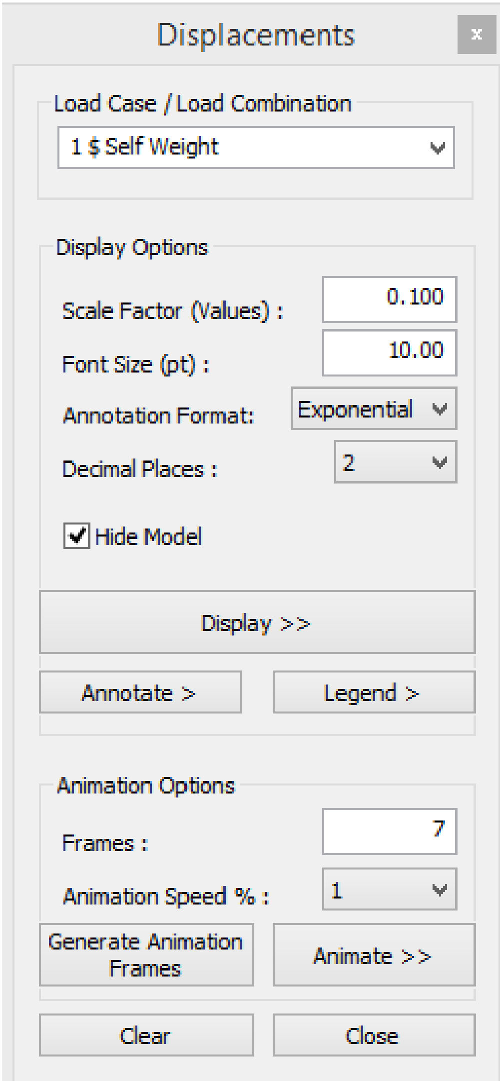


Step #34. Annotate displacements: Click on the icon  Annotate Displacements, under the "Deformation" Drop Button, located in "Display Results" at Ribbon Area, to annotate displacements of the deformed model at specific Joints of interest.



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").



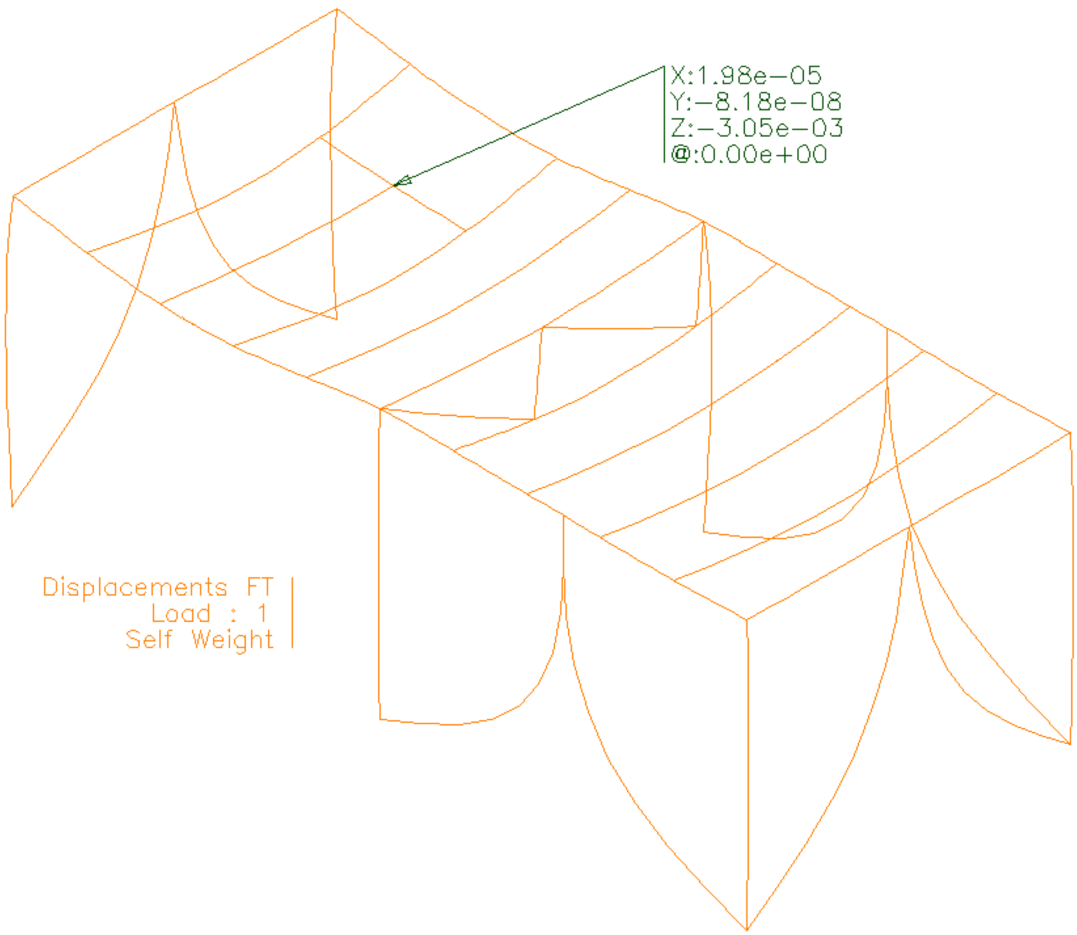
Select:

- *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.

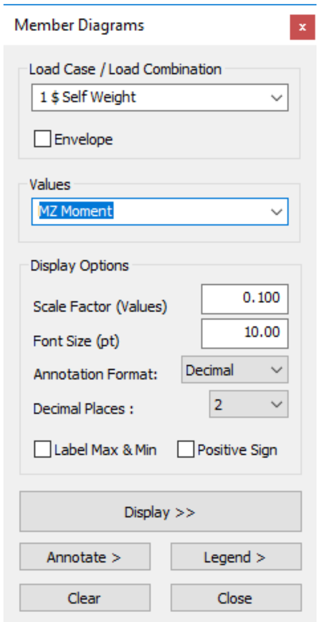
Press on "Legend >" and click at any part of the screen to place the legend of the diagram.

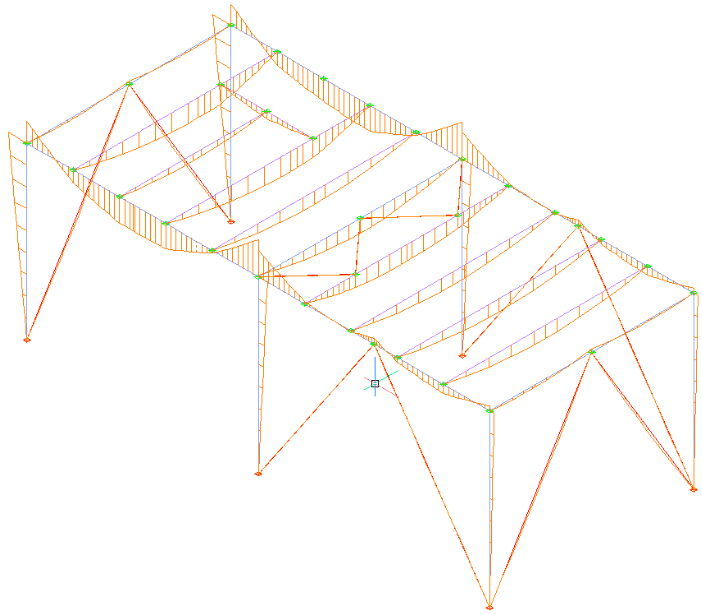


In order to display the animation, press “Generate Animation Frames” and then “Animate >>”. To terminate the animation press “Stop” button.

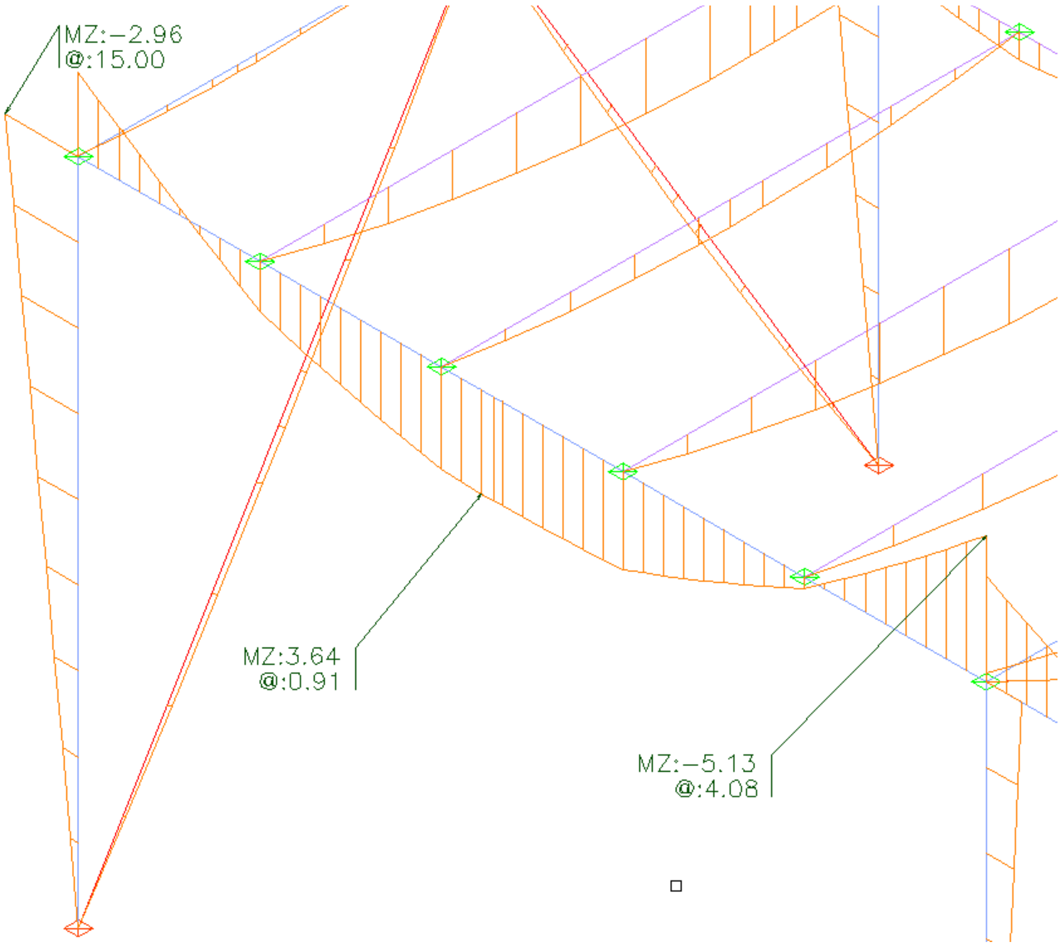
In order to exit the command, uncheck “Hide Model”, press “Clear” Button and “Close”.

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”.





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	0.0000170
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

File Edit Columns Filter Sort Units Help

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

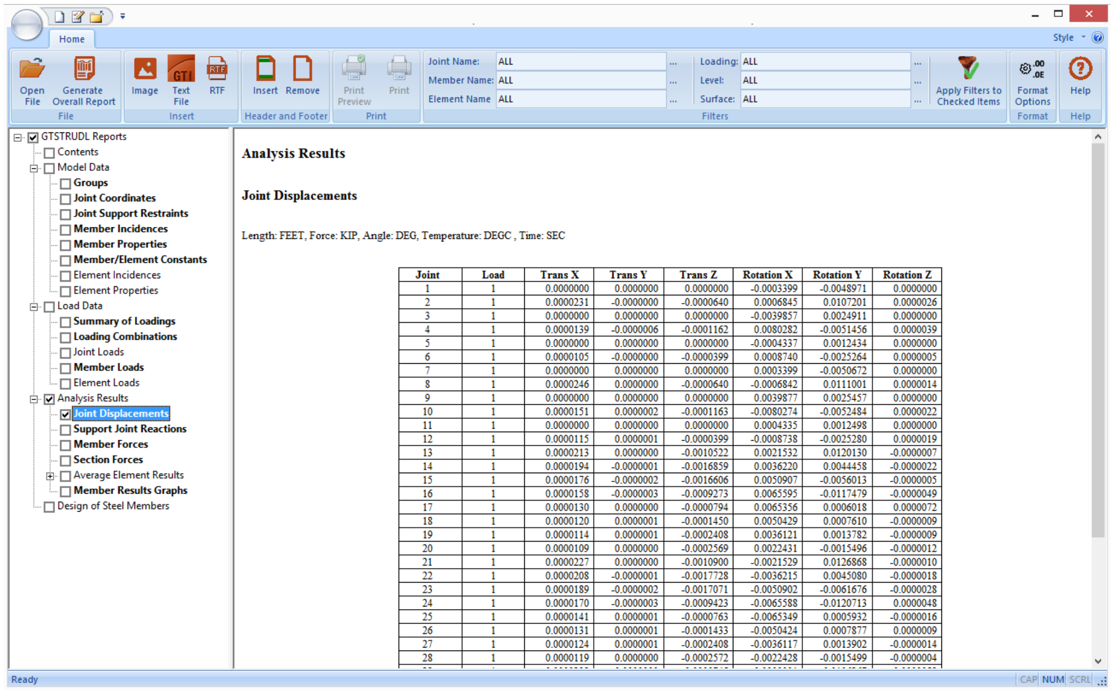
Click on the icon Reactions to view reaction results in datasheets.

GTSTRUDL - Joint Reactions Datasheet, display Units: Feet Kips

File Edit Columns Filter Sort Units Help

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.



Analysis Results

Joint Displacements

Length: FEET, Force: KIP, Angle: DEG, Temperature: DEGC , Time: SEC

Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y	Rotation Z
1	1	0.0000000	0.0000000	0.0000000	-0.0003399	-0.0048971	0.0000000
2	1	0.0000231	-0.0000000	-0.0000640	0.0006845	0.0107201	0.0000026
3	1	0.0000000	0.0000000	0.0000000	-0.0039857	0.0024911	0.0000000
4	1	0.0000139	-0.0000006	-0.0001162	0.0080282	-0.0051456	0.0000039
5	1	0.0000000	0.0000000	0.0000000	-0.0004337	0.0012434	0.0000000
6	1	0.0000105	-0.0000000	-0.0000399	0.0008740	-0.0025264	0.0000005
7	1	0.0000000	0.0000000	0.0000000	0.0003399	-0.0050672	0.0000000
8	1	0.0000246	0.0000000	-0.0000640	-0.0006842	0.0111001	0.0000014
9	1	0.0000000	0.0000000	0.0000000	0.0039877	0.0025457	0.0000000
10	1	0.0000151	0.0000002	-0.0001163	-0.0080274	-0.0052484	0.0000022
11	1	0.0000000	0.0000000	0.0000000	0.0004335	0.0012498	0.0000000
12	1	0.0000115	0.0000001	-0.0000399	-0.0008738	-0.0025280	0.0000019
13	1	0.0000213	0.0000000	-0.0010522	0.0021532	0.0120130	-0.0000007
14	1	0.0000194	-0.0000001	-0.0016859	0.0036220	0.0044458	-0.0000022
15	1	0.0000176	-0.0000002	-0.0016606	0.0050907	-0.0056013	-0.0000005
16	1	0.0000158	-0.0000003	-0.0009273	0.0065595	-0.0117479	-0.0000049
17	1	0.0000130	0.0000000	-0.0000794	0.0065356	0.0006018	0.0000072
18	1	0.0000120	0.0000001	-0.0001450	0.0050429	0.0007610	-0.0000009
19	1	0.0000114	0.0000001	-0.0002408	0.0036121	0.0013782	-0.0000009
20	1	0.0000109	0.0000000	-0.0002569	0.0022431	-0.0015496	-0.0000012
21	1	0.0000227	0.0000000	-0.0010900	-0.0021529	0.0126868	-0.0000010
22	1	0.0000208	-0.0000001	-0.0017728	-0.0036215	0.0045080	-0.0000018
23	1	0.0000189	-0.0000002	-0.0017071	-0.0050902	-0.0061676	-0.0000028
24	1	0.0000170	-0.0000003	-0.0009423	-0.0065588	-0.0120713	0.0000048
25	1	0.0000141	0.0000001	-0.0000763	-0.0065349	0.0005932	-0.0000016
26	1	0.0000131	0.0000001	-0.0001433	-0.0050424	0.0007877	0.0000009
27	1	0.0000124	0.0000001	-0.0002408	-0.0036117	0.0013902	-0.0000014
28	1	0.0000119	0.0000000	-0.0002572	-0.0022428	-0.0015499	-0.0000004

6. Appendix – List of Commands

Command	Icon	Menu	Command Prompt	Link
Units	Units	*GTS Modeling>Units*	`GTSUnits`	2.6.1
Materials	Materials	*GTS Modeling>Materials*	`GTSMaterials`	2.6.2
Sections	Sections	*GTS Modeling>Cross Sections>Table*	`GTSParams`	2.6.9
Prismatic Sections	-	*GTS Modeling>Cross Sections>Prismatic*	`GTSPrismatic`	2.6.9
Levels	Levels	*GTS Modeling>Levels*	`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	*GTS Display>All Levels ON*	`GTSSetAllVisible`	2.6.3
Grid	Grid	*GTS Modeling>Grid>Create*	`GTSGrid`	2.6.4
Change Grid	Change Grid	*GTS Modeling>Grid>Change*	`GTSGridChange`	2.6.4
Generate Joint	Generate	*GTS Modeling>Joint>Generate Joint*	`GTSJoint`	2.6.5
At Level (Joint)	At Level	*GTS Modeling>Joint>Generate Joint at Level*	`GTSJointLevel`	2.6.5
Find (Joint)	Find Joint	*GTS Modeling>Joint>Find*	`GTSFJID`	2.6.6
Support	Support	*GTS Modeling>Joint>Support*	`GTSJointSupport`	2.6.7
Change (Joint)	Change Joint	*GTS Modeling>Joint>Change*	`GTSJointChange`	2.6.8
Generate (Member)	Generate	*GTS Modeling> Members>Generate Beam Members*	`GTSBeam`	2.6.10

Vertical (Member)	Vertical	*GTS Modeling> Member>Generate Vertical Member*	GTSColumn	2.6.10
Find (Member)	Find Member	*GTS Modeling>Member>Find*	GTSFMID	2.6.11
Split (Member)	Split	*GTS Modeling>Member>Split Member*	GTSSplitMember	2.6.12
Split to Crossing Members	Split to Members	*GTS Modeling>Member>Split to Crossing Members*	GTSSplitToMembers	2.6.13
Merge (Member)	Merge	*GTS Modeling> Member>Merge Members*	GTSMergeMembers	2.6.14
Change (Member)	Change Member	*GTS Modeling> Member>Change*	GTSBeamChange	2.6.15
Filter (Members)	Filter	*GTS Modeling> Member>Filter*	GTSFilterMembers	2.6.16
Generate Quad	Quad	*GTS Modeling>Shell>Generate quad at joints*	GTSShell	2.6.17
Generate Triangle	Triangle	*GTS Modeling>Shell>Generate triangle at joints*	GTSShellT	2.6.17
Reverse Incidence Order	Reverse	*GTS Modeling>Shell> Reverse Incidence Order*	GTSShellReverse	2.6.18
Find (Shell)	Find Shell	*GTS Modeling>Shell>Find*	GTSFeid	2.6.19
Change (Shell)	Change Shell	*GTS Modeling>Shell>Generate triangle at joints*	GTSShellChange	2.6.20
Locate Duplicates	Joints Duplicates	*GTS Modeling>Checks>Joints Duplicates*	GTSCheckDuplicateJoints	2.6.21
Locate Floating	Joints Floatings	*GTS Modeling>Checks>Joints Floatings*	GTSCheckFloatingJoints	2.6.22
Joints Interference	Joints Interference	*GTS Modeling>Checks>Joints Interference*	GTSCheckInteferenceJoints	2.6.23
Members Duplicates	Members Duplicates	*GTS Modeling>Checks>Members Duplicates*	GTSCheckDuplicateMembers	2.6.24
Members Zero Length	Members Zero Length	*GTS Modeling>Checks>Members Zero Length*	GTSCheckMembersZeroLength	2.6.25

Physical Members	Physical Members	*GTS Modeling>Checks>Physical Members*	`GTSCheckPhysicalMembers`	2.6.26
Shells Duplicates	Shells Duplicates	*GTS Modeling>Checks>Shells Duplicates*	`GTSCheckDuplicateShells`	2.6.27
Names Duplicates	Names Duplicates	*GTS Modeling>Checks>Names Duplicates*	`GTSCheckNames`	2.6.28
Renumber Names	6, 2, 8, 1... 1, 2, 6, 9... Renumber Names	*GTS Modeling>Checks>Renumber Names*	`GTSRenumber`	2.6.29
Database Integrity	Database Integrity	*GTS Modeling>Checks>Database Integrity*	`GTSCheckDatabase`	2.6.30
1D Curve (Meshing)	1D Curve	*GTS Modeling>Mesh Generation>1D Along Line or Curve or Circle*	`GTSMesh1D`	2.6.31
2D 2Curves (Meshing)	2D 2Curves	*GTS Modeling>Mesh Generation>2D Between 2 Lines or Curves*	`GTSMesh2D2L`	2.6.32
2D 4Curves (Meshing)	2D 4Curves	*GTS Modeling>Mesh Generation>2D Between 4 Lines or Curves*	`GTSMesh2D4L`	2.6.33
2D Area (Meshing)	2D Area	*GTS Modeling>Mesh Generation>2D Between 4 Lines or Curves*	`GTSMesh2DPoly`	2.6.34
3D Extrude (Meshing)	3D Extrude	*GTS Modeling>Mesh Generation>3D Extrude PolyLine*	`GTSExtrudePoly`	2.6.35
2D 3Curves (Meshing)	3D 3Curves	GTS Modeling>Mesh Generation>3D Between 3 Lines or Curves	`GTSMesh3D3L`	2.6.36
Array 3D Advanced	Array 3D Advanced	-	`GTSArray3D`	2.6.37
Soil Springs	Soil Springs	*GTS Modeling>Soil Springs*	`GTSFoundationSprings`	2.6.38
Export STR	C2	*GTS Modeling> CAESAR II>Export STR*	`GTSExportSTR`	2.6.39
Convert Lines/Polylines to Members/Shells	DXF Convert Lines to Members	*GTS Modeling> Convert Lines/Polylines to Members/Shells*	`GTSDXFRead`	2.6.40

Model Wizard	Model Wizard	*GTS Modeling>Model Wizard*	`GTSModelWizard`	2.6.41
List (Group)	List	*GTS Modeling>Groups>Manage*	`GTSGroups`	2.6.42
+Joints (Group)	+Joints	*GTS Modeling>Groups>Add Joints*	`GTSGroupJoints`	2.6.42
+Members (Group)	+Members	*GTS Modeling>Groups>Add Members*	`GTSGroupMembers`	2.6.42
+Shells (Group)	+Shells	*GTS Modeling>Groups>Add Shells*	`GTSGroupShells`	2.6.42
Self Weight	Self Weight	GTS Modeling>Loads>Self Weight	`GTSSelfWeight`	2.6.43
Load Cases	Load Cases	*GTS Modeling>Loads>Load Cases*	`GTSNewLoadCase`	2.6.44
Load Combinations	Combinations	*GTS Modeling>Loads>Load Combinations*	`GTSLoadCombination`	2.6.54
Standardized Combinations	Standardized Combinations	*GTS Modeling>Loads>Standardized Combinations*	`GTSLoadCombinationStandardized`	2.6.55
Joint Load	Joint Load	*GTS Modeling>Loads>Joint Load*	`GTSJointLoad`	2.6.45
Member Load	Member Load	*GTS Modeling>Loads>Member Load*	`GTSBeamLoad`	2.6.46
Shell Load	Shell Load	*GTS Modeling>Loads>Shell Load*	`GTSShellLoad`	2.6.47
Area Load	Area Load	*GTS Modeling>Loads>Area Load*	`GTSAreaLoad`	2.6.48
Wind Load ASCE 705	Wind Load ASCE 705	*GTS Modeling>Loads>Wind Load ASCE 705*	`GTSWindLoadsASCE705`	2.6.49
Wind Load ASCE 710	Wind Load ASCE 710	*GTS Modeling>Loads>Wind Load ASCE 710*	`GTSWindLoadsASCE710`	2.6.50
Seismic Load	Seismic Load	*GTS Modeling>Loads>Seismic Load*	`GTSSeismicLoading`	2.6.53
Steel Design Parameters		*GTS Modeling>Steel Design Parameters*	`GTSSteelDesignParameters`	2.6.56
Create Input File	GTI GTS	*GTS Modeling>Create Input File*	`GTSExportGTI`	2.6.57
Edit GTI	Edit GTI	*GTS Modeling>Edit GTI*	`GTSEditGTI`	2.6.58

Execute GT STRUDL	GTS	*GTS Modeling>Execute*	`GTSExecuteGTI`	2.6.59
Read Results	Read Results	*GTS Modeling>Read Results*	`GTSResultsGTI`	2.6.60
Import GTI	Import GTI		`GTSGTIRead`	2.6.61
Set View	Set View	*GTS Display>Set View*	`GTSSetView`	2.6.62
3D View	3D	*GTS Display>3D Sections*	`GTSSet3D`	2.6.63
Analytical/Physical	Analytical/Physical	*GTS Display>Analytical/Physical*	`GTSDisplayPhysicalMembers`	2.6.64
Frame View	Frame	*GTS Display>Frame*	`GTSSet1D`	2.6.63
Options (View)	Options	*GTS Display>Options*	`GTSDisplay`	2.6.66
Colors	Colors	*GTS Display>Colors*	`GTSColorView`	2.6.65
Annotate	Annotate	*GTS Display>Annotate*	`GTSAnnotate`	2.6.67
Select	Select	-	`GTSSelect`	2.6.68
Display Member Local Axes	Member Local Axes	GTS Display>Member Local Axes	`GTSDisplayLocalAxes`	2.6.69
Display Member Releases	Releases	GTS Display> Member Releases	`GTSDisplayReleases`	2.6.70
Display Shell Planar Axes	Shell Planar Axes	GTS Display> Shell Planar Axes	`GTSDisplayPlanarAxes`	2.6.71
Display Joint Supports	Joint Supports	GTS Display> Joint Supports	`GTSDisplaySupports`	2.6.72
Display Joint Loads	Joint Loads	*GTS Display>Joint Loads*	`GTSDisplayJointLoads`	2.6.73
Display Member Loads	Member Loads	*GTS Display>Member Loads*	`GTSDisplayMemberLoads`	2.6.74
Display Shell Loads	Area Loads	*GTS Display>Shell Loads*	`GTSDisplayElementLoads`	2.6.75
Display Area Loads	Shell Loads	*GTS Display>Area Loads*	`GTSDisplayAreaLoads`	2.6.76
Deformed Structure	Deformed	*GTS Display>Deformed Structure*	`GTSDisplayJointDisplacements`	2.6.77
Undeformed Structure	Undeformed	*GTS Display> Udeformed Structure*	`GTSDisplayJointDisplacements`	2.6.77
Annotate Joint Displacements	Annotate Displacements		`GTSAnnotateJointDisplacements`	2.6.78

Displacements	Displacements	*GTS Display> Displacements*	`GTSDisplaySectionDisplacements`	2.6.79
Member Diagrams	Diagrams	*GTS Display>Member Diagrams*	`GTSDisplayMemberForces`	2.6.80
Finite Element Results	Elements	*GTS Display>Element Results*	`GTSDisplayElementResults`	2.6.81
Finite Element Results Selection	Selection	*GTS Display>Element Results Selection*	`GTSDisplayElementResultsSel`	2.6.82
Member Code Check Results	Code Check	*GTS Display>Member Code Check Results*	`GTSColorCodeCheck`	2.6.83
Displacements Datasheets	Displacements	*GTS Display>Results Datasheets> Displacements*	`GTSDataSheetJointDisp`	2.6.84
Member Forces Datasheets	Member Forces	*GTS Display>Results Datasheets> Displacements*	`GTSDataSheetMemberForces`	2.6.84
Section Forces Datasheets	Section Forces	*GTS Display>Results Datasheets> Displacements*	`GTSDataSheetSectionForces`	2.6.84
Reactions Datasheets	Reactions	*GTS Display>Results Datasheets> Displacements*	`GTSDataSheetReactions`	2.6.84
Stresses Datasheets	Stresses	*GTS Display>Results Datasheets> Displacements*	`GTSDataSheetStresses`	2.6.84
Code Check Datasheets	Code Check	*GTS Display>Results Datasheets> Displacements*	`GTSDataSheetCodeCheck`	2.6.84
Report Builder		-	`GTSReportBuilder`	2.6.85
Clear Results	Clear	*GTS Display>Clear Results Layer*	`GTSDisplayResultsClear`	2.6.86
Current Version	-	*GTS Display>Version*	`GTSVersion`	2.6.87