Notice

This GT STRUDL User Guide - CAD Modeler Getting Started Guide is applicable to GT STRUDL Version 2017 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

Copyright © 2017 Intergraph® Corporation. All Rights Reserved. Intergraph is part of Hexagon.

Including software, 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- Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States.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Unpublished - rights reserved under the copyright laws of the United States.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


Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on eCustomer, 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 End User License Agreement ("EULA") 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 EULA 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. 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 EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.
The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

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

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

Limitation of Damages

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

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

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

Export Controls

Intergraph Corporation’s software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.


c. To any entity when Licensee 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 Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation’s Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Intergraph, the Intergraph logo, and GT STRUDL are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Microsoft and Windows are registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries. Other brands and product names are trademarks of their respective owners.
## Table of Contents

### NOTICES

### Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table of Contents</td>
<td>v</td>
</tr>
<tr>
<td>1. Getting Started</td>
<td>1</td>
</tr>
<tr>
<td>1.1. Introduction</td>
<td>1</td>
</tr>
<tr>
<td>1.2. Installing CAD Modeler under Windows 10 and 7</td>
<td>1</td>
</tr>
<tr>
<td>2. Using CAD Modeler</td>
<td>6</td>
</tr>
<tr>
<td>2.1. Overview of Using CAD Modeler and configuring AutoCAD/BricsCAD</td>
<td>6</td>
</tr>
<tr>
<td>2.2. Running CAD Modeler</td>
<td>6</td>
</tr>
<tr>
<td>2.3. Menu Bar and Ribbon Area</td>
<td>7</td>
</tr>
<tr>
<td>2.4. AutoCAD/BricsCAD Commands</td>
<td>8</td>
</tr>
<tr>
<td>2.5. CAD Modeler Commands</td>
<td>9</td>
</tr>
<tr>
<td>2.5.1. Units</td>
<td>9</td>
</tr>
<tr>
<td>2.5.2. Materials</td>
<td>10</td>
</tr>
<tr>
<td>2.5.3. Levels</td>
<td>10</td>
</tr>
<tr>
<td>2.5.4. Grid</td>
<td>11</td>
</tr>
<tr>
<td>2.5.5. Creating Joints</td>
<td>13</td>
</tr>
<tr>
<td>2.5.6. Finding Joints</td>
<td>14</td>
</tr>
<tr>
<td>2.5.7. Joint Supports</td>
<td>14</td>
</tr>
<tr>
<td>2.5.8. Joint Properties</td>
<td>14</td>
</tr>
<tr>
<td>2.5.9. Duplicate Joints</td>
<td>15</td>
</tr>
<tr>
<td>2.5.10. Floating Joints</td>
<td>16</td>
</tr>
<tr>
<td>2.5.11. Sections</td>
<td>16</td>
</tr>
<tr>
<td>2.5.12. Creating Members</td>
<td>19</td>
</tr>
<tr>
<td>2.5.13. Finding Members</td>
<td>20</td>
</tr>
<tr>
<td>2.5.14. Splitting Members</td>
<td>21</td>
</tr>
<tr>
<td>2.5.15. Merging Members</td>
<td>21</td>
</tr>
<tr>
<td>2.5.16. Member Properties</td>
<td>21</td>
</tr>
<tr>
<td>2.5.17. Member Filters</td>
<td>24</td>
</tr>
<tr>
<td>2.5.18. Creating Shell Finite Elements</td>
<td>25</td>
</tr>
<tr>
<td>2.5.19. Reverse Incidence Order</td>
<td>26</td>
</tr>
<tr>
<td>2.5.20. Finding Shells</td>
<td>26</td>
</tr>
</tbody>
</table>
2.5.21. Shell Properties ................................................................. 26
2.5.22. Meshing along a curve ....................................................... 27
2.5.23. Meshing between two lines .............................................. 29
2.5.24. Meshing between four lines .............................................. 30
2.5.25. Meshing inside a polyline .................................................. 30
2.5.26. Meshing by extruding a polyline ....................................... 32
2.5.27. Meshing using 3 curves ..................................................... 32
2.5.28. Groups ........................................................................ 32
2.5.29. Self - Weight .................................................................. 33
2.5.30. Load Cases ..................................................................... 34
2.5.31. Joint Loads .................................................................... 35
2.5.32. Member Loads ................................................................ 36
2.5.33. Shell Loads ..................................................................... 39
2.5.34. Area Load ....................................................................... 39
2.5.35. Load Combinations ........................................................... 41
2.5.36. Create GTI ...................................................................... 42
2.5.37. Edit GTI .......................................................................... 43
2.5.38. Execute GT STRUDL ........................................................ 43
2.5.39. Read Analysis Results ....................................................... 43
2.5.40. Import GTI ....................................................................... 44
2.5.41. Set Views .......................................................................... 45
2.5.42. 3D or Wireframe View of the Structure .............................. 45
2.5.43. Colors and Visible Elements ............................................ 46
2.5.44. Display Options ................................................................. 47
2.5.45. Annotate ......................................................................... 48
2.5.46. Select CAD Modeler’s entities .......................................... 49
2.5.47. Display Member Local Axes ............................................. 50
2.5.48. Display Member Releases .................................................. 50
2.5.49. Display Shell Planar Axes .................................................... 50
2.5.50. Display Joint Supports ...................................................... 50
2.5.51. Display Joint Loads ............................................................. 51
2.5.52. Display Member Loads ....................................................... 51
2.5.53. Display Shell Loads ............................................................ 52
4. Tutorial Example #2 ........................................................................................................ 117
   4.1. Introduction ........................................................................................................... 117
   4.2. Open CAD Modeler and start working ................................................................. 118
   4.3. Define the basic geometry of the model ............................................................... 118
   4.4. Create the bottom of the tank ............................................................................. 121
   4.5. Create the walls of the tank ................................................................................ 123
   4.6. Create Supports ................................................................................................. 130
   4.7. Check the model ............................................................................................... 132

3. Tutorial Example #1 .................................................................................................... 59
   3.1. Introduction .......................................................................................................... 59
   3.2. Open CAD Modeler and start working ............................................................... 59
   3.3. Define the basic geometry of the model ............................................................. 60
   3.4. Create the 1st floor ............................................................................................ 65
   3.5. Create the 2nd floor .......................................................................................... 74
   3.6. Create the 3rd floor .......................................................................................... 76
   3.7. Create bracing .................................................................................................... 78
   3.8. Create girders ..................................................................................................... 83
   3.9. Create an opening ............................................................................................. 89
   3.10. Create Supports ............................................................................................... 90
   3.11. Check the model ............................................................................................ 91
   3.12. Define Groups .................................................................................................. 92
   3.13. Define Loads ..................................................................................................... 95
   3.14. GT STRUDL Input File .................................................................................. 107
   3.15. Display Results .............................................................................................. 109

2.5.54. Display Area Loads ..................................................................................... 53
2.5.55. Display Deformed Structure ......................................................................... 53
2.5.56. Display Section Displacements ..................................................................... 54
2.5.57. Display Member Diagrams ............................................................................ 54
2.5.58. Display Finite Element Results ...................................................................... 55
2.5.59. Display Finite Element Selection Results ................................................... 56
2.5.60. Display Member Code Check Results .......................................................... 57
2.5.61. Clear Results Layer ...................................................................................... 58
2.5.62. Version ........................................................................................................... 58

3.15 GT STRUDL Input File.................................................................................. 107
4.8. Define Groups........................................................................................................ 133
4.9. Define Loads ........................................................................................................ 136
4.10. Create GT STRUDL Input File ............................................................................ 141
4.11. Display Results .................................................................................................. 145
5. Appendix – List of Commands ................................................................................ 148
1. Getting Started

CAD Modeler is an add-on to AutoCAD® or BricsCAD®, 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 10 and 7

In order to install CAD Modeler check the box “Install CAD Modeler” on the form shown on the next page during the GT STRUDL main installation procedure.

An AutoCAD version (2012-2017) or a BricsCAD version (Platinum or Pro, 16.2.x or 17.1.x) has to be installed in the computer prior to CAD Modeler installation.
CAD Modeler Installation can also be launched independently, after GT STRUDL installation, by executing the file “CADM_setup.exe” which is located in the CADModeler folder in the GTStrudl 2017 installation directory. The following steps are common regardless if the installation was launched from the GT STRUDL main installation or independently.

The first screen is a welcome dialog that prompts you to close all other applications before continuing the installation. It is essential to close any running instances of GT STRUDL, CAD Modeler, AutoCAD or BricsCAD before continuing the installation.
The next step is to select the components to be installed. The CAD Modeler Main Files are installed by default, and in addition you have to choose at least one version of the AutoCAD or BricsCAD CAD Modeler Interface to be installed depending on the version of AutoCAD or BricsCAD that is currently installed in your computer.
The last screen summarizes your selection and by pressing “Install” the installation procedure starts.
CAD Modeler is installed in the same installation directory with GT STRUDL, under the subdirectory “CADModeler”. For example, “C:\Program Files (x86)\GTStrud\2017\CADModeler” is a typical CADModeler installation directory.

When the copy process is completed, a new dialog named “Finalizing Setup” appears and you are prompted to enter the corresponding AutoCAD and/or BricsCAD Installation Directories.

Press the browse “…” button to find and select the file “acad.exe” or “bricscad.exe”, in the AutoCAD or BricsCAD installation directory, and press “Open”. The name of the directory is copied to the corresponding edit box, and by pressing “End”, the installation is complete.
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 dialog by selecting the “CAD Modeler” icon. 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:

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. CAD Modeler Commands

2.5.1. 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.

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

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

2.5.3. Levels

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

Using the Level Properties form you can:
- Set the Height for each level, in current length units
- Define the visible status of each level: if Visible or not
- Add Levels to the model
- Delete Levels from the model
- Detect Levels Automatically using an algorithm to detect levels along the height of the structure by identifying locations having at least four horizontal members.
- Merge Levels, by selecting two or more levels and merging them to one.
- Define if the Vertical Axis is the global Z or global Y.
- Update Levels for All Entities, in order to assign the correct Level to each entity (joint, member or finite element) depending on its coordinates along the height of the structure.
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.

2.5.4. 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 Starting From item, which can be a number or letter depending on the Type.
- Select the levels that this grid will be applied to. You can apply the grid to more than one levels and/or have multiple grids per level.

By pressing OK, you are prompted to enter the Insert Point of the grid, meaning the coordinates of the lower left corner of the grid. The grid lines are then created as shown in the figure on the next page.
You can also change the properties of an existing grid by expanding the “Levels” tab and selecting the ribbon icon, or from the menu “GTS Modeling>Grid>Change” or by typing GTSGridChange at the command prompt, and then selecting the Grid to be edited.

2.5.5. Creating Joints

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

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

2.5.6. Finding Joints
You can find an individual joint from the ribbon command 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).

2.5.7. Joint Supports
You can find an individual joint from the ribbon command 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)

2.5.8. Joint Properties
You can change the properties of a joint from the ribbon command 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.

2.5.9. Duplicate Joints

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 menu “GTS Modeling>Checks>Check for Duplicate Joints” or by typing GTSCheckDuplicateJoints at the command prompt. You then have to enter the desired merge accuracy (Enter Merge Accuracy <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:
2.5.10. Floating Joints

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. Using the command “GTS Modeling>Checks>Check for Floating Joints” from the Menu or by typing GTSCheckFloatingJoints at the command prompt, floating joints are automatically identified, and using the corresponding form as shown below, they can be deleted.

2.5.11. 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 GTSTRUDL.

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 sectional properties in the current unit system.
You can access the existing cross section library (GT STRUDL tables) from the ribbon command 

Sections, or from the Menu “GTS Modeling>Cross Sections>Table” or by typing GTSPrams 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 have to manually open the corresponding .ds file in GTSTRUDL prior to the "Execute GTSTRUDL" Command.
2.5.12. Creating Members

You can generate individual members from the ribbon command Generate or from the menu “GTS Modeling>Members>Generate Beam Members” or by typing 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 level 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.
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.

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.

2.5.13. 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
2.5.14. 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.

2.5.15. 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.

2.5.16. 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)
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.

2.5.17. 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 $F_x - F_y - F_z - M_x - M_y - M_z$ for both ends and section forces $F_x - F_y - F_z - M_x - M_y - M_z$.

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

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

Filtered members may be:
- Added to any Group
- Selected as AutoCAD’s/BricsCAD’s selection (to be edited, moved, copied, moved etc), using the option “Keep Selected after closing form”
- Made the only visible entities of the structure, by hiding all other entities, using the option “Make them the only visible”
2.5.18. 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 \(\text{Quad}\) 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 \(\text{Triangle}\) 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).
2.5.19. Reverse Incidence Order
The Incidence Order (clockwise or counterclockwise) of selected shell elements can be reversed using the icon Reverse or from the menu “GTS Modeling>Shell>Reverse Incidence Order” or by typing GTSShellReverse at the command prompt. The Incidence Order defines the orientation of the Element’s Planar Z and Local Z Axes which then also affects the Local and Planar X and Y Axes.

2.5.20. 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).

2.5.21. 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.

2.5.22. 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 shown on the next page.
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.

2.5.23. 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

2.5.24. Meshing between four lines
You can create Members OR Finite Elements between four selected AutoCAD/BricsCAD linear entities, that can be Lines or Arcs, from the ribbon command 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:

![Diagram of meshing between four lines](image)

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

2.5.25. 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 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 or if you try to control the maximum area of the finite elements.
- The maximum area of each finite element.
- The quality of the triangles that are going to be generated.
- Add one or multiple (MultiA) 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 “Boundary Maximum 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.
- 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.

2.5.26. 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 GTEXtrudePoly 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.

2.5.27. Meshing using 3 curves
You can create Members OR Finite Elements between three selected AutoCAD/BricsCAD linear entities, that can be Lines or Arcs, from the ribbon command 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.

2.5.28. 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 enter joints, members and shell elements to it using the commands:
- +Joints ribbon icon, or “GTS Modeling>Groups>Add Joints” or by typing GTSGroupJoints at the command prompt
- +Members ribbon icon, or “GTS Modeling>Groups>Add Members” or by typing GTSGroupMembers at the command prompt
- +Shells ribbon icon, or “GTS Modeling>Groups>Add Shells” or by typing GTSGroupShells at the command prompt

2.5.29. Self - Weight

The Self-weight load of the structure can be created from the ribbon command 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

2.5.30. Load Cases

A new load case can be created from the ribbon command 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.
2.5.31. Joint Loads

A Joint Load can be entered from the ribbon command 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.
2.5.32. Member Loads

A Member Load can be entered from the ribbon command 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.
2.5.33. Shell Loads

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

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

![Element Properties Form]

2.5.34. Area Load

An Area Load can be entered from the ribbon command \[\text{Area}\] 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/length²). A positive value is applied in the negative global DIRECTION
- The Global direction of the loading plane (X, Y or Z) and the tolerance
- The position (Elevation) of the plane, defined by a coordinate or an existing Joint. All members belonging to the plane having this elevation are located and loaded by GT STRUDL.
- The Distribution of the load (one way or two way)

Advanced Optional Features:

- Outline Region: Select the members that form the closed perimeter of the loaded area. If an Outline Region is defined then the area load is applied ONLY to the selected area and NOT to the whole plane having the elevation defined above.
- Exclude Area: Internal openings or islands not being loaded may be specified by selecting the members that define the perimeter of the excluded area.
- Ignore Members: Select members that you do not want to be loaded (e.g., bracing members)

By selecting “Display >>” you are able to graphically view the loaded area, marked with a yellow hatch pattern.
2.5.35. Load Combinations

A new load combination can be created from the ribbon command 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.
2.5.36. 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 GTExportGTI 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.

2.5.37. Edit GTI

The GT STRUDL Text Input file can be edited from the ribbon command 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.

2.5.38. Execute GT STRUDL

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

2.5.39. Read Analysis Results

After performing the stiffness analysis in GT STRUDL, results can be read back to CAD Modeler, from the ribbon command 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.

2.5.40. Import GTI

An existing GTI file can be imported it into CAD Modeler from the menu “GTS Modeling>Import> GT.STRUDL GTI” or by typing `GTSGTIRead` at the command prompt. Note that the GTI should be generated by the command “File>Save>Text Input File...” from GT STRUDL main menu.
2.5.41. 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.

2.5.42. 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 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 ribbon command or from the menu “GTS Display>Whole Structure” or by typing GTSSetAllVisible at the command prompt.
2.5.43. 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.

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 group, only the 1st group is taken into account.
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.

### 2.5.44. Display Options

You can set the display options from the ribbon command 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 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.

2.5.45. 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.

2.5.46. 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 `GTSSSelect` 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.
2.5.47. 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 2.5.44).

2.5.48. 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 2.5.44).

2.5.49. 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 2.5.44).

2.5.50. 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 2.5.44).
2.5.51. Display Joint Loads

You can view the joint loads applied in the structure from the icon (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.

2.5.52. Display Member Loads

You can view the member loads applied in the structure from the icon (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.

2.5.53. 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.

2.5.54. Display Area Loads

You can view the area loads applied in the structure from the icon (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 (2.5.34) to bring up the area load dialog, select the specific area load and click “Display >>”.

2.5.55. Display Deformed Structure

You can view the deformed shape of the structure from the icon (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 (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.

2.5.56. Display Section Displacements

You can view the displacements of each cross section (including 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 “Section Displacement” form appears where you can select:

- The desired Load Case or Combination
- The Scale factor
- The Font Size (in pts) for Annotations
- 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. If there are any hidden members their deformed shapes are not displayed.
- The “Annotate >” button allows you to annotate any value on the deformed shape by first clicking on the deformed shape curve and then at the position that annotation will be placed.

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

2.5.57. 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
- 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 screen, having information about the load case and member diagram.

2.5.58. 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.
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. $S_{xx}$, $S_{yy}$, $S_{zz}$
- The Location: Top, Middle or Bottom surface of the element which is defined by the local or planar Z axes of the shell element. The top surface is in the positive Z direction.
- The “Annotate >>” button allows you to annotate any value of the diagram by first clicking on a joint and then at the position that annotation will be placed.
- The “Display >>” button creates the contour and a popup legend with the limits of each color appears.

2.5.59. Display Finite Element Selection Results

You can view the finite element results of selected elements from the icon (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.

2.5.60. Display Member Code Check Results

You can view the pass/fail result of a Steel Code check or design from the icon (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

2.5.61. 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.
2.5.62. 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 \[\text{Sections}\]. Click on the *European* list and then on the *HEB* table of profiles. Select the profile *HE320B* that will be used for the columns, by double clicking on it.
The profile is added to the project and it appears in the left list-box having a black dot in front of it.

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

Note: You can add additional profiles at any time by following this procedure and also view the full list of profiles used in your model and add more profiles if needed.
**Step #5.** Define the 3 levels of the model by pressing the icon `Levels`. Press the Add Level button 3 times to add 3 levels to your model. Modify the height of the 1st level by selecting the Height cell of the 1st Level and entering 4. Make sure that Z Vertical Axis option is checked and press OK to close the form.

![Level Properties](image)

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

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

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

![Grid Placement](image)
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 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.
3.4. Create the 1st floor

**Step #7.** Start entering the columns by clicking on the icon \[Image\]. 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.

**Step #8.** You can easily change to an isometric view of the structure by pressing the small house icon in AutoCAD’s or BricsCAD’s View Cube. 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).
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.

Press “Place Member(s) >>” button.

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

Then you have to define the Ending Point (x,y,z) so click on joint 8, as shown in the picture below.
Members 9, 10 and 11 will be created.

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

Since the command is still active, you are prompted to enter the Starting Point (x,y,z), repeat the same procedure by clicking on joint 10 (top of column at position A-2), and then click at joint 16 (top of column at position D-2). Members 12, 13 and 14 will be created.
Step #10. Enter the beams, along Y axis. The command Generate Beams should be still active, else you can call it again by clicking again on the icon Generate. Keep the same settings at the Place Member Form, as in the previous step, regarding the cross section and Beta angle, but do NOT click on Split Intersecting Members. Press the “Place Member(s) >>” button.

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

Repeat the same procedure by clicking on the joints 4 and 12 to generate member 16, joints 6 and 14 to generate member 17 and joints 8 and 16 to generate member 18. Then, press ESC to terminate the command.
Step #11. Create an arc on the right side of the structure:

Type `ARC` and

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

Step #12. Generate Members along the Arc: Click on the icon `1D Curve` and when the prompt message *Select Curve (Line or Arc)* appears, click on the Arc that you have created in the previous step.
The Select Mesh Properties 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 \(\text{Options}\) in the ribbon bar and then uncheck the Visible Labels option for Joints, Members and 2D Elements.

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

Note: You can also delete or hide the Arc line as it is no longer needed.

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

Then, type \text{MIRROR} and when you get the notification \textit{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: 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 1\text{st} and the 2\text{nd} click of the mouse at the points 1 and 2 as shown in the picture below and press \textless \text{ENTER}\textgreater. You will get a confirmation that 44 objects were found (or 45 if you still have the arc).

When you get the message \textit{Specify second point of mirror line}, click on the joints at points 3 and 4 as shown in the picture above.

Then, press \textless \text{ENTER}\textgreater and reply to the question \textit{Erase source objects? [Yes/No]} \textless N\textgreater, 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 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.

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 that 101 objects were found and then press <ENTER>.

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

In order to **Specify second point or [Array] <use first point as displacement>:** click at the top of the same column such as point 2 of the picture and then press <ENTER> 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 or BricsCAD’s view cube 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, 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 that 77 objects were found and press `<ENTER>`.

In order to Specify 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 Specify second point or <use first point as displacement>: 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 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:** click on the same two points that were used in the previous MOVE command as shown in the following figure.

![Diagram](image)

Since this is a crossing window, the columns are automatically selected. You will get a notification that 77 objects were found and press `<ENTER>`.

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

In order to **Specify second point or <use first point as displacement>:** type `@0,0,3` and press `<ENTER>`.

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 , 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 \( \uparrow \) Higher Level and \( \downarrow \) Lower Level icons. The current level appears in the top caption of AutoCAD’s screen.

You can make whole structure visible by clicking on the icon .

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 \( \text{ } \text{ } \) to display the 3D solid view of the model, replacing the wireframe view as shown in the following figure:
Save your model, using a different file name (Save As...). By saving your model with a different name each time, it is easier to back up to a previous state of the model.

3.7. Create bracing

**Step #23.** Place bracing members at the front:

Press the icon ![Frame](icon) to switch back to wireframe view to be able to process CAD Modeler’s and AutoCAD’s/BricsCAD’s commands faster.

<table>
<thead>
<tr>
<th>Levels</th>
<th>Height</th>
<th>Elevation</th>
<th>Visible</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.000000</td>
<td>4.000000</td>
<td>✓</td>
</tr>
<tr>
<td>2</td>
<td>3.000000</td>
<td>7.000000</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3.000000</td>
<td>10.000000</td>
<td></td>
</tr>
</tbody>
</table>

Click at icon ![Levels](icon), check the visible property for level 1 and uncheck it for all other levels and press OK.
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 again on joint located at Point 2 and then on the joint located at Point 3 and the second bracing member is created.

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

Click again on the joint located at Point 5, click on the joint located at Point 6 and the fourth bracing member is created.
Press ESC to terminate the Generate Beam command.

**Step #24.** Change the properties of the Bracing Members: Click on the icon in the Members 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: click on the 4 bracing members created in a previous Step and press **<ENTER>**.

In order to define **Specify base point or [Displacement/mOde] <Displacement>:** click on the Joint at Point 1 of the following image.

In order to define **Specify second point or [Array] <use first point as displacement>:** click on the Joint at Point 2 in 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 as shown in the following image:
Save your model, using a different file name (Save As...).

Make the entire structure visible by clicking on the icon \[ \text{All} \].

Click on the icon \[ \text{Frame} \] to switch back to the wireframe view.

3.8. Create girders

**Step #27.** Split the beam members at the top level: Clicking on the \[ \text{Higher Level} \] 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](image)

Click on the icon \[ \text{Split} \] and click on the members A and B as shown at the following image, and then press <ENTER>.

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

![Split](image)

Click again on the icon \[ \text{Split} \] and click on the members C and D as shown at the following image, and then press <ENTER>. 
In order to define the Distance for splitting the member or the number of parts (negative number), enter \(-4\), so that the beams C and D will be split into 4 equal parts.

**Step #28.** Place girder members at the top level: Click on the icon 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:

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

Click on the icon 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.

Enter 0.25 for the Z Starting Eccentricity and 0.25 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 or BricsCAD’s view Cube 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 on the icon 📎 All and press Z and E (Zoom Extents).

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

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.

For the Merge Accuracy <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\textsuperscript{nd} time, you should get the notification that \textit{0 duplicate joints found}.

\textbf{Step #33.} Check for floating joints: In order to check for joints not connected to the model, click on the icon \textbullet\ Locate Floatings. If your model was created as described so far, you should get a notification \textit{0 floating joints found}.

\section*{3.12. Define Groups}

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

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

![](Groups.png)

Press the \textit{Add Group} button and enter \textit{Columns} as \textit{Name} of the group.

Press the \textit{Add Group} button and enter \textit{Beams} as \textit{Name} of the group.

Press the \textit{Add Group} button and enter \textit{Girders} as \textit{Name} of the group.

Press the \textit{Add Group} button and enter \textit{Bracing} as \textit{Name} of the group.

Press OK to close the form.

\textbf{Step #35.} Add Columns to their Group.

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

Press OK.

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

Press OK.

Click on the icon 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 and make only profiles 60x60x5 to be visible by unchecking 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 ![Self Weight](image) 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 Exit to close the dialog.

**Step #40.** Define Load Cases: Click on the icon ![Load Cases](image) and the Load dialog appears. Enter:

- LL as Name
- *Live Load* as the Load Description

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>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 [Multiple Selection] form appears having the tab “Joint 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 \( \text{Frame} \) to switch to wireframe view.

On the Menu Bar, click on GTS Display>Joint 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  untill “Level 1” is displayed.

Click on the icon  under the panel.
Type:

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

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

Press Clear to remove the solid hatch pattern and then **OK** to store the area load AL1.
**Step #46.** Define Area Load for Level 3: An area load equal to 1.0kN/m² along the vertical direction will be applied only to the two middle openings. Switch to Level 3 by clicking on the icon until “Level 3” is displayed. Click on the icon under the panel.

Click on the icon under the panel.

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 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 “Step48.dwg” and click on the icon ![GTI Icon](image) and the Create GT STRUDL Input file dialog appears. Keep the default GTI filename, check all options except “Read Finite Element Results” as shown in the following image and press OK.

![Create GT STRUDL Input File Dialog](image)

**Step #49.** View/Edit GTI: Click on the icon ![Edit GTI](image) 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 ![Execute GTI](image) and the GTI file created in the previous step will be sent to GT STRUDL main program that is waiting in the background.
Stiffness analysis is automatically performed and DBX result files are automatically created.

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

PARAMETERS

CODE EC3 ALL MEMBERS

CHECK ALL MEMBERS AS BEAM

WRITE REPLACE CODE 'Step48.25' 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 and the Read GT.STRUDL Results dialog appears. Check all options except “Read Finite Element Results” as shown below and press OK.
3.15. Display Results

**Step #52.** Display Deformed Model: on the Menu Bar, click on \( \text{Deformed} \) (ribbon tab “GTS Display”) and then select SW as load Case and press ENTER twice. The deformed structure will be drawn as shown below.
Repeat and select PL as the Load Case and the deformed structure appears as shown in the following image:
Click on \textit{Undeformed} (ribbon tab “GTS Display”) to return to the original undeformed position of the model.

\textbf{Step \#53.} Display Section Displacements: Click on the icon \textit{Frame} to switch back to the wireframe view. Click on \textit{Displacements} (ribbon tab “GTS Display”).
Select:
- SW as Load Case
- 0.1 as Scale Factor
- 10.00 as Font Size (default)
- 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 exit the command, uncheck “Hide Model”, press “Clear” Button and “Close”.

112
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, uncheck “Clear” Button and “Close”.

**Step #55.** Display Code Check Results: Click on ☑ Code Check (ribbon tab “GTS Display”) and select all members 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”.
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.

![Welcome to GT STRUDL](image)

4.3. Define the basic geometry of the model

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

---

118
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 or BricsCAD’s View Cube.

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 / BricsCAD’s View Cube 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 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 Edge Size**: 0.50
- **Mesh Quality**: High

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

Press the Create button to create the finite elements and joints on the bottom of the tank.
**Step #6.** Turn OFF labeling and view mesh:

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

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

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

---

4.5. **Create the walls of the tank**

**Step #7.** Generate the finite elements that will model the Wall of the Tank by extruding the polyline: Switch to Isometric view and click on the icon ![3D Extrude](image)

(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
- **Boundary Maximum 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 `Locate Duplicates`.

For the *Merge Accuracy* `<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 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 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.
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:
Press the icon \( \text{Frame} \) 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 or BricsCAD’s view cube.

Click on the icon \( \text{Support} \) 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.

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 `Locate Duplicates`.

For the Merge Accuracy <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 `Locate Floating`. If your model was created as described so far, you should get a notification that *0 floating joints found*.

**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. 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 2.5.44).

4.8. Define Groups

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

In the Group panel, click on the icon and the Group dialog appears.
Press the Add Group button and enter **Bottom** as Name of the group.

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

Press OK to close the form.

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

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

Click on the icon ![Groups](image) 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 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 an 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 Create New to create the new loading and then Exit to close the dialog.
Step #19. Define Load Cases: Click on the icon \( \text{Load Cases} \) and the Load dialog appears.

Enter:

- \( LL \) as Name
- \( \text{Live Load} \) as the Load Description

and press Create New.

Enter:

- \( PL \) as Name
- \( \text{Pressure Load} \) 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](image). Select the 2nd Tab in order to colorize elements by their group and make only the Group Bottom visible and press OK.

![Shell Properties (Multiple Selection)](image)

Click on the icon ![Shell](image). 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 **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.

**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]. 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.

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 ![GTI icon](image) 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  and the GTI file created in the previous step will be opened by the system’s default text editor.

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

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

In addition, you can enter GTMENU to view the solid model and the results as described in the GTMENU User Guide. You can also click on Results > Finite Element Results > Contour Stresses, 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 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: In the menu bar, click on GTS Display>Deformed Structure and then select PL as the load Case and press ENTER twice. The deformed structure will be drawn as shown below.

On the Menu Bar, click on GTS Display>Undeformed Structure to return to the original undeformed position of the model.

**Step #28.** Show Finite Element Results: On the Menu Bar, click on GTS Display>Element Results 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:
On the Menu Bar, click on GTS Display>Clear Results Layer and the contours and legend are cleared.
### 5. Appendix – List of Commands

<table>
<thead>
<tr>
<th>Command</th>
<th>Icon</th>
<th>Menu</th>
<th>Command Prompt</th>
<th>Link</th>
</tr>
</thead>
<tbody>
<tr>
<td>Units</td>
<td><img src="image" alt="Units Icon" /></td>
<td>GTS Modeling&gt;Units</td>
<td>GTSUnits</td>
<td>2.5.1</td>
</tr>
<tr>
<td>Materials</td>
<td><img src="image" alt="Materials Icon" /></td>
<td>GTS Modeling&gt;Materials</td>
<td>GTSMaterials</td>
<td>2.5.2</td>
</tr>
<tr>
<td>Sections</td>
<td><img src="image" alt="Sections Icon" /></td>
<td>GTS Modeling&gt;Cross Sections&gt;Table</td>
<td>GTSPrams</td>
<td>1.1.1</td>
</tr>
<tr>
<td>Prismatic</td>
<td><img src="image" alt="Prismatic Sections Icon" /></td>
<td>GTS Modeling&gt;Cross Sections&gt;Prismatic</td>
<td>GTSPrismatic</td>
<td>1.1.1</td>
</tr>
<tr>
<td>Levels</td>
<td><img src="image" alt="Levels Icon" /></td>
<td>GTS Modeling&gt;Levels</td>
<td>GTSElevels</td>
<td>2.5.3</td>
</tr>
<tr>
<td>Higher Level</td>
<td><img src="image" alt="Higher Level Icon" /></td>
<td>-</td>
<td>GTSLevelUp</td>
<td>2.5.3</td>
</tr>
<tr>
<td>Lower Level</td>
<td><img src="image" alt="Lower Level Icon" /></td>
<td>-</td>
<td>GTSLevelDown</td>
<td>2.5.3</td>
</tr>
<tr>
<td>Grid</td>
<td><img src="image" alt="Grid Icon" /></td>
<td>GTS Modeling&gt;Grid&gt;Create</td>
<td>GTSGrid</td>
<td>2.5.4</td>
</tr>
<tr>
<td>Change Grid</td>
<td><img src="image" alt="Change Grid Icon" /></td>
<td>GTS Modeling&gt;Grid&gt;Change</td>
<td>GTSGridChange</td>
<td>2.5.4</td>
</tr>
<tr>
<td>Generate Joint</td>
<td><img src="image" alt="Generate Joint Icon" /></td>
<td>GTS Modeling&gt;Joint&gt;Generate Joint</td>
<td>GTSJoint</td>
<td>2.5.5</td>
</tr>
<tr>
<td>At Level (Joint)</td>
<td><img src="image" alt="At Level Icon" /></td>
<td>GTS Modeling&gt;Joint&gt;Generate Joint at Level</td>
<td>GTSJointLevel</td>
<td>2.5.5</td>
</tr>
<tr>
<td>Find (Joint)</td>
<td><img src="image" alt="Find Joint Icon" /></td>
<td>GTS Modeling&gt;Joint&gt;Find</td>
<td>GTSFJID</td>
<td>2.5.6</td>
</tr>
<tr>
<td>Support</td>
<td><img src="image" alt="Support Icon" /></td>
<td>GTS Modeling&gt;Joint&gt;Support</td>
<td>GTSJointSupport</td>
<td>2.5.7</td>
</tr>
<tr>
<td>Change (Joint)</td>
<td><img src="image" alt="Change Joint Icon" /></td>
<td>GTS Modeling&gt;Joint&gt;Change</td>
<td>GTSJointChange</td>
<td>2.5.8</td>
</tr>
<tr>
<td>Locate Duplicates</td>
<td><img src="image" alt="Locate Duplicates Icon" /></td>
<td>GTS Modeling&gt;Checks&gt;Check for Duplicate Joints</td>
<td>GTSCheckDuplicateJoints</td>
<td>2.5.9</td>
</tr>
<tr>
<td>Locate Floating</td>
<td><img src="image" alt="Locate Floatings Icon" /></td>
<td>GTS Modeling&gt;Checks&gt;Check for Floating Joints</td>
<td>GTSCheckFloatingJoints</td>
<td>2.5.10</td>
</tr>
</tbody>
</table>
149
<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
<th>Code</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D Area (Meshing)</td>
<td>GTS Modeling&gt;Mesh Generation&gt;2D Between 4 Lines or Curves</td>
<td>GTSMesh2DPoly</td>
<td>2.5.25</td>
</tr>
<tr>
<td>3D Extrude (Meshing)</td>
<td>GTS Modeling&gt;Mesh Generation&gt;3D Extrude PolyLine</td>
<td>GTSExtrudePoly</td>
<td>2.5.26</td>
</tr>
<tr>
<td>2D 3Curves (Meshing)</td>
<td>GTS Modeling&gt;Mesh Generation&gt;3D Between 3 Lines or Curves</td>
<td>GTSmesh3D3L</td>
<td>2.5.27</td>
</tr>
<tr>
<td>List (Group)</td>
<td>GTS Modeling&gt;Groups&gt;Manage</td>
<td>GTSGroups</td>
<td>2.5.28</td>
</tr>
<tr>
<td>+Joints (Group)</td>
<td>GTS Modeling&gt;Groups&gt;Add Joints</td>
<td>GTSGroupJoints</td>
<td>2.5.28</td>
</tr>
<tr>
<td>+Members (Group)</td>
<td>GTS Modeling&gt;Groups&gt;Add Members</td>
<td>GTSGroupMembers</td>
<td>2.5.28</td>
</tr>
<tr>
<td>+Shells (Group)</td>
<td>GTS Modeling&gt;Groups&gt;Add Shells</td>
<td>GTSGroupShells</td>
<td>2.5.28</td>
</tr>
<tr>
<td>Self Weight</td>
<td>GTS Modeling&gt;Loads&gt;Self Weight</td>
<td>GTSSelfWeight</td>
<td>2.5.29</td>
</tr>
<tr>
<td>Load Cases</td>
<td>GTS Modeling&gt;Loads&gt;Load Cases</td>
<td>GTSNNewLoadCase</td>
<td>2.5.30</td>
</tr>
<tr>
<td>Load Combinations</td>
<td>GTS Modeling&gt;Loads&gt;Load Combinations</td>
<td>GTSLoadCombination</td>
<td>2.5.35</td>
</tr>
<tr>
<td>Joint Load</td>
<td>GTS Modeling&gt;Loads&gt;Joint Load</td>
<td>GTSJJointLoad</td>
<td>2.5.31</td>
</tr>
<tr>
<td>Member Load</td>
<td>GTS Modeling&gt;Loads&gt;Member Load</td>
<td>GTSSBeamLoad</td>
<td>2.5.32</td>
</tr>
<tr>
<td>Shell Load</td>
<td>GTS Modeling&gt;Loads&gt;Shell Load</td>
<td>GTSSShellLoad</td>
<td>2.5.33</td>
</tr>
<tr>
<td>Area Load</td>
<td>GTS Modeling&gt;Loads&gt;Area Load</td>
<td>GTSAreaLoad</td>
<td>2.5.34</td>
</tr>
<tr>
<td>Create GTI</td>
<td>GTS Modeling&gt;Create GT.STRUDL GTI</td>
<td>GTSExportGTI</td>
<td>2.5.36</td>
</tr>
<tr>
<td>Edit GTI</td>
<td>GTS Modeling&gt;Edit GT.STRUDL GTI</td>
<td>GTSEditGTI</td>
<td>2.5.37</td>
</tr>
<tr>
<td>Feature</td>
<td>Description</td>
<td>Code</td>
<td>Page</td>
</tr>
<tr>
<td>--------------------------------</td>
<td>--------------------------------------</td>
<td>-------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Member Diagrams</td>
<td>GTS Display&gt;Member Diagrams</td>
<td>GTSDisplayMemberDiagrams</td>
<td>2.5.57</td>
</tr>
<tr>
<td>Finite Element Results</td>
<td>GTS Display&gt;Element Results</td>
<td>GTSDisplayElementResults</td>
<td>2.5.58</td>
</tr>
<tr>
<td>Finite Element Results Selection</td>
<td>GTS Display&gt;Element Results Selection</td>
<td>GTSDisplayElementResultsSel</td>
<td>2.5.59</td>
</tr>
<tr>
<td>Member Code Check Results</td>
<td>GTS Display&gt;Member Code Check Results</td>
<td>GTSColorCodeCheck</td>
<td>2.5.60</td>
</tr>
<tr>
<td>Clear Results</td>
<td>GTS Display&gt;Clear Results Layer</td>
<td>GTSDisplayResultsClear</td>
<td>2.5.61</td>
</tr>
<tr>
<td>Current Version</td>
<td>GTS Display&gt;Version</td>
<td>GTSVersion</td>
<td>2.5.62</td>
</tr>
</tbody>
</table>