GT STRUDL[®] Version 2020 User Guide



CAD Modeler Getting Started Guide

Release Date: May 2020



Notice

This CAD Modeler Getting Started Guide is applicable to GT STRUDL Version 2020 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

Copyright © 2020 Hexagon AB and/or its subsidiaries and affiliates. All rights reserved.

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

U.S. Government Restricted Rights Legend

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- Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States.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

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation 305 Intergraph Way Madison, AL 35758

Documentation

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

Other Documentation

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

Terms of Use

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

Disclaimer of Warranties

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

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

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

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

Limitation of Damages

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

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

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

Export Controls

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

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

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

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

Trademarks

Intergraph[®], the Intergraph logo[®], Intergraph Smart[®], SmartPlant[®], SmartMarine[®], SmartSketch[®], SmartPlant Cloud[®], PDS[®], FrameWorks[®], I-Route, I-Export, Isogen[®], SPOOLGEN, SupportManager[®], SupportModeler[®], SAPPHIRE[®], TANK, PV Elite[®], CADWorx[®], CADWorx DraftPro[®], GTSTRUDL[®], and CAESAR II[®] are trademarks or registered trademarks of Intergraph Corporation or its affiliates, parents, subsidiaries. Hexagon and the Hexagon logo are registered trademarks of Hexagon AB or its subsidiaries. Microsoft and Windows are registered trademarks of Microsoft Corporation. ACIS is a registered trademark of SPATIAL TECHNOLOGY, INC. Infragistics, Presentation Layer Framework, ActiveTreeView Ctrl, ProtoViewCtl, ActiveThreed Ctrl, ActiveListBar Ctrl, ActiveSplitter, ActiveToolbars Ctrl, ActiveToolbars Plus Ctrl, and ProtoView are trademarks of Infragistics, Inc. Incorporates portions of 2D DCM, 3D DCM, and HLM by Siemens Product Lifecycle Management Software III (GB) Ltd. All rights reserved. Gigasoft is a registered trademark, and ProEssentials a trademark of Gigasoft, Inc. VideoSoft and VXFlexGrid are either registered trademarks of Oracle Corporation and/or its affiliates. Tribon is a trademark of AVEVA Group plc. Alma and act/cut are trademarks of the Alma company. Other brands and product names are trademarks of their respective owners.

Table of Contents

Ν	OTICES	i		iii
Та	able of	Cont	ents	v
1.	Gett	ting S	tarted	10
	1.1.	Intro	oduction	10
	1.2.	Insta	Illing CAD Modeler under Windows 10	10
2.	Usir	ng CAI	D Modeler	13
	2.1.	Over	view of Using CAD Modeler and configuring AutoCAD/BricsCAD	13
	2.2.	Runi	ning CAD Modeler	13
	2.3.	Men	u Bar and Ribbon Area	14
	2.4.	Auto	CAD/BricsCAD Commands	15
	2.5.	Auto	CAD/BricsCAD Drawing Units	16
	2.6.	CAD	Modeler Commands	17
	2.6.	1.	Units	17
	2.6.	2.	Materials	18
	2.6.	3.	Levels	18
	2.6.4	4.	Grid	19
	2.6.	5.	Creating Joints	21
	2.6.	6.	Finding Joints	22
	2.6.	7.	Joint Supports	22
	2.6.	8.	Joint Properties	22
	2.6.	9.	Sections	23
	2.6.	10.	Creating Members	26
	2.6.	11.	Finding Members	27
	2.6.	12.	Splitting Members	28
	2.6.	13.	Splitting to Crossing Members	28
	2.6.	14.	Merging Members	28
	2.6.	15.	Member Properties	28
	2.6.	16.	Member Filters	32
	2.6.	17.	Creating Shell Finite Elements	33
	2.6.	18.	Reverse Incidence Order	
	2.6.	19.	Finding Shells	34

2.6.20.	Shell Properties	. 34
2.6.21.	Joints Duplicates	. 35
2.6.22.	Joints Floatings	. 36
2.6.23.	Joints Interference	. 36
2.6.24.	Members Duplicates	. 37
2.6.25.	Members Zero Length	. 37
2.6.26.	Physical Members	. 37
2.6.27.	Shells Duplicates	. 38
2.6.28.	Names Duplicates	. 39
2.6.29.	Renumber Names	. 39
2.6.30.	Database Integrity	. 39
2.6.31.	Meshing along a curve	. 40
2.6.32.	Meshing between two lines	. 42
2.6.33.	Meshing between four lines	. 42
2.6.34.	Meshing inside a polyline	. 42
2.6.35.	Meshing by extruding a polyline	. 44
2.6.36.	Meshing using 3 curves	. 44
2.6.37.	Array 3D Advanced	. 45
2.6.38.	Soil Springs	. 45
2.6.39.	Export to CAESAR II	. 46
2.6.40.	Convert Lines/Polylines to Members/Shells	. 46
2.6.41.	Model Wizard	. 47
2.6.42.	Groups	. 48
2.6.43.	Self - Weight	. 49
2.6.44.	Load Cases	. 49
2.6.45.	Joint Loads	. 50
2.6.46.	Member Loads	. 51
2.6.47.	Shell Loads	. 53
2.6.48.	Area Load	. 53
2.6.49.	Wind Load ASCE 705	. 55
2.6.50.	Wind Load ASCE 710	. 57
2.6.51.	Seismic Load	. 59
2.6.52.	Load Combinations	. 60

2.6.53.	Standardized Combinations	. 61
2.6.54.	Steel Design Parameters	. 62
2.6.55.	Create GTI	. 63
2.6.56.	Edit GTI	. 64
2.6.57.	Execute GT STRUDL	. 64
2.6.58.	Read Analysis Results	. 65
2.6.59.	Import GTI	. 66
2.6.60.	Set Views	. 66
2.6.61.	3D or Wireframe View of the Structure	. 66
2.6.62.	Analytical/Physical Member View	. 67
2.6.63.	Colors and Visible Elements	. 67
2.6.64.	Display Options	. 69
2.6.65.	Annotate	. 70
2.6.66.	Select CAD Modeler's entities	. 71
2.6.67.	Display Member Local Axes	. 72
2.6.68.	Display Member Releases	. 72
2.6.69.	Display Shell Planar Axes	. 72
2.6.70.	Display Joint Supports	. 72
2.6.71.	Display Joint Loads	. 73
2.6.72.	Display Member Loads	. 73
2.6.73.	Display Shell Loads	. 74
2.6.74.	Display Area Loads	. 75
2.6.75.	Display Deformed Structure	. 75
2.6.76.	Annotate Joint Displacements	. 76
2.6.77.	Display Displacements	. 76
2.6.78.	Display Member Diagrams	. 77
2.6.79.	Display Finite Element Results	. 78
2.6.80.	Display Finite Element Selection Results	. 80
2.6.81.	Display Member Code Check Results	. 80
2.6.82.	Results Datasheets	. 81
2.6.83.	Report Builder	. 82
2.6.84.	Clear Results Layer	. 82
2.6.85.	Version	. 82

3. Tut	torial Example #1	
3.1.	Introduction	
3.2.	Open CAD Modeler and start working	
3.3.	Define the basic geometry of the model	
3.4.	Create the 1 st floor	89
3.5.	Create the 2 nd floor	
3.6.	Create the 3 rd floor	101
3.7.	Create bracing	
3.8.	Create girders	109
3.9.	Create an opening	114
3.10.	Create Supports	115
3.11.	Check the model	116
3.12.	Define Groups	117
3.13.	Define Loads	120
3.14.	GT STRUDL Input File	133
3.15.	Display Results	136
3.16.	Results Datasheets	
3.17.	Report Builder	
4. Tut	torial Example #2	
4.1.	Introduction	
4.2.	Open CAD Modeler and start working	
4.3.	Define the basic geometry of the model	
4.4.	Create the bottom of the tank	
4.5.	Create the walls of the tank	153
4.6.	Create Supports	
4.7.	Check the model	
4.8.	Define Groups	
4.9.	Define Loads	
4.10.	Create GT STRUDL Input File	171
4.11.	Display Results	
4.12.	Results Datasheets	
4.13.	Report Builder	
5. Tut	torial Example #3	

	5.1.	Introduction	. 184
	5.2.	Open CAD Modeler and start working	. 184
	5.3.	Define the basic geometry of the model	. 185
	5.4.	Create Columns	. 189
	5.5.	Create beams and girders	. 192
	5.6.	Create girders	. 198
	5.7.	Define supports	. 218
	5.8.	Define Loads	. 219
	5.9.	Perform analysis	. 220
	5.10.	Read analysis results	. 227
	5.11.	Display analysis results	. 227
6.	Арр	endix – List of Commands	. 237

GT STRUDL[®] CAD MODELER Getting Started Guide

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

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 (2012-2020) or a BricsCAD version (Platinum or Pro, 16.2.x, 17.1.x18.1.x, 19.x or 20.x) has to be installed in the computer prior to CAD Modeler installation.

GT STRUDL 2020 Installation Options	
Installation folder	
CAD Modeler	Cancel Install

CAD Modeler Installation can also be launched independently, after GT STRUDL installation, by executing the file "CADM_setup.exe" which is in the CADModeler folder in the GTStrudl 2020 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 choose at least one version of the AutoCAD or BricsCAD CAD Modeler Interface to be installed depending on the versions of AutoCAD or BricsCAD that are currently installed in your computer.

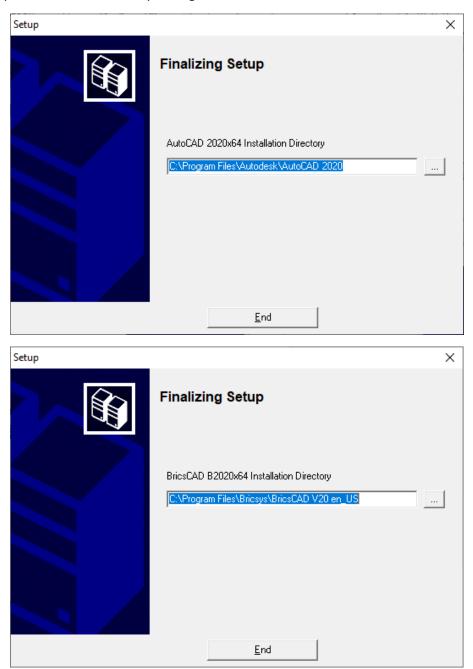
17.2 ME	~
17.2 MB	~
	×
	\sim
	·
010110	·
0.7 110	
	5.8 ME 6.0 ME 5.9 ME 5.6 ME 5.6 ME 5.7 ME

The next screen summarizes your selection and by pressing "Install" the installation procedure starts.

🕞 Setup - CAD Modeler 2020		_		×
Ready to Install Setup is now ready to begin installing (CAD Modeler on your o	computer.	Q	
Click Install to continue with the installa change any settings.	ation, or click Back if y	ou want to revie	ew or	
Setup type: Custom installation			^	
Selected components: CAD Modeler Main Files BricsCAD v20 64-bit Interface BricsCAD v18 64-bit Interface AutoCAD 2020 64-bit Interface				
<			>	
	< <u>B</u> ack	Install	Can	icel

CAD Modeler is installed in the same installation directory with GT STRUDL, under the subdirectory "CADModeler". For example, "C:\Program Files\GTSTRUDL\2020\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 screen by selecting the "CAD Modeler" icon in the Start a New Project group. A new instance of AutoCAD or BricsCAD, having CAD Modeler automatically loaded, is created. You will be able to identify that CAD Modeler is loaded successfully by having two additional menus ("GTS Modeling" and "GTS Display") next to the AutoCAD or BricsCAD main menus and the "GTS CAD Modeler" and "GTS Display" tabs in the ribbon area.

 Copen Cam more about GT STRUDL Command Mode	
Contraction of the second seco	CADWorx® Analysis Sc
* CADModeler is not installed. * Advanced license required	
Learn More Learn More Learn More Learn More Learn More Learn More Introduction to GT STRUDL View an informative webinar on the new live system presented on January 28, 2020 by here.	clicking
Image: Sector of the product that can be used a introduction to the most frequently used features. Image: Sector of the product that can be used a introduction to the most frequently used features. Image: Sector of the product that can be used a introduction to the most frequently used features.	is an
GT Menu CAD Modeler Command Mode Mizard Base Plate Wizard CAD-Based Modeling for Structural Engine February 20, 2020	<u>915</u>
D:\GTQA\Testing\2020-04-27_UTC-03-22-51\GTS 2020 Shell\M0069_gti	
Recent On-Demand Webinar: GT STRUDI, Webina Model Your Way, with GT STRUDI, Record Date: December 13, 2018	
On-Demand Webinar: GT STRUDL Webina Structural Workflow with Integraph Smart 3 Recorded Date: October 18, 2018	

2.3. Menu Bar and Ribbon Area

CAD Modeler commands can be accessed from AutoCAD's or BricsCAD's Ribbon Area at the top of the window by selecting the two tabs at the right: GTS CAD Modeler and GTS Display. CAD Modeler now support BricsCAD version 20 including all new functionality and the dark scheme.

Home Inse	st Annotate Parametric V	Sew Manage	Output Collab	borate GTS I	Modeling	GTS Display 🕢 +													
X Units	Levels Grid A Higher Level	d At Level	Sections Generate	Vertical Split	10.00	 Find Joint * Change Joint * Joints Duplicates * Find/Change/Check 	Cuad Cuad Triangle	1D Curve	2D 2Curves Meshino			List Groups	Elf Weight • Load Cases Combinations • Loads	Create GTI	GTS Execute GT STRUDL GT ST	📑 Import GTI	Analytical/Physic Options	al ③ Set View O Colors	C Annotate C Select Clear Inquire
Home	Insert Annotate	Paramet	tric View	Manag	e Out	tput Collabor	ate GT	S Moo	deling	GTS D	isplay	۲) +						
🚿 Units	i 📥 i	oint Suppo	rts			/ Deformed	Dis	place	ments	🔲 Di	isplaceme	ents	Reactions		111	A Frame		🗑 3D	
≻ Mem	nber Local Axes 📝 R	eleases	🛓 Join		Shell	ww Diagrams	C/ Co	de Ch	leck	M	lember Fo	orces	Stresses		Ĩ	🚞 Analvt	ical/Physical	Set View	
	Planar Axes		🖽 Me	mber 🧮	Area	Elements	🖉 Sel						Code Che		Report Builder	Option		Colors	Clear
	Fialial AKCS					Liements	Jei Jei	ection			cuonino	Ces	Code che	CK	builder	St obrio	15		
	Display Model		Dis	splay Load	ls	Displ	ay Results	5			Resul	ts Dat	tasheets		Report		Display		Clear

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.

GTS Modeling GTS Display Units Frame A Import > 3D Sections 1316 Materials Analytical/Physical Cross Sections > All Levels ON E Levels Set View Grid > Colors... Groups > Annotate... Joint > Member Local Axes Member > **Display Member Releases** Shell > Shell Planar Axes 17 Soil Springs Joint Supports Mesh Generation Joint Loads > Member Loads Loads > M Shell Loads Checks > Area Loads Steel Design Parameters **Deformed Structure** 17 GT STRUDL > Undeformed Structure Displacements CAESAR II > Member Diagrams W Convert Lines/PolyLines to Members/Shells Element Results Model Wizard Element Results Selection 2 Member Code Check Results ⊘, Clear Results Layer Results Datasheets > Options...

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

0

Current Version...

- Copy: Joint, Member and Element Loads and Supports are not copied
- Mirror: Joint, Member and Element Loads and Supports are not copied or mirrored. The Beta Angle of members is not mirrored. Element incidence order is mirrored so that element's orientation, that defines the Z Planar Axis, remains the same.

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

2.5. AutoCAD/BricsCAD Drawing Units

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

0% • 🏘 •	Architectural Architectural
N	Drawing Units
Length Type:	Angle Type:
Decimal	✓ Decimal Degrees ✓ * · · · · ·
Architectural	Precision:
Decimal	
Engineering Fractional Scientific	Sets the current format for units of measure. The values includ Architectural, Decimal, Engineering, Fractional, and Scientific. Engineering and Architectural formats produce feet-and-inche displays and assume that each drawing unit represents one inc
Fractional	Architectural, Decimal, Engineering, Fractional, and Scientific. Engineering and Architectural formats produce feet-and-inche
Fractional Scientific	Architectural, Decimal, Engineering, Fractional, and Scientific. Engineering and Architectural formats produce feet-and-inche displays and assume that each drawing unit represents one inc The other formats can represent any real-world unit.
Fractional Scientific	Architectural, Decimal, Engineering, Fractional, and Scientific. Engineering and Architectural formats produce feet-and-inche displays and assume that each drawing unit represents one inc The other formats can represent any real-world unit.
Fractional Scientific Insertion scale Units to scale in	Architectural, Decimal, Engineering, Fractional, and Scientific. Engineering and Architectural formats produce feet-and-inche displays and assume that each drawing unit represents one inc The other formats can represent any real-world unit.

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

2.6. CAD Modeler Commands

2.6.1. Units

The following form is used to define the active units, either from ribbon command ^X Units or from the menu "GTS Modeling>Units" or by typing GTSUnits at the command prompt.

Units									
Length	Force	Angles							
O Inches (in)	O Pounds (lbs)	Degrees							
O Feet (ft)	◯ Kips	○ Radians							
Meters (m)	◯ Tons	○ Cycles							
O Centimeters (cm)	◯ Kilograms								
O Millimeters (mm)	O Metric Tons	Time							
Temperature	○ Newtons	 Seconds 							
◯ Fahrenheit	 KiloNewtons 	○ Minutes							
 Centigrade 	OMegaNewtons	O Hours							
Scale non-structura (grids, structural line		OK Cancel							

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

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

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

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

Joint Co	ordinates and Of	fsets
	Coordinates	
x :	2487.85	
Υ:	1644.03	
Ζ:	0	

2.6.2. Materials

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

ID	Name	E	G	Density	Poisson	CTE
1	Steel	1.9994800E+008	7.5842000E+007	7.6901000E+001	3.000000E-001	1.1700000E-005
2	Concrete	2.4821100E+007	9.9284600E+006	2.3561600E+001	1.700000E-001	9.900000E-006
3	Aluminum	6.8947600E+007	2.5855400E+007	2.6601800E+001	3.300000E-001	2.3400000E-005

2.6.3. Levels

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



Levels or from the menu "GTS Modeling>Levels" or by typing GTSLevels at the command prompt.

Using the *Level Properties* form you can:

- Set the *Height* for each level, in current length units
- Define the visible status of each level: if Visible or not
- Add Levels to the model
- Delete Levels from the model
- Detect Levels Automatically using an algorithm to detect levels along the height of the structure by identifying locations having at least four horizontal members.
- *Merge Levels*, by selecting two or more levels and merging them to one.
- Define if the Vertical Axis is the global Z or global Y.

- Update Levels for All Entities, in order to assign the correct Level to each entity (joint, member or finite element) depending on its coordinates along the height of the structure.

Levels	Height	Elevation	Visible	Add Level
1	4	4.000000	✓	
2	3.000000	7.000000	✓✓	Delete Level
3	3.000000	10.000000		Detect Levels Automaticaly
				Merge Levels
				Base Elevation 0.0000
				Z Vertical Axis (else Y)
				Update Levels for All Entitie

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 Vower Level icons in the ribbon area. You can also type GTSLevelUp and GTSLevelDown at the command prompt.

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

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

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

2.6.4. Grid

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

₩

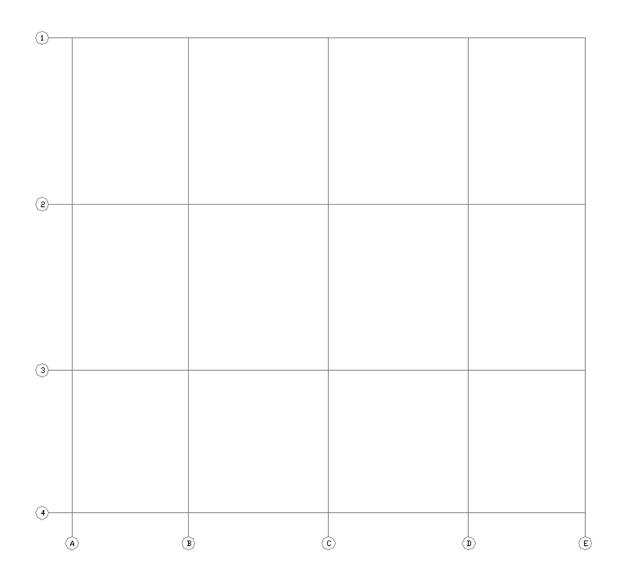
Grid , or from the menu "GTS Modeling>Grid>Create" or by typing GTSGrid at the command prompt.

		Grid		>
Placement Horizontal Spacing 5.0000 6.0000 7.0000	Sidelong Distance: 7.0000 Add Edit Delete	Angles Grid Angle X-Axis : Horizontal - Sidelong Angle : Labeling Image: Show Labels Text Height : Numbers Distance : Position Image: Left O Right Type Image: Numbers CLetter Start From : 1	0.00 90.00 0.20 1.00	Apply to Levels
				OK Cancel

Using the *Grid* form you can:

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

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



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

2.6.5. Creating Joints

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

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

2.6.6. Finding Joints

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

2.6.7. Joint Supports

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

Restraints	s & Spring va	alues	5		
Quick Sel	ection :	Pi	n		~
Restraint	Spring		Restraint	Spring	
√ Fx		0	Mx		0
√ Fy		0	Му		0
√ Fz		0	Mz		0

2.6.8. Joint Properties

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

Joint Properties		×
Model Joint Generalized Loads	Joint Coordinates and Offsets	
Name : 1 Level : 1 Rotation	Coordinates X: 1 Y: 2 Z: 0 Restraints & Spring values Quick Selection:	
Groups Inactive Remove Groups that joint belongs	Restraint Spring Fx 0 Fy 0 My 0 Fz 0	
	OK Cancel Apply Help	

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

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

2.6.9. Sections

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

Prismatic cross sections can be created from the Menu "GTS Modeling >Cross Sections>Prismatic" or by typing GTSPrismatic at the command prompt.

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

New Section Properties Section Name (Up to 15 Characters) :	OK
Section Name (Up to 15 Characters) :	Cancel
Ax :	Ix:
Ay :	Ιу:
Az :	Iz:
Sy:	Ey:
Sz :	Ez :
Yd :	Yc :
Zd :	Zc :
Shape Code :	

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 GTSParams at the command prompt.

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

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

AISC 14th Edition 2011	W-AISC14											
 W40x215 M/S/HP14 	NAME1	ND	AX	YD	WBTK	ZD	FLTK	KDES	KDET	К.		
RECHSS14	W44x335	1.118	0.06355	1.118	0.02616	0.4039	0.04496	0.06502	0.06668	0.03334		
C-AISC14	W44x290	1.118	0.0551	1.107	0.02197	0.4013	0.04013	0.05994	0.06191	0.03175		
T WTAISC14	W44x262	1.118	0.04981	1.1	0.01994	0.4013	0.03607	0.05588	0.05715	0.03016		
V L-EQ-14	W44x230	1.118	0.04374	1.09	0.01803	0.4013	0.03099	0.05105	0.05239	0.03016		
➤ L-UN-14	W40x593	1.016	0.1123	1.092	0.04547	0.4242	0.08204	0.112	0.1143	0.05398		
➤ L-ALL-14	W40x503	1.016	0.09548	1.069	0.03912	0.4166	0.0701	0.1001	0.1016	0.0508		
⊐⊏ 2L-EQ-14	W40x431	1.016	0.08194	1.049	0.03404	0.4115	0.05994	0.08992	0.09208	0.0476:		
⊐⊏ 2L-LL-14 ⊐⊏ 2L-SL-14	W40x397	1.016	0.07548	1.041	0.03099	0.4089	0.05588	0.08585	0.0889	0.04604		
⊐⊏ 2L-SL-14	W40x372	1.016	0.07097	1.031	0.02946	0.4089	0.05207	0.08204	0.08414	0.04604		
O RDHSS13	W40x362	1.016	0.06839	1.031	0.02845	0.4064	0.05105	0.08103	0.08255	0.04445		
I WBEAM-14	W40x324	1.016	0.06148	1.021	0.0254	0.4039	0.04597	0.07595	0.07779	0.04286		
T WCOL-14	W40x297	1.016	0.05632	1.011	0.02362	0.4013	0.04191	0.07188	0.07461	0.04286		
AISC 14th Edition Metric	W40x277	1.016	0.05258	1.008	0.02108	0.4013	0.04013	0.0701	0.07303	0.04128		
AISC 13th Edition 2005	W40x249	1.016	0.04742	1.000	0.01905	0.4013	0.03607	0.06604	0.06826	0.03965		
AISC 13th Edition 2005 Me	W40x215	1.016	0.04097	0.9906	0.01651	0.4013	0.03099	0.06096	0.0635	0.0396		
AISC 9th Edition 1989	W40x199	1.016	0.03794	0.983	0.01651	0.4013	0.02718	0.05715	0.05874	0.03969		
AISC LRFD 3rd Edition	W40x133	1.016	0.03734	1.057	0.03607	0.4015	0.06401	0.09398	0.09684	0.0330		
AISC 9th Edition Metric AISC 8th Edition 1978	W40x332	1.016	0.06303	1.036	0.03099	0.3099	0.0541	0.03330	0.03004	0.0452		
AISC 8th Edition 1978	W40x331	1.016	0.06303	1.036	0.03033	0.3033	0.0541	0.08407	0.08573	0.04604		
AISC 6th Edition 1963	W40x327 W40x294	1.016	0.05561	1.036	0.02557	0.3073	0.0341	0.08407	0.08096	0.0460		
ANSI Pipe	W40x234 W40x278	1.016	0.05561	1.026	0.02652	0.3048	0.04502	0.07695	0.08036	0.0444:		
Brazilan Standard, NBR 58	W40x278 W40x264	1.016	0.0531	1.021	0.02616	0.3048	0.04397	0.07391	0.0775	0.0444:		
British Standard 5950	W40x264 W40x235				0.02438							
European	W40x235 W40x211	1.016	0.04458	1.008	0.02108	0.3023	0.04013	0.0701	0.07303	0.04128		
Intergraph Smart 3D tables		1.016	0.04006	1.001		0.2997	0.03607	0.06604	0.06826			
Indian Standard Tables fro	W40x183	1.016	0.03439	0.9906	0.01651	0.2997	0.03048	0.06045	0.0635	0.0396		
Unistrut	<									>		
➡ Chinese GB Tables ➡ User Defined Sections												

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

Unistrut
 User Defined Sections
 MYAISC14
 A_C
 MYLUN13
 MYRECHSS
 ABEAM9

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

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

2.6.10. Creating Members

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

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

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

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

Note: Each time you create a member, the orientation of the cross section will appear in the middle of the element, unless you clear it with command "Clear" (see 2.6.84).

Cross Section	
Table Section:	
<select section="" table=""></select>	~
or Member Dimensions:	
Rectangle Concrete	~
or Same as Member:	
<select existing="" member=""></select>	~
Material	
Steel	~
Sice	•
Releases	Beta (o)
00 Pin-Pin 🗸	0 ~
Split Intersecting Members Split Ending Members	Physical Member
Place Member(s))>>
Section Properties	
	в:
	н:
↑ ^y ↑ z	

You can select:

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

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

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

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

2.6.11. Finding Members

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

2.6.12. Splitting Members

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

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

2.6.13. Splitting to Crossing Members

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

X Split to Members

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

2.6.14. Merging Members

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

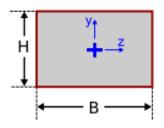
2.6.15. Member Properties

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

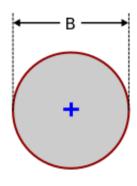
The "Member Properties" form appears, and at the "Model" tab you can enter the Name of the Member (up to 8 characters), the Level that the member belongs to (optional), the Type of the Member (Space Frame or Space Truss), Starting and Ending Joints, Beta Angle, the Groups that the member belongs to,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

Member Properties		×
Model Section Properties Member Loads	Member Temperature Loads Member Distrortions	
General	Section Properties	Releases & Elastic Connection spring values
Name :	Section : 🗸 📈	Quick Selection :
Level :	Ax : Ix :	Start Spring End Spring
	Ay: Iy:	
Type - Incidences	Az : Iz :	□Fy 0 □Fy 0
Type : SPACE FRAME V	Sy: Ey:	□Fz 0 □Fz 0
Start 1	Sz : Ez :	□ Mx 0 Mx 0
End : 2	Yd: Yc:	My 0My 0
Beta Angle : 0	Zd : Zc :	Mz 0Mz 0
Physical Member	Shape Code :	
Groups		End Sizes OR Member Eccentricities (Offsets) Start End
Inactive Remove	Material Properties	
Groups that Member belongs	Material: Steel ~	Sizes :
	E: 1.99948e+08 Density 76.901	X: 0 X: 0
	G: 7.5842e+07 CTE: 1.17e-05	Y: 0 Y: 0
	Poisson 0.3	Z: 0 Z: 0
		OK Cancel Apply Help

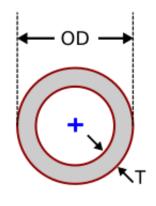
Using the "Section Properties" tab, you can define typical concrete shapes or other shapes that appear in the next image. Shapes having the identifier "Concrete" should be used for thick sections.



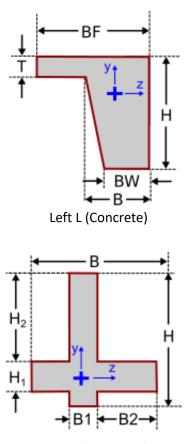
Rectangle (Concrete)



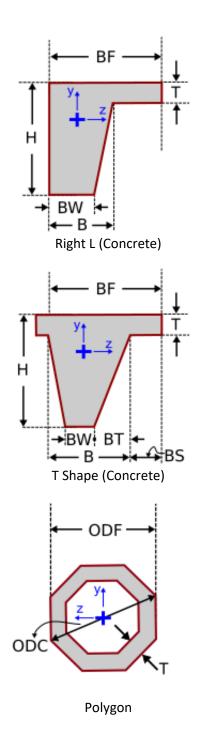
Circle (Concrete)

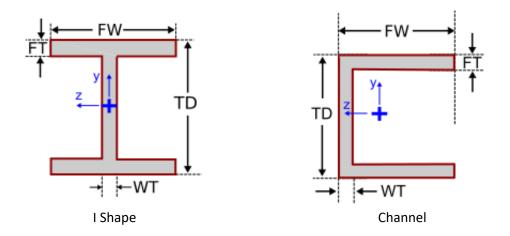






Cross (Concrete)





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.

	Physical Member Propertie	S	
Model Section Properties Member Loa	ads		
General	Section Properties	Releases & Elastic Connection sp	ring values
Name : PM000003	Section : IPE270 IPE European V	Quick Selection :	~
Level : 1	Ax: 0.00459 Ix: 1.6e-07	Start Spring End	Spring
	Ay: 0.00164736 Iy: 4.2e-06	□ Fx 0 □ F	x 0
Type - Incidences	Az : 0.001836 Iz : 5.79e-05	□ Fy 0 □ F	y 0
Type : SPACE FRAME V	Sy: 6.2222e-05 Ey: 0	Fz 0 F	z 0
Start 6	Sz : 0.00042889 Ez : 0	Mx 0M	1x 0
End : 18	Yd: 0.27 Yc: 0.135	My 0N	1y 0
Beta Angle : 90	Zd: 0.135 Zc: 0.0675		1z 0
Physical Member PM000003 V	Shape Code : 1.0		
		End Sizes OR Member Eccentricit	
Groups Remove	Material Properties	Start	End
Inactive Remove	Material : Steel 🗸	Sizes :	0
14	E: 1.99948e+08 Density 76.901	x: 0 x:	0
29 15	G: 7.5842e+07 CTE: 6.49865e-06	Y: 0 Y:	0
21	Poisson 0.3	Z: 0 Z:	0
	Г		
	L	OK Cancel	Apply Help

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

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

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

2.6.16. Member Filters

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

Members can be filtered forming queries of three different categories:

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

You can set multiple (up to 5) conditions of the <u>same</u> 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"

				Filter	Members			
LECT Memb	ers							ОК
	Section	¥	= ~	IPE33	0 IPE European		~	Cancel
AND 🗸	Level	¥	< •	3			~	
AND 🗸	Beta Angle	~	>= \	90			~	
~		~	= \v				~	
¥		¥	= \v				~	
	() (Marshaa Daardha (C		
mber Loads	s (Units: kN, m)				Member Results /	Section Force		
		~		~		~	¥	<u> </u>
× ×	×	*		~	× _	* *	~	
		*		- -	· · · · ·	• •	¥	
~	· · · · · · · · · · · · · · · · · · ·	~		~	×	~	~	
L Query (W	/HERE)						Result	
(nSection = AND (nLev AND (beta/							9 10 11 12 13 14	
Exe	cute >>		Clear				15 16 17 18	
tion							19 20 21	
Items Selec	ted: 70						21 22 23	
ADD T	o Group >>			~			24 25	
	ected after closing form		ake them the only N				26 33	v

2.6.17. Creating Shell Finite Elements

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

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

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

2.6.18. Reverse Incidence Order

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

2.6.19. Finding Shells

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

2.6.20. Shell Properties

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

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

Element Properties ×			
Model Element Loads			
Model Element Loads General Name : Level : 1 Type - Incidences Type : SBHQ6 V Joint 1 : 13 Joint 2 : 14 Joint 3 : 19 Joint 4 : 18 Section Properties Thickness : 0.2 Area :	Material Properties Material: Concrete E: 2.48211e+007 Density 23.5616 G: 9.92846e+006 CTE: 9.9e-006 Poisson 0.17 0.17 Groups Remove Groups that Element belongs		
	OK Cancel Apply Help		

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

2.6.21. Joints Duplicates

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

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

st			Select All
Joint	Duplicate	Merge	
1	9		Unselect All
2	5	✓	
3	25		OK
			Cancel

2.6.22. Joints Floatings

Floating Joints are the joints that are not connected to any member or finite element, therefore they may cause instability in the solution of the mathematical model. Use the ribbon command

I Joints Floatings in the "Find/Change/Check" panel or select "GTS Modeling>Checks>Joint Floatings" from the Menu or type GTSCheckFloatingJoints at the command prompt, floating joints are automatically identified, and using the corresponding dialog as shown below, they can be deleted.

	Floating Joints	×
List		Select All
Joint 1	Deleted	Unselect All
		ОК
		Cancel

2.6.23. Joints Interference

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

use the ribbon command in the "Find/Change/Check" panel or select "GTS Modeling>Checks>Joints Interference" from the Menu or type GTSCheckInteferenceJoints at the command prompt. You then must enter the desired tolerance (Enter Check Tolerance <0.050800>).

2.6.24. Members Duplicates

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

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

D	uplicate Members			×
	List			Select All
	Member	Duplicate	Delete	
	3	5		Unselect All
	4	6	✓	
				ОК
				Cancel

2.6.25. Members Zero Length

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

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

2.6.26. Physical Members

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

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

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

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

- #3. Checks that all member do not have internal releases. If not, there is an error messages: Physical Member %s: Member %s has internal RELEASES Physical Member %s: Starting Member %s has internal RELEASES at its end Physical Member %s: Ending Member %s has internal RELEASES at its start
- #4.Checks that there is a sequential order of members. If not, there is an error message: Physical Member %s: Members %s (end) and %s (start) do not have common joint (are not SEQUENTIAL)
- #5.Check that all members form a straight line.. If not, there is an error message: Physical Member %s: Member %s is not LINEAR with the physical member
- If all 5 checks are successful an informative message appears: Physical Member %s: OK

2.6.27. Shells Duplicates

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



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

Duplicate Shells			×
List			Select All
Shell	Duplicate	Delete	
3	5		Unselect All
4	6		
			OK
			Creat
			Cancel

2.6.28. Names Duplicates

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

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

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

2.6.29. Renumber Names

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

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

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

2.6.30. Database Integrity

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

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

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

2.6.31. Meshing along a curve

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

a Line, an Arc or a Circle, from the ribbon command ^{Curve} or from the menu "GTS Modeling>Mesh Generation>1D Along Line or Curve or Circle" or by typing GTSMesh1D at the command prompt.

Select Mesh Properties ×				
Generate				
Members Elements				
Material Steel V				
Element Attributes				
Type BPHQ V Thickness 0.20				
Member Attributes				
Type FRAME V Beta 0.0				
Section V				
Spacing U Direction				
O Uniform 4 ∨				
O Variable				
O Defined by Line/Curve				
Spacing V Direction				
● Uniform 4 V				
Variable				
O Defined by Line/Curve				
Spacing W Direction				
Uniform 4				
🔾 Variable 🦳				
O Defined by Line/Curve				
Labeling				
More >> U Primary Direction				
Preview Create Close				

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

1D

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

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

Total # of spaces :	0	Curve L	.ength :	0.000000
Current # Spaces :		Current	t Length :	
Remaining # Spaces :	0	Remain	ing Length :	1847.094685
ariable Spacing # Spaces	Distance		or Percent	^
				- 1

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

Enter Labeling Rules		
Properties		
Joints Prefix		
First Joint's ID	1	
Elements Prefix		
First Element's ID	1	
Members Prefix		
First Member's ID	1	
< < Apply Calculate ID)s	

2.6.32. Meshing between two lines

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



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

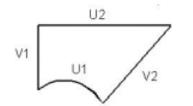
The dialog has the same options as in the 1D mesh command and in addition you can also define:

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

2.6.33. Meshing between four lines

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

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



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

2.6.34. Meshing inside a polyline

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



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

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

Select Mesh Properties				
Generate				
Material Steel V				
Element Attributes				
Type SBHT6 V Thickness 0.20				
Mesh Geometry				
External Boundary obj-563				
Boundary Maximum Edge Size 3.242418				
✓ Do not split boundary more than Max				
Element Maximum Area 10.513274				
Mesh Quality Very Low				
Internal Boundaries Internal Joints				
Add + Remove -				
Multi+ Multi - Add Remove				
Spacing Extrude Direction				
● Uniform 4 V				
Variable				
O Defined by Curve, Size: 3.242418				
Labeling More >>				
Preview Clear Create Close				

- The *Material* of the elements to be generated
- Type of Finite Elements from the available GT STRUDL Finite Element library
- Thickness of Finite Elements
- The *Maximum 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.

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

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

2.6.35. Meshing by extruding a polyline

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

3D

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

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

2.6.36. Meshing using 3 curves

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



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

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

2.6.37. Array 3D Advanced

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

Array 3D

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

2.6.38. Soil Springs



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

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

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

General Information	Ks Values
In-plane (orthogonal) springs	-0.00 0.08
In-plane Spring Ratio : 0.000000	0.08 0.15
	0.15 0.23
Direction	0.23 0.31
⊖ Global X ⊖ Global Y	0.31 0.38
Append Existing Spring Values	0.38 0.46
	0.46 0.54
	0.54 0.62
Modulus of Subgrade Reaction	0.62 0.69
Ks : 1.000000	0.69 0.77
	0.77 0.85
	0.85 0.92
Place Springs >>	0.92 1.00

2.6.39. Export to CAESAR II

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

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

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

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

2.6.40. Convert Lines/Polylines to Members/Shells

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



from the ribbon command convert them to structural members. Or open an existing DXF or DWG, select some lines and polylines, and convert them to members and elements.

D:\Hexagon\DXFReader\Run\DXFReader.dxf → 477 Line 2 Polygon entities found!	Number of Joints Read: 243
	Number of Members Read: 477
Output File: D:\Hexagon\DXFReader\Run\DXFReader.gti	Number of Shell Elements Read: 2
Done! 243 nodes 477 members 2 elements are created	
	Number of Sections Read: 0
Create Joints, Members and/or Finite Elements Create Construction Points, Lines for Meshing	Number of Groups Read: 0
Create construction points, Lines for Mesning	Number of Load Cases Read: 0
Read >>> Properties Convert	Number of Load Combinations Read:
Help Exit	Number of Load Combinations Read:

2.6.41. Model Wizard

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

🔆 Model Wizard

or by typing GTSModelWizard at the command prompt.

GTStrudl ModelWizard 4.2				
To start using Mod the palette below.	lefWizard, select a str	ucture type from		
H		AT I		
Plane Frame	Braced Frame	Space Frame		
Finite Element Mesh	Circular Plate	Bridge Truss		
Tank	Vault / Rect Tank	Continuous Beam		
1	OK	Cancel		

2.6.42. Groups

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

 _	_
	_
	_
	_
	_
 -	
_	

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

Using the Groups Form you can:

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

roup Proper	ties		Add Group
ID	Name	Physical	
1	Columns		Delete Group
2	Beams		
3	Girders		OK
4	Bracing		
5	Slabs		Cancel

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

For the second second

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

+Shells ribbon icon, or "GTS Modeling>Groups>Add Shells" or by typing GTSGroupShells at the command prompt

2.6.43. Self - Weight

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

Self Weight o	r Dead Loads
Load Information	
Name : SW v Description : Self Weight	
Create New Save /	Modify Delete
Loads applied parallel to this Glo	obal Axis
O Negative Y	O Positive Y
Negative Z	O Positive Z
O Negative X	O Positive X
Factor :	☑ Indude Finite Elements
	OK Cancel

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

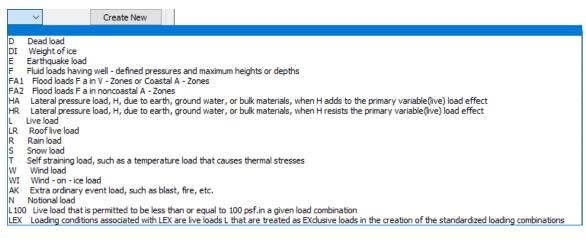
- the global direction of the self-weight
- the load factor (default = 1.0) for the self-weight
- if the self-weight of finite elements will be taken into account or not

2.6.44. Load Cases

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

Load Case				×
Load Case Information				
Load ID (up to 8 chars)		Design variable :	~	Create New
Description :	Live Load			Save / Modify
Load ID List :	LL			Delete
	PL			
				Exit

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



2.6.45. Joint Loads

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

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

Joint Properties		×
Model Joint Generalized Loads		
Applied Load Cases	Loading	
PL \$ Point Load \$(1)	Load Case : PL Create New Description : Point Load Save / Modify Loads in this Load Case : 1 V	
	Applied Load Values	
	Force X : 0 Force Y : 0 Force Z : -3 Moment X : 0 Moment Y : 0 Moment Z : 0	
	Applied Displacement Values	
Empty Load Cases	Tran X : Tran Y : Tran Z : Rot X : Rot Y : Rot Z :	
	OK Cancel Apply Hel	p

2.6.46. Member Loads

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

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

				Me	ember Pro	perties		×
Model	Section Properties	Member Loads	Member Tempera	ature Loa	ads Member	Distrortions		
Арр	lied Load Cases		Loading					
	. \$Live Load \$[125]		Load Case : (Description : (Loads in this Lo	LL Live Loa ad Case		v		Create New Save / Modify Delete
			Load Distribution		Member	Start		
			Uniform		*	/		
	oty Load Cases						• L2	
PI	L \$ Point Load \$[0]		-Load Type and D	irection -		Loading Magnitude	-Loading Locatio	n from Start
			Force	⊖x ⊖y	OLocal	V1: -5	Fractional	L1: 0
			OMoment	€Z	Global	V2: 0	○ Absolute	L2: 1
						ОК	Cancel	Apply Help

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

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

Member Properties						
Model Section Properties Member Load	Is Member Temperature Loads	Member Distrortions				
Applied Load Cases	Loading					
	Load Case : Description : Loads in this Load Case :	Create New Save / Modify Delete				
	Temperature Change	depth Change : 10				
	Location: Length experiencin	g temperature change				
Empty Load Cases LL \$ Live Load \$[0] PL \$ Point Load \$[0]	Fractional (% 0-1) Absolute (len)	Starting Location : 0				
		Ending Location :				

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

2.6.47. Shell Loads

A Shell Load can be entered from the ribbon command ^{III} 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.

oplied Load Cases	Loading	101			Create New
	Description :	general			Save / Modify
	Loads in this Lo	oad Case :	*		Delete
	Force Type	Direction	System	Load Distribution	Values (B or S)
	OBody	Ox	OLocal	Uniform	v1: -1.00
	Surface	OY	O Planar		v2:
	() bandee	01	Global	() Variable	v3 :
		⊻ ⊚z	O Projected		v4:
npty Load Cases					
101 3[0]					

2.6.48. Area Load

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

Area Load ×
Generate Name : AL1 Area Load 1
Description : Create New Save / Modify Delete
Load - Direction
Global Direction Perpendicular to the Loading Plane : O X O Y O Z Plane Tolerance : 0.0508
Elevation Plane Perpendicular at : Value (coordinate) Joint
Distribution Two way X One way Y Custom X 0.0000 Y 0.0000
Advanced Features Define Outline Region >> Reset
Exclude Area >> Reset Ignore Members >> Reset
Display >> Clear OK Cancel

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 <u>positive</u> value is applied in the <u>negative</u> global DIRECTION.
- The Global direction of the loading plane (X, Y or Z) and the tolerance
- The position (Elevation) of the plane, defined by a coordinate or an existing Joint. All members belonging to the plane having this elevation are located and loaded by GT STRUDL.
- The Distribution of the load (one way or two way)

Advanced Optional Features:

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

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

2.6.49. Wind Load ASCE 705

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

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

Wind Load Generation ASCE 7-05	×
Wind Load ID: WLx Delete Wind Load Description: Wind Load X	
General Wind Load Data Member Wind Load Data	
Active Units: M KN DEG SEC StdMASS Change Units	
V: 20 • mph · m/s Gust Factor (G): 0.85	
Elevation Axis	
Exposure Category Directionality Factor (Kd): 1 Importance Factor (I): 1	
Topographic Factor © Kzt Kzt: 0 © K1, K2, K3 K1: 0 K2: 0 K3: 0	
Minimum Velocity Pressure (QZmin): 0.478803 Gross Area (Ag): 0	
Added Force Area (AFadd): 0	
Review Data Save Commands Report Wind Load Only Save Load Done	

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

Wind Load Generation ASCE 7-05
Wind Load ID: WLx Delete Wind Load Description: Wind Load X
General Wind Load Data
Active Units: M KN DEG SEC StdMASS Change Units
Select Members Selected Members: 123
Wind Load Component Type: Industrial Frame
Af and Cf Parameters
Properties C Round D:
C Rectangle B: 0 H: 0
O Af: 0
Hctrs: 0 Dctrs: 0 C-T-R-S ID:
THice: 0 THins: 0 CDg: 0
Misc. Parameters/Overrides
Kz: 0 Cf: 0 QZ: 0 FLD: 1
Add to Member Data Clear Member Data Delete Selection
Review Data
Save Commands Report Wind Load Only Save Load Done Help

Active Units					×
Length C Inches C Feet C Millimeters C Centimeters Meters	Force Pounds Kips Tons Kilograms Metric Tons Newtons KiloNewtons	Rotation C Radians C Degrees C Cycles	C Fahrenheit C Fahrenheit C Centigrade	Time © Seconds © Minutes © Hours	Mass Standard Mass Grams Kilograms Metric Tons
	C MegaNewtons		OK	Help	Cancel

2.6.50. Wind Load ASCE 710

A Wind Load can be entered from the ribbon command or from the menu "GTS Modeling>Loads>Wind Load ASCE 710" or by typing GTSWindLoadsASCE710 at the command prompt.

Wind Load Generation ASCE 7-10	Х
Wind Load ID: WLx Delete Wind Load Description: Wind Load X	
General Wind Load Data Member Wind Load Data	1
Active Units: M KN DEG SEC StdMASS Change Units	
Design Wind Speed V: 20 Image: Complex Speed Gust Factor (G): Image: Complex Speed	
Elevation Axis C Y C Z Wind Direction Angle: 0	
Exposure Category Directionality Factor (Kd): 1	
Topographic Factor	
Г Каза Каза Каза Каза Каза Г Каза Г Каза Г Каза Г Каза Г Г Каза Г Г Каза Г Каза Г Г Г Каза Г Г Г Каза Г <td></td>	
Minimum Velocity Pressure (QZmin): 0.766085 Gross Area (Ag): 0	
Added Force Area (AFadd):	
Review Data	
Report Wind Load Only Save Load Done Help	

Wind Load Generation ASCE 7-10	K
Wind Load ID: WLx Delete Wind Load Description: Wind Load X	
General Wind Load Data Member Wind Load Data Active Units: M KN DEG SEC StdMASS Select Members: 123	
Wind Load Component Type: Industrial Frame	
© Properties C Round D: 0 C Rectangle B: 0 H: 0 C Af: 0	
Hctrs: 0 Dctrs: 0 C-T-R-S ID: THice: 0 THins: 0 CDg: 0 0	
Misc. Parameters/Overrides Kz: 0 Cf: 0 QZ: 0 FLD: 1	
Add to Member Data Clear Member Data Delete Selection	
Review Data Save Commands Report Wind Load Only Save Load Done Help	1

2.6.51. Seismic Load

You can define Seismic Loads using a similar form with GT Shell. Seismic Loads can be entered

from the ribbon command or from the menu "GTS Modeling>Loads>Seismic Load" or by typing GTSSeismicLoading at the command prompt. In addition you can select joints (to define story heights) interactively with mouse picks in the CAD environment.

Seismic Load Generation X
Load X Name SLX Load Y Name Load Z Name Description Seismic Load Global Description Description Standard General Data
Standard C ASCE 7-05 (* ASCE 7-10 Story Heights Story Heights from C Values Story Heights Story Heights from C Values EXISTING 17 35 55 77 97 117 Load Direction Values Y Z Lateral Seismic Factor (LSF) 1.00 Seismic Weight Load SW Vertical Seismic Factor (VSF) 0.00 Seismic Dead Load
 C Specified Response Coefficient (Cs) SDS Response Coefficient from Standard Ground Motion Spectral Response Accelerations Site Class B: Rock ▼ Risk Category IV: Essential Facilities ▼ Long Period (TL) 12 Seconds I SS, S1 from Map USA ▼ Latitude 36 Longitude -89 Degrees C SS S1 9 C SDS SD1 S1 9
Seismic Forces (* Fundamental Period from Standard SMR: Steel Moment-Resisting Analysis Mode, Direction X Y Z C Specified Fundamental Period (Ta) Seconds Response Modification Coefficient (R) 3.5 Accidental Torsional Factor (ATMF) 1.00
OK Done Help

2.6.52. Load Combinations

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

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

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

	New Form Load or	Load Combinat	tion	×
Load Information Name : CB1 Description : Load Combination	n 1		Type Load Combination Form Load	
	1.35000 'LL' 1.50000 'PL' 1.50000	ADD >> Factor : 1.00000	SW 1.35000 LL 1.50000 PL 1.50000	Delete Item
All Formed Loads or Combinations CB1 (Load Combination 1) 'SW' Edit Delete	: 1,35000 'LL' 1.50000 'PL' 1.50000			
				Done

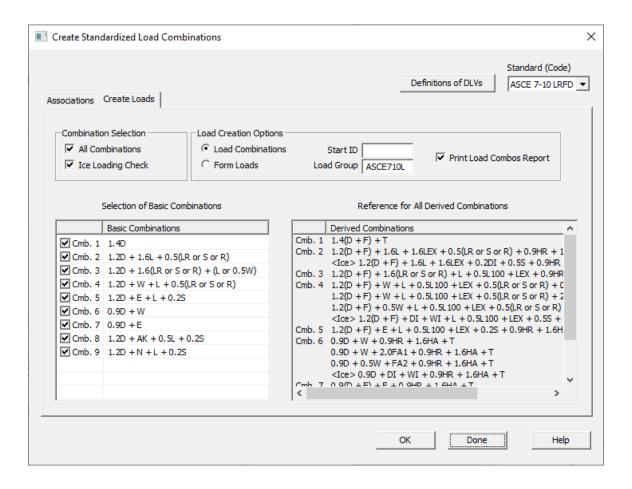
2.6.53. Standardized Combinations

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

Standardized Combinations

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

Create Sta	ndardized Load Combinations			
sociations	Create Loads			Definitions of DLVs ASCE 7-10 LRFD
	Unassociated Loads		Loa	ds Associated with Design Load Variables (DLVs)
Load ID	Description		DLV	Associated Load IDs
D1	D1		D	
D2	D1		DI	'DI'
F	F		E	'E1' 'E2' 'E3'
· ·			F	
		Associate >>	FA1	'FA1'
			FA2	
			HA	'HA'
			HR	'HR'
		Remove	L	Υ.
		Association	LR	'LR1'
			R	
			S	'S'
			T	Т
		Reset	W	1
		Reset	WI	'WI1' 'WI2'
			AK	
			N	'N1' 'N2'
			L100	
			LEX	'LEX1' 'LEX2'
,				
				OK Done Help
			_	



2.6.54. Steel Design Parameters

You can specify steel design parameters for AISC14, EC3, IS800, CSA-2014 and all other codes.

Steel design parameters can be defined from the ribbon command *Pesign* or menu "*GTS Modeling>Steel Design Parameters*" or by typing GTSSteelDesignParameters at the command prompt. Parameters can be applied to ALL members or to specific members that can be selected interactively with mouse picks in the CAD environment.

DE Name EC	3-2005		•	Units MET	ERS K	ILONEW	TONS
Commonly Used A	II Parameters A	lphabe	tic	Member List Proc	essing		
Parameter Name	Parameter Valu Values)	ues (D	efault 🔺		. 1		ок
Print-K	YES			All Mer	nbers		
Buckling Length	Parameters						Canc
LY	Computed			Member Lis	t Selection	UNITS	Cane
LZ	Computed						
FRLY	1.0						Help
FRLZ	1.0			Same as Prev			
Flexural-Torsiona	al Buckling Para	meter	s	Lis	st		
LX	Computed						
lp Z		^	Selected Order Alph	nabetic			
braced length for out the local Z ax	is of the		Parameter Names	Parameter Values	Member, Joint or Load	UNITS	
ofile. The default the length of the			CODE	EC3-2005	ALL MEMBERS		
- ne parameter valu	e 'Computed'		LZ	2	MEMBER 32 31 30 5	METERS	KILONEW
eans that the para							

2.6.55. Create GTI



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

	Create GT.STRU	DL Input File	×
GTI Commands			
GTI File :	F:\00.STRUDL Europe\05-AT	LAS \Models \Southern \PG2	
Perform Stiffn	ess Analysis		
Append Other GTI	Files/Macros		
			+ - Up
Copy Comman	ds from GTI Files/Macros (not	CINPUT)	Down
Create Commands	to Read Results		
Read Joint Di	splacements		
Read Member	Forces		
Read Section	Forces	Number of Sections:	10
Read Section	Displacements	Number of Sections:	10
Read Finite El	ement Results		
Read Code C	heck Results		
		ОК	Cancel

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

2.6.56. Edit GTI

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

2.6.57. 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.

2.6.58. Read Analysis Results

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

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

Read GT.ST	RUDL Results	×
Read DBX Results		
GTI Directory : F:\00.STRUDL Europe\0	5-ATLAS\Documentation	
✓ Read Joint Displacements		
✓ Read Member Forces		
Read Section Forces	Number of Sections:	10
✓ Read Section Displacements	Number of Sections:	10
Read Finite Element Results		
Type of element result :	<all></all>	~
Surface :	<all></all>	~
Read Code Check Results		
GTI Commands		
Corresponding GT STRUDL commands:		
DBX BINARY WRITE REPLACE JOINT RESULTS JOINTS WRITE REPLACE MEMBER RESULTS MEMB WRITE REPLACE SECTION FORCES NS 10 WRITE REPLACE SECTION DISPLACEMENT	ERS EXISTING MEMBERS EXISTING	ING
	OK	Cancel

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

2.6.59. Import GTI

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

2.6.60. Set Views

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

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

View Options
Z Axis Up
Available Views :
GTS Isometric View
GTS XY Plane along -Z axis GTS XZ Plane along -Y axis
GTS YZ Plane along -X axis

2.6.61. 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 ^{ID} 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 A Frame or from the menu "GTS Display>Frame" or by typing GTSSet1D at the command prompt. When the wireframe view is selected, all members and shell edges are displayed as lines.

If some parts of the structure are hidden (i.e. using Level's form) you can display the whole

structure from the menu "GTS Display>All Levels ON" All Levels ON or by typing GTSSetAllVisible at the command prompt.

2.6.62. Analytical/Physical Member View

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

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

2.6.63. Colors and Visible Elements

You can control the color of each member or element, and its visibility from the ribbon command ^{O Colors} or from the menu "*GTS Display>Colors*" or by typing GTSColorView at the command prompt.

Using the tab "Sections" in the "Color Options" form shown below, you can assign a different color for each cross-section profile and set its visibility to ON or OFF. By pressing "Reset Colors", all colors are set to defaults. By clicking on the color value cell a pop-up dialog appears, where you can select the color that you want to be assigned to this section. If you select "BYLAYER" or

256 then the color is assigned by the color set at GTS Disaplay>Options $^{\textcircled{Options}}$, as explained in <u>2.6.64.</u>

Sections		Color	Visible
HE320B		161	
IPE330		191	
IPE120		1	
60x60x5		50	
Note: If you s at View > Opt	et a color to 256 (BYLAYE ions Dialog	R) then the color is	

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

Groups	Color	Visible
Columns	161	✓
Beams	191	
Girders	1	
Bracing	50	
UnGrouped data	7	✓
lote: If you set a colorto 256 (BYLAYE t View > Options Dialog	R) then the color is	defined by the color set

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

2.6.64. Display Options

You can set the display options from the ribbon command ^{Options} or from the menu "GTS Display>Options" or by typing GTSDisplay at the command prompt. Using the "Display Options" form shown below, you can:

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

- set type of Annotation Format: Decimal, Exponential or Generic (automatic) and the number of decimal places
- set object sizes
- set the shrink factor for finite elements and members. This option makes it is easier for you to detect members that lie along finite element edges.
- Do Not Display Thickness in 3D. If you check this option, elements will be displayed as being 2D instead of a 3D display which shows the thickness of the elements. This option may increase the display speed in very large finite element models.
- Display Members as Analytical or Physical (see 2.6.62)

Display O	ptions ×
Visible Objects	Visible Labels
✓ Joints	✓ Joints
✓ Members	✓ Members
✓ 2D Elements	✓ 2D Elements
✓ 3D Elements	✓ 3D Elements
Label Settings - Font Sizes	Object Sizes
Joints :9.842520Members :9.8425202D Elements :9.8425203D Elements :9.842520Annotation (pts) :10.000000Annotation Format:Decimal ♥Decimal Places :2	Joint : 1.968504 Load Arrowhead (pts): 10.00000 Display Members / Elements Shrink Factor : 1.0 v Do Not Display Thickness in 3D Members As: Analytical v
Scale Factors Concentrated Load (pts) : 72.000000 Distributed Load (pts) : 72.000000	OK Cancel

2.6.65. 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.

Annotate ×
Inquire
 Coordinates
O Dimensions / Distance
◯ Joint Names
O Member - Element Names
O Member Forces
Arrowhead (pts): 10.000000
Font Size (pts) : 10.000000
Annotate Close

The available inquire options are:

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

2.6.66. Select CAD Modeler's entities

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

GT	S Select ×
Select Options	
 Line 	Bounded
O Plane	
○ Volume	OUnBounded
Filter Options	
Joints M	1embers Elements

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

Volume or its extension

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

2.6.67. Display Member Local Axes

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

In the legend, the X axis is displayed in cyan, Y axis in red and Z axis in yellow. The size of the arrow and its arrowhead is controlled by the value given at Display Options > Object Sizes > Load Arrowhead and the size of the legend fonts is controlled by the value given at Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in <u>2.6.64</u>).

2.6.68. Display Member Releases

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

A text identifying the released degrees of freedom appears next to members having releases. No text appears for members that do not have releases. The size of the legend fonts is controlled by the value given at Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in 2.6.64).

2.6.69. 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 <u>2.6.64</u>).

2.6.70. Display Joint Supports

You can view the support status of each joint from the icon Display) or from the menu "GTS Display>Joint Supports" or by typing GTSDisplaySupports at the command prompt.

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

arrowhead is controlled by the value given at Display Options > Object Sizes > Load Arrowhead and the size of the legend fonts is controlled by the value given at Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in 2.6.64).

2.6.71. Display Joint Loads

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

Display Loads	E
Load Case	
PL	~
Display Loads Applied To	
Joints Members	Elements
Display Options	
Scale Factor Concentrated (pts)	72.000000
Scale Factor Distributed (pts) :	72.000000
Arrowhead Size (pts) :	10.000000
Font Size (pts) :	8.000000
Show Clear	Close

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

2.6.72. Display Member Loads

You can view the member loads applied in the structure from the icon ^{•••} Member (Ribbon GTS Display) or from the menu "GTS Display>Member Loads" or by typing GTSDisplayMemberLoads at the command prompt.

Display Loads		x
Load Case		
PL	~	
Display Loads Applied To		
Joints Members	Elements	
Display Options		
Scale Factor Concentrated (pts)	72.000000	
Scale Factor Distributed (pts) :	72.000000	
Arrowhead Size (pts) :	10.000000	
Font Size (pts) :	8.000000	
Show Clear	Close	

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

2.6.73. Display Shell Loads

You can view the finite element loads applied in the structure from the icon ^{III} Area</sup> (Ribbon GTS Display) or from the menu "GTS Display>Shell Loads" or by typing GTSDisplayElementLoads at the command prompt.

Display Loads	2
Load Case	
11	~
Display Loads Applied To	
Joints Members	 Elements
Display Options	
Scale Factor Concentrated (pts)	72.000000
Scale Factor Distributed (pts) :	72.000000
Arrowhead Size (pts) :	10.000000
Font Size (pts) :	10.000000
Show Clear	Close

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

2.6.74. Display Area Loads

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

2.6.75. Display Deformed Structure

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

Select Load Case		
Load Case		
<select case="" load=""></select>	¥	

You can switch back to original view from the icon ^{II} ^{Undeformed} (Ribbon GTS Display) or from the menu "GTS Display>Undeformed Structure" or by typing GTSResetJointDisplacements at the command prompt.

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

2.6.76. Annotate Joint Displacements

At the deformed state, you can annotate joint displacements of the structure from the icon Annotate Displacements (Ribbon GTS Display) or by typing GTSAnnotateJointDisplacements at the command prompt. This command is valid when the deformed shape of the structure is visible, where you select a joint and then a point in screen for annotation position.

2.6.77. Display Displacements

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

Displaceme	nts ×
Load Case / Load Combinat	tion
SW \$ Self Weight	~
Display Options	0.100
Scale Factor (Values) :	
Font Size (pt) :	10.00
Annotation Format: De	ecimal 🗸
Decimal Places :	2 🗸
Hide Model	
Display >>	
Annotate >	Legend >
Animation Options	
Frames :	7
Animation Speed % :	1 4
Generate Animation Frames	Animate >>
Clear	Close

The "Displacement" form appears where you can select:

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

2.6.78. Display Member Diagrams

You can view the force and moment diagrams from the icon ^{The Diagrams} (Ribbon GTS Display) or from the menu "GTS Display>Member Diagrams" or by typing GTSDisplayMemberForces at the command prompt.

Member D	iagrams ×
Load Case / Load Com	bination
SW \$ Self Weight	~
Envelope	
Values	
MZ Moment	~
Display Options	
Scale Factor (Values)	0.100
Font Size (pt)	10.00
Annotation Format:	Decimal 🗸
Decimal Places :	2 🗸
Label Max & Min	Positive Sign
Display	>>
Annotate >	Legend >
Clear	Close

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

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

2.6.79. Display Finite Element Results

You can view the finite element results from the icon ^{Herents} (Ribbon GTS Display) or from the menu "GTS Display>Element Results" or by typing GTSDisplayElementResults at the command prompt.

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

A	Graphics Performance	x
۲	rdware Setup	
V	deo Card: NVIDIA GeForce GTX 660M	
D	iver Version: 9.18.13.697 Virtual Device: gdi13.hdi(Software)	
Ef	iects Settings	_
Н	ardware Acceleration Off	
	sable hardware acceleration only if you are experiencing aphics issues or have an incompatible video card.	

Liementi	(Courto
Load Case	
PL \$ Pressure load	¥
Туре	
Resultants	~
Мхх	~
Middle	~
Display Options	
Scale Factor (Values)	0.100
Font Size (pt)	10.00
Annotation Format:	Decimal 🗸 🗸
Decimal Places :	2 🗸
Display	>>
Annotate >	Legend >
Animation Options	
Frames :	7
Animation Speed % :	10 🗸
Generate Animation Frames	Animate >>
Clear	Close

Element Results

The "Element Results" form appears where you can select:

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

2.6.80. 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.

2.6.81. Display Member Code Check Results

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

Code Check	Results ×
Display Text	
 None 	
O Actual/Allowable S	tress Ratios
O Controlling Stress	Code Provision
O Actual/Allowable K	L/r Ratios
🔘 Controling KL/r Co	de Provision
All Greater Than	0.00
0.1	0.00
O Less Than	0.00
Display Options Font Size (pt)	10.00
Display	>>
Legend >	
Clear	Close

2.6.82. Results Datasheets

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.

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

Description	Menu Item	Command	
Displacements	GTS Display>Results	GTSDataSheetJointDisp	
Displacements	Datasheets> Displacements	GISDataSheet001htDISp	
Member Forces	GTS Display>Results	GTSDataSheetMemberForces	
Member Forces	Datasheets> Member Forces	GISDataSheetMeniberForces	
Section Forces	GTS Display>Results	GTSDataSheetMemberForces	
Section Forces	Datasheets> Section Forces	GISDataSheetMemberroites	
Reactions	GTS Display>Results	GTSDataSheetReactions	
Reactions	Datasheets> Reactions	GISDataSheetKeactions	
Stresses	GTS Display>Results	GTSDataSheetStresses	
Stresses	Datasheets> Stresses	GISDataSheetStiesses	
Code Check	GTS Display>Results	GTSDataSheetCodeCheck	
	Datasheets> Code Check	GISDataSheetCodeCheck	

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

oint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y
1	SW	0.0000	0.0000	0.0000	0.0000106	-0.0000138
2	SW	0.0000	0.0000	0.0000	-0.0000232	0.0000019
3	SW	0.0000	0.0000	0.0000	-0.0000178	0.0000063
4	SW	0.0000	0.0000	0.0000	-0.0000217	-0.0000060
5	SW	0.0000	0.0000	0.0000	-0.0000299	0.0000074
6	SW	0.0000	0.0000	0.0000	-0.0000240	-0.0000108
7	SW	0.0000	0.0000	0.0000	-0.0000245	0.0000075
8	SW	0.0000	0.0000	0.0000	-0.0000541	0.0000093
9	SW	0.0000	0.0000	0.0000	-0.0000309	-0.0000440
10	SW	0.0000	0.0000	0.0000	-0.0000478	0.0000193
11	SW	0.0000	0.0000	0.0000	-0.0000426	-0.0000289
12	SW	0.0000	0.0000	0.0000	-0.0000393	0.0000310
13	SW	0.0000	0.0000	0.0000	-0.0000511	-0.0000206
14	SW	0.0000	0.0000	0.0000	-0.0000470	0.0000164
15	SW	0.0000	0.0000	0.0000	-0.0000420	-0.0000298
16	SW	0.0000	0.0000	0.0000	-0.0000307	0.0000099
17	SW	0.0000	0.0000	0.0000	-0.0000394	0.0000016
18	SW	0.0000	0.0000	0.0000	-0.0000175	-0.0000173
19	SW	0.0000	0.0000	0.0000	-0.0000129	0.0000066
20	SW	0.0000	0.0000	0.0000	-0.0000321	0.0000080
21	SW	0.0000	0.0000	0.0000	-0.0000101	-0.0000294
22	SW	0.0000	0.0000	0.0000	0.0000268	0.0000065
23	SW	0.0000	0.0000	0.0000	0.0000194	0.000086
24	SW	0.0000	0.0000	0.0000	0.0000164	0.0000071
25	SW	0.0000	0.0000	0.0000	0.0000193	0.0000075
26	SW	0.0000	0.0000	0.0000	0.0000132	0.000096
27	SW	0.0000	0.0000	0.0000	0.000088	0.000062
28	SW	0.0000	0.0000	0.0000	-0.0000073	0.000003

2.6.83. Report Builder



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

2.6.84. 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 $\textcircled{\mbox{Clear}}$ (Ribbon GTS Display) menu "GTS Display>Clear Results Laver" typing or from the or bv GTSDisplayResultsClear at the command prompt. This command should be given after any of the previous "display" commands.

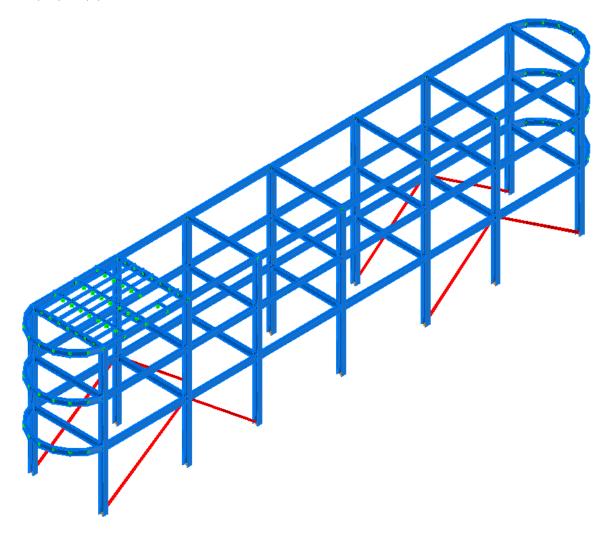
2.6.85. Version

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

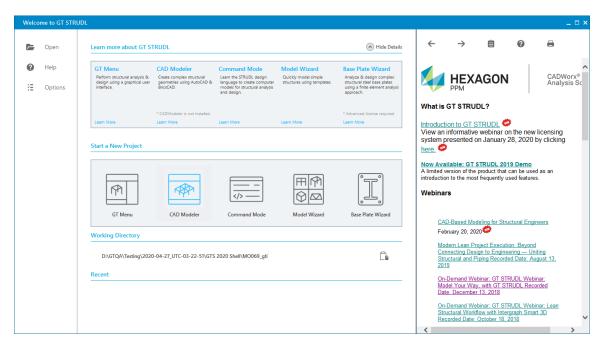
3.1. Introduction

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.

mome ins	ert Annotate Parametric V	new Manage	e Output Collaborate GTS Modeling	GIS Display							
≫ Units ■ Materials	Levels Grid All Levels ON	Generate At Level Support	Sections Generate Vertical Split	 Find Joint * Change Joint * Joints Duplicates * 	☐ Quad ▲ Triangle	10 20 30 Lurve 2Curves Extrude	Array 3D L	Uist · · · · · · · · · · · · · · · · · · ·	GTI GTS Create Execute Input File GT STRUDL	A Frame im 30 im 30 im 30 im Analytical/Physical ⊙ Set View ⊙ Options Oclors	Clear
Model	Levels	Joints	Members	Find/Change/Check	Shells	Meshing	Advanced Gro	oups Loads	GT STRUDL	Display	Inquire

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 $\overset{\text{W Units}}{\longrightarrow}$ and select *Meters (m)* and *KiloNewtons* in the *Units Form*.

Units		×				
Length	Force	Angles				
O Inches (in)	O Pounds (lbs)	Degrees				
O Feet (ft)	◯ Kips					
Meters (m)	◯ Tons	○ Cycles				
O Centimeters (cm)	◯ Kilograms					
O Millimeters (mm)	O Metric Tons	Time				
Temperature	○ Newtons	 Seconds 				
◯ Fahrenheit	 KiloNewtons 	○ Minutes				
Centigrade		OHours				
Scale non-structural CAD entities (grids, structural lines, curves etc) OK Cancel						

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

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

SY SZ 3.34E-005 9E-005 5.3E-005 0.000144 857E-005 0.0002157 0.0001111 0.0003113 0.0001511 0.0004256 0.0002 0.00057 0.000282 0.0007355 0.0003267 0.000383 0.000346 0.001148 0.0005707 0.001376 0.0005707 0.001678 0.000616 0.001226	SHAPE 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2
5.3E-005 0.000144 857E-005 0.0002157 0.0001111 0.0003113 0.0002 0.0004256 0.0002 0.00057 0.0003267 0.0009383 0.0003267 0.0013148 0.0004707 0.001376 0.0004707 0.001678 0.0005707 0.001678 0.000646 0.002157	1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2
857E-005 0.0002157 0.0001111 0.0003113 0.0001511 0.0004256 0.0002 0.00057 0.0003267 0.0009383 0.0003267 0.001148 0.0004707 0.001376 0.0005707 0.001376 0.0005707 0.001678 0.000566 0.001926	1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2
0.0001111 0.0003113 0.0001511 0.0004256 0.0002 0.00057 0.002582 0.0007355 0.0003267 0.0007383 0.000346 0.0011148 0.0004707 0.001376 0.0005707 0.001578 0.0005616 0.001326	1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2
0.0001511 0.0004256 0.0002 0.00057 0.002582 0.0007355 0.003267 0.000383 0.0004707 0.001148 0.0004707 0.001376 0.0005707 0.001376 0.0005616 0.001926 0.0005616 0.0012157	1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2
0.0001511 0.0004256 0.0002 0.00057 0.002582 0.0007355 0.003267 0.000383 0.0004707 0.001148 0.0004707 0.001376 0.0005707 0.001376 0.0005616 0.001926 0.0005616 0.0012157	1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2
0.0002 0.00057 0.002582 0.0007355 0.003267 0.0009383 0.003946 0.001148 0.004707 0.001376 0.0005707 0.001678 0.000616 0.001926 0.000646 0.002157	1.2 1.2 1.2 1.2 1.2 1.2 1.2 1.2
0.0002582 0.0007355 0.0003267 0.0009383 0.0003946 0.001148 0.004707 0.001376 0.0005707 0.001678 0.000616 0.001926 0.000646 0.002157	1.2 1.2 1.2 1.2 1.2 1.2 1.2
0.0003267 0.000383 0.0003946 0.001148 0.0004707 0.001376 0.0005707 0.001678 0.000616 0.001926 0.000646 0.002157	1.2 1.2 1.2 1.2 1.2
0.0003946 0.001148 0.0004707 0.001376 0.0005707 0.001678 0.000616 0.001926 0.000646 0.002157	1.2 1.2 1.2 1.2
0.0004707 0.0005707 0.0005707 0.000616 0.001926 0.000646 0.002157	1.2 1.2 1
0.0005707 0.001678 0.000616 0.001926 0.000646 0.002157	1.2
0.000616 0.001926 0.000646 0.002157	1
0.000646 0.002157	
0.000676 0.002399	1
0.0007213 0.002884	1
0.0007813 0.003551	1
0.0008413 0.004288	1
0.000872 0.004971	1
0.000902 0.0057	1
0.000932 0.00648	1
0.0009627 0.00734	1
0.0009933 0.008978	1
0.001055 0.01098	1
0.001085 0.01289	1
). (0.000902 0.0057 0.000932 0.00648 0009627 0.00734 0009933 0.008978 0.001055 0.01098



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

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

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

 _
-
-
•

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

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

Levels	Height	Elevation	Visible	Add Level
1	4.000000	4.000000	✓	, 100 20701
2	3.000000	7.000000	✓	Delete Level
3	3.000000	10.000000	 Image: A start of the start of	Detect Levels Automaticaly
				Base Elevation 0.0000
				Base Elevation 0.0000

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



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

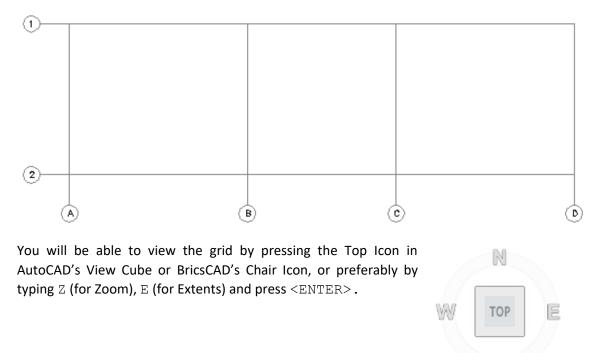
Placement			
 Horizontal 	◯ Sidelong		
Spacing			
6.0000 5.0000	Distance: 6.0000		
6.0000	Add		

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

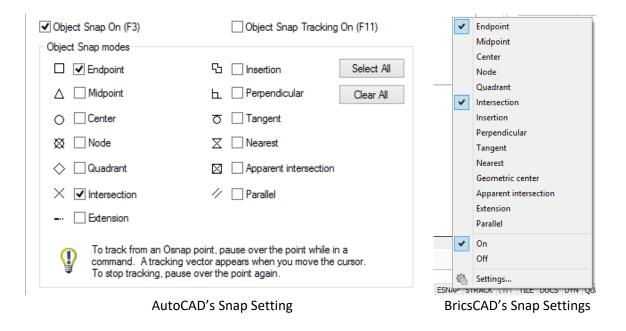
By pressing OK you are prompted (command prompt) to enter the Insert Point for the grid. Type 0, 0, 0 and press <ENTER>.

WCS 🗢

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



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

Cross Section	
Table Section:	
HE320B HEB European	¥
or Member Dimensions:	
<select shape=""></select>	~
or Same as Member:	
<select existing="" member=""></select>	~
Material Steel	v
Releases Fix-Fix V	Beta (o) 0 ✓
Place Member	(s) >>

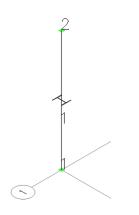
Select *HEB320B* as the cross section for the columns. Make sure that Material is set to *Steel*, Releases to *Fix-Fix* and the Beta angle is *0*. There is no need to close this dialog manually.

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

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

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

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



Note: Each time you create a member the orientation of the cross section will appear in the middle of the element, unless you clear it with command "Clear" (see 2.6.84).

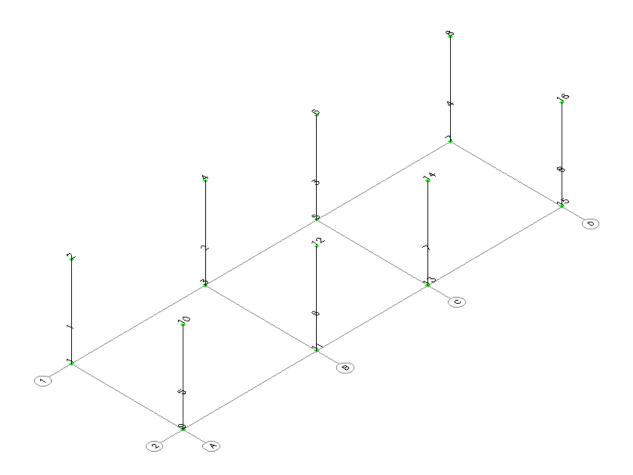
Step #8. You can easily change to an isometric view of the structure by pressing the small house icon in AutoCAD's View Cube or the small arrow "Top Front Left" of the following image in BricsCAD's View Chair. As you can see in the isometric view below, column members 1 to 8 were created together with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





AutoCAD's View Cube for Isometric View

BricsCAD's View Cube for Isometric View





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

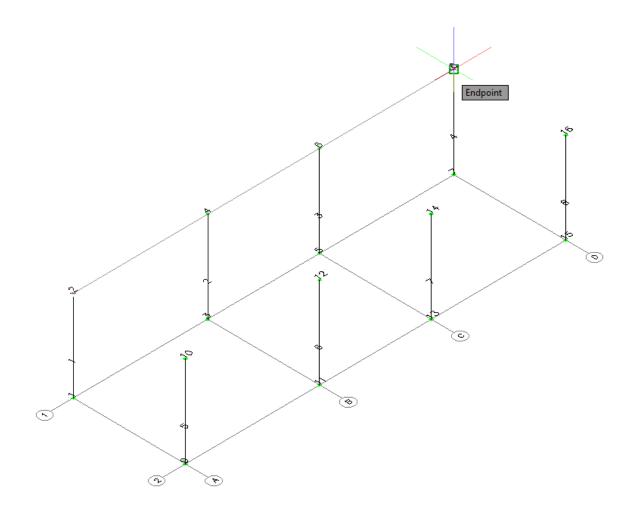
Cross Section	
Table Section:	
IPE330 IPE European	¥
or Member Dimensions:	
<select shape=""></select>	~
or Same as Member:	
<select existing="" member=""></select>	~
Material	
Steel	v
Releases	Beta (o)
Fix-Fix 🗸	90 🗸
 Split Intersecting Members Split Ending Members 	Physical Member
Place Member(s	s) >>

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

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

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

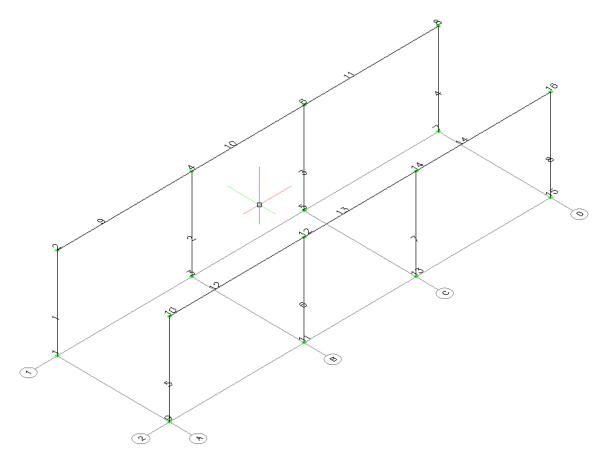
Then you have to define the *Ending Point* (x,y,z) so click on joint 8, as shown in the picture below.

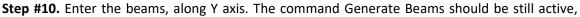


Members 9, 10 and 11 will be created.

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

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

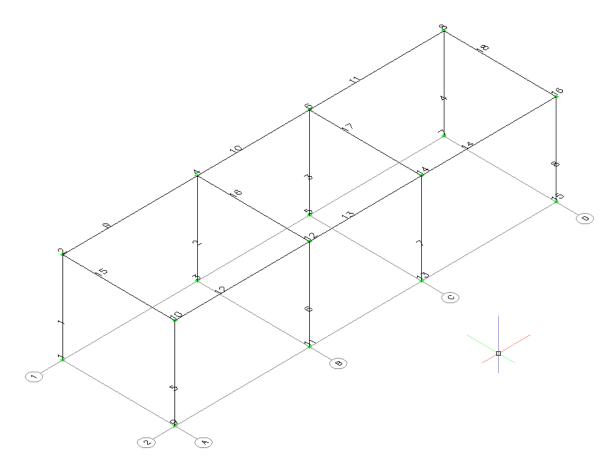




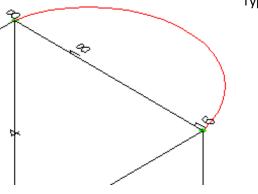
else you can call it again by clicking again on the icon Generate. Keep the same settings at the Place Member Form, as in the previous step, regarding the cross section and Beta angle, but do NOT click on *Split Intersecting Members*. Press the "Place Member(s) >>" button.

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 ^{Curve} and when the prompt message *Select Curve (Line or Arc)* appears, click on the Arc that you have created in the previous step.

Select Mesh Properties ×
Generate
Members Elements
Material Steel 🗸 …
Element Attributes
Type BPHQ V Thickness 0.20
Member Attributes
Type FRAME V Beta 90
Section IPE330 IPE European V
Spacing U Direction
● Uniform 8 🗸
🔾 Variable 🦳
O Defined by Line/Curve

The Select Mesh Properties form appears where you enter:

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

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

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

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

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

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

Home Insert Anno	tate Parame	tric View	Manage	Output	Plug-ins	Online	Express Tools	GTS CAD Mo
Line Polyline Circle	Arc	+‡+ Move S Copy	Mirror	Fillet	- 60			
Draw ▼ [-] [Custom View] [2D Wi	reframe]		Modify			 	GRID_CIRCLE	s ^

Step #14. Turn OFF labeling:

Visible Labels	
Joints	
Members	
2D Elements	
✓ 3D Elements	

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

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

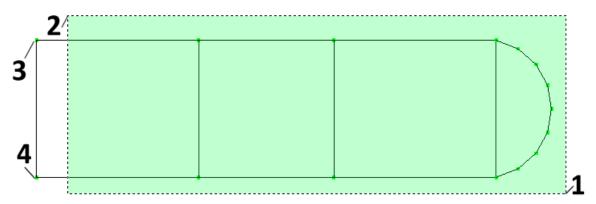
Note: You can also delete or hide the Arc line as it is no longer needed. It can be deleted by selecting it with the mouse, and then pressing the keyboard button <Delete>. Be carefull not to select any members or joints, but only the Arc line.

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

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

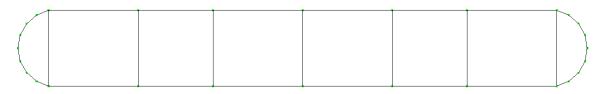
NOTE: This option is very useful when mirroring structures having shell finite elements and the mirror line lies along the element's XY plane. If you choose to maintain the incidence order, then after the mirroring the local Z axis of the source and copied finite elements have the same orientation.

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



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

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



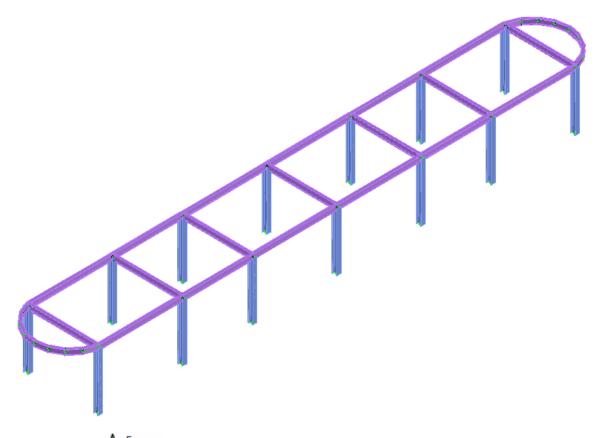
Step #16. Switch to 3D View: Press the house (AutoCAD) or chair (BricsCAD) icon to change the

view to Isometric, and type Z and E (Zoom, Extents). Click on the icon ^{O Colors} to set different colors for each profile.

Sec	ctions Groups						
	Categories						
	Sections	Color	Visible				
	HE320B	161	 				
	IPE330	191	 				
	IPE120	1	 				
	60x60x5	50	✓				

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

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

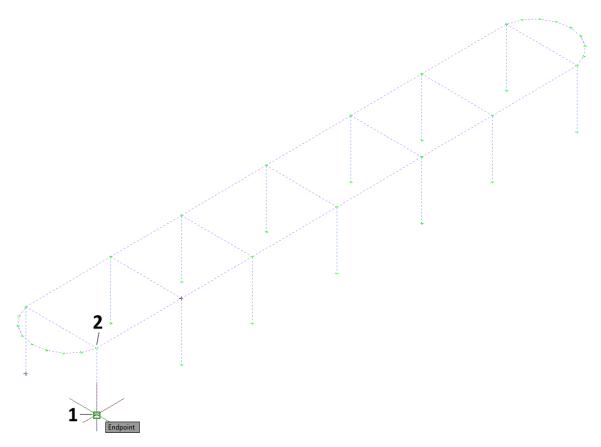


Press the icon \bigwedge Frame to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

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

3.5. Create the 2nd floor

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



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

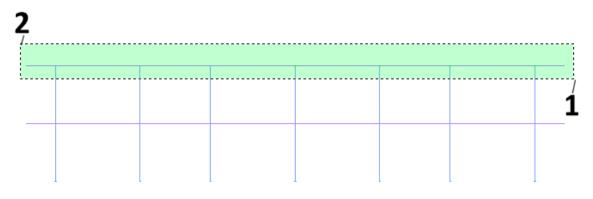
In order to *Enter second point* : click at the top of the same column such as point 2 of the picture and then press ESC in order to terminate the copy function.

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

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

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

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



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

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

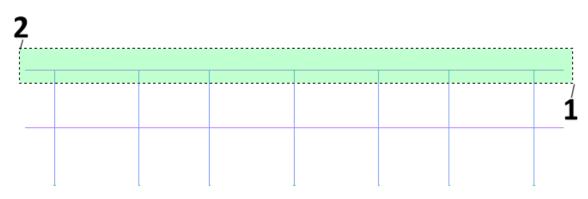
Now the height of the 2nd floor is correct and equal to 3.00 meters.

Note that you can reach the same result by switching to the 2^{nd} Level using the icon \blacktriangle Higher Level and start entering the columns one by one as you did in the 1^{st} level. Columns

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

3.6. Create the 3rd floor

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

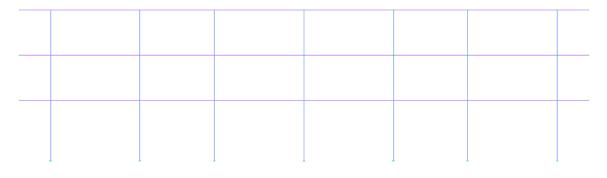


Since this is a crossing window, the columns are automatically selected. You will get a notification about the selected entities and press <ENTER>.

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

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

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



Step #21. Assign Level Properties:

Options				
Add Level				
Delete Level				
Detect Levels Automaticaly				
Merge Levels				
Base Elevation 0.0000				
Z Vertical Axis (else Y)				
Update Levels for All Entities				

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

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

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

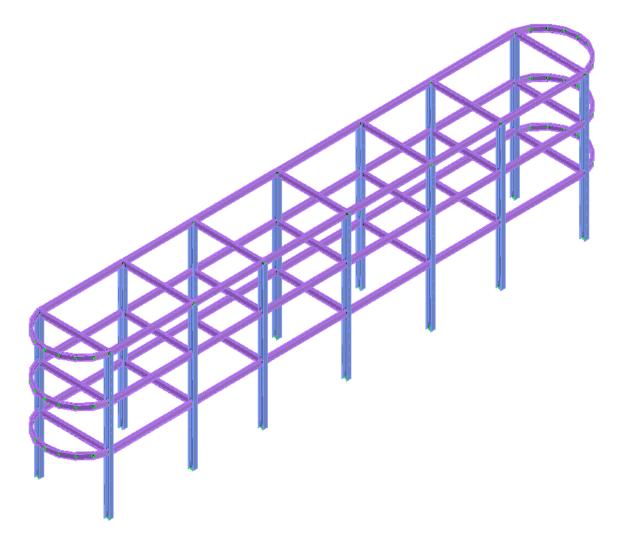
You can switch between the levels of the model using the

▲ Higher Level and ▼ Lower Level icons. The current level appears in the top caption of AutoCAD's screen.

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

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

Press the icon not 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 A Frame to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

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

Ħ

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

Make sure that "Level :1" appears at the top of CAD window, else use the \triangleleft Higher Level and \bigtriangledown Lower Level icons to move to Level 1.

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

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

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

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

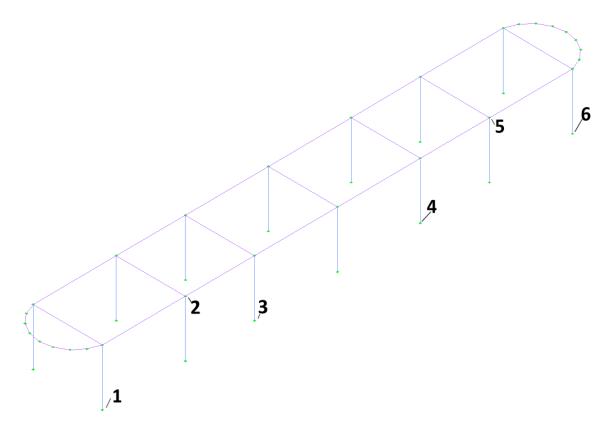
Click on joint located at Point 2 and the first bracing member is created.

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

Cross Section				
Table Section:				
60x60x5 BSEQANGL European V				
or Member Dimensions:				
<select shape=""></select>	~			
or Same as Member:				
<select existing="" member=""></select>	*			
Material	¥			
Releases Fix-Fix ✓	Beta (o) 0 v			
 Split Intersecting Members Split Ending Members 	Physical Member			
Place Member(s) >>				

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

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



Press ESC to terminate the Generate Beam command.

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

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

Member Properties [Multiple Selection]			×			
Model Section Properties Member Loads	Member Temperature Loads Member Distrortions					
General	Section Properties	Releases & Elastic Connection spring valu	les			
Name :	Section : <select section=""> ····</select>	Quick Selection : Fix-Fix	\sim			
Level :	Ax : Ix :	Start Spring End Sprin	ng			
	Ay : Iy :	Fx Fx				
Type - Incidences	Az : Iz :	Fy Fy				
Type : SPACE TRUSS V	Sy: Ey:	Fz Fz				
<select type=""> Start SPACE FRAME</select>	Sz : Ez :	Mx Mx				
SPACE TRUSS End :	Yd: Yc:	MyMy				
Beta Angle :	Zd: Zc:	Mz Mz				
Physical Member	Shape Code :	End Sizes OR Member Eccentricities (Off	sets)			
Groups		Start End				
Inactive Remove	Material Properties Material : <select material=""> V</select>	Sizes :				
Groups that Member belongs						
	E : Density	X: X:				
	G: CTE:	Y: Y:				
	Poisson	Z: Z:				
OK Cancel Apply Help						

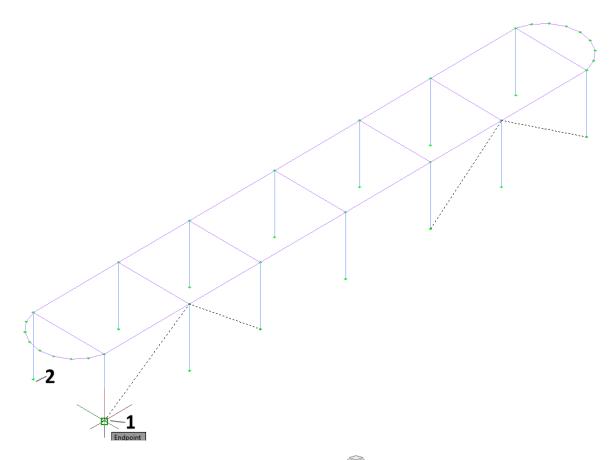
This modification applies to all selected members.

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

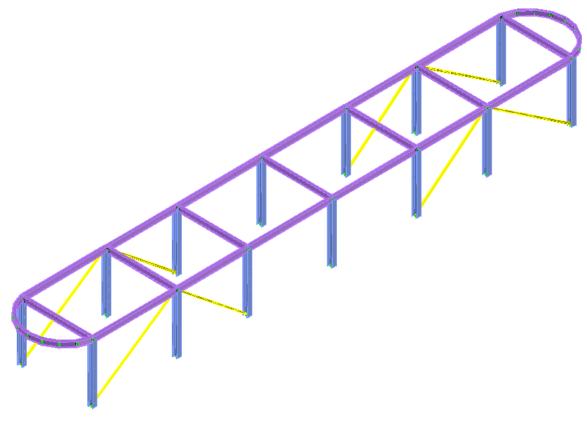
In order to *Enter base point:* click on the Joint at Point 1 of the following image.

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

Press ESC to terminate the COPY command.



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



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

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

Click on the icon \clubsuit 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 \bigcirc Higher Level icon move to level 3. The current level is displayed at the top caption of AutoCAD/BricsCAD:

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

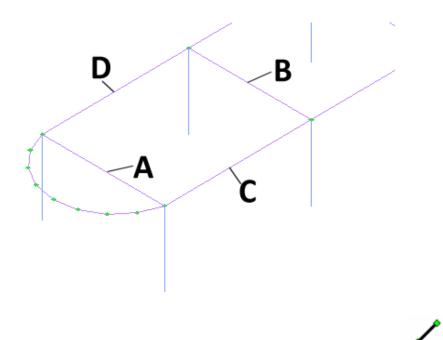
Click on the icon Split and click on the members A and B as shown at the following image, and then press <ENTER>.

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

/

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

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



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

Cross Section	
Table Section:	
IPE120 IPE European	¥
or Member Dimensions:	
<select shape=""></select>	~
or Same as Member:	
<select existing="" member=""></select>	~
Material Steel	¥
Releases	Beta (o)
Fix-Fix ¥	90 🗸
✓ Split Intersecting Members ○ Split Ending Members	Physical Member
Place Member(s)	>>

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

Press the "Place Member >>" button.

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

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

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

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

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

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

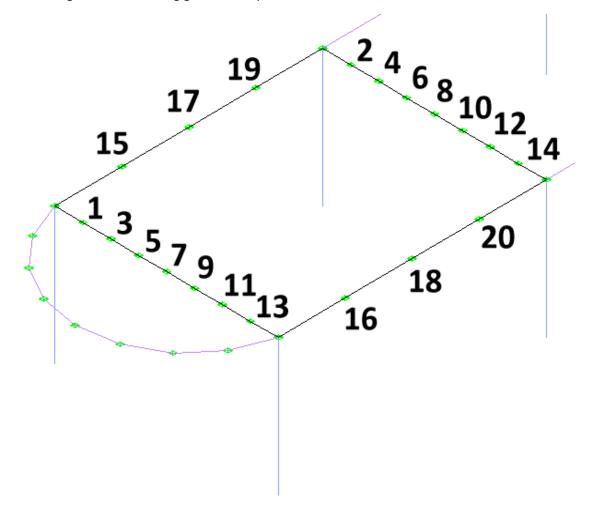
Continue by clicking on joints at points 13 and 14 and another girder member is generated.

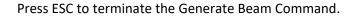
Make sure that the option "Split Intersecting Members" is ON, so that common joints will be created along the previously created X-direction girders. Click on the joint located at point 15 and then click on joint at point 16 and the girder member is generated. Existing girders are split.

Repeat the procedure:

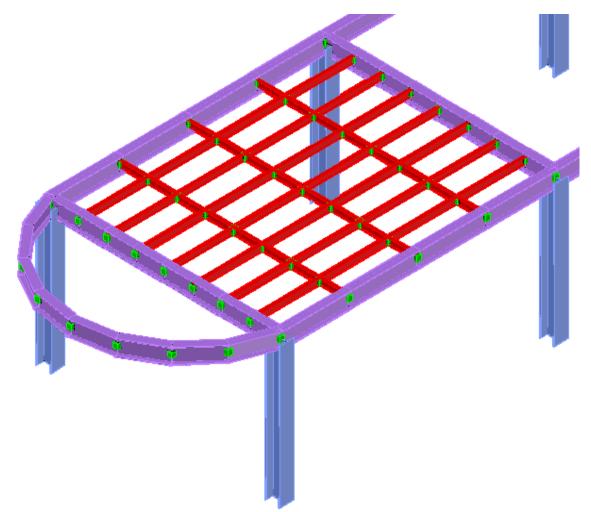
Click on the joint located at point 17 and then click on the joint at point 18 and the girder member is generated. Existing girders are split.

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



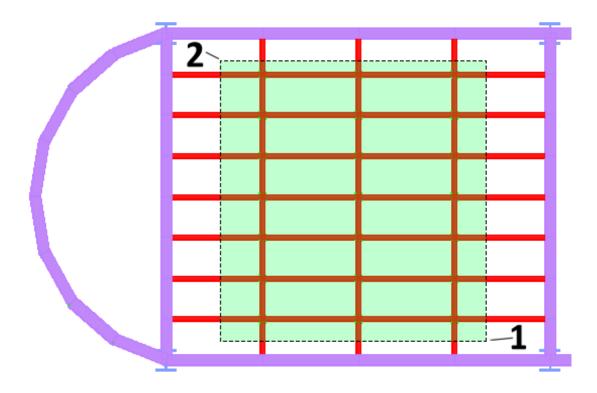


Step #29. Add eccentricities to the Girders: Press the icon is to display the 3D solid view and then the icon Colors and OK. The model looks like:

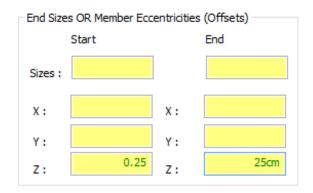


Click on the Top Icon on AutoCAD's View Cube or BricsCAD's View Chair to switch to the top view in order to be able to select girder members easily.

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



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

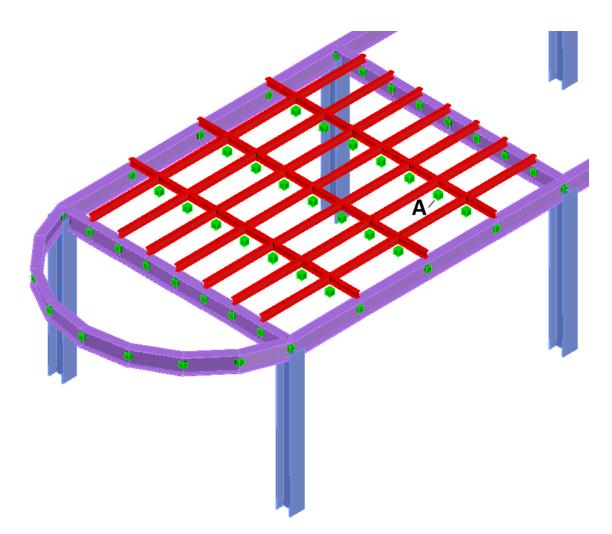


Note: Some Edit Boxes appear in yellow background and green fonts, like Eccentricities. You can use mixed units in the yellow edit boxes. To understand this feature enter 0.25 for the Z Starting Eccentricity and 25cm as the Z Ending Eccentricity.

Press OK.

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

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

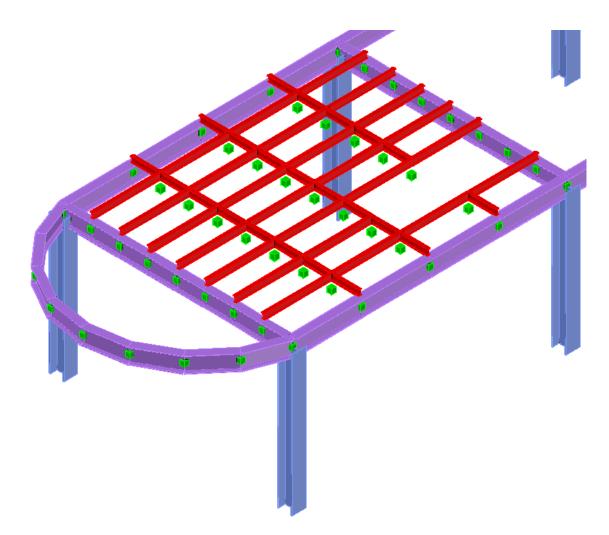


3.9. Create an opening

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

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

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



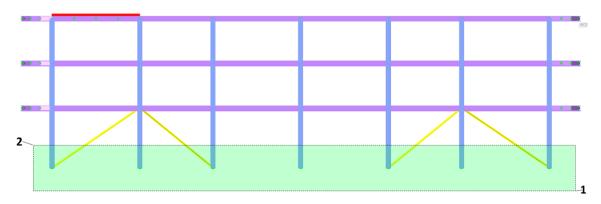
3.10. Create Supports

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

Make the entire structure visible by clicking at the icon $\widehat{\mathfrak{S}}^{All Levels ON}$ and press Z and E (Zoom Extents).

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

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



-Restraints & Spring va	lues
Quick Selection :	Pin 🗸
Restraint Spring	Restraint Spring
Fx	Mx
√ Fy	My
√ Fz	Mz

The Joint Properties [Multiple Selection] form appears.

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

Press OK.

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

3.11. Check the model

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

click on the icon ^{Joints Duplicates}, under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area.

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

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

	Merge	Joints	
List			Select All
Joint	Duplicate	Merge	
8	25		Unselect All
16	17	✓	
			ОК
			Cancel

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

Step #33. Check for floating joints: In order to check for joints not connected to the model, click

on the icon ^{Joints Floatings}, under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area. If your model was created as described so far, you should get a notification *O floating joints found*.

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

3.12. Define Groups

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

	_
	_
	_
•	_

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

Group Proper	ties		Add Group
ID	Name	Physical	
1	Columns		Delete Group
2	Beams		
3	Girders		ОК
4	Bracing		

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

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

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

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

Press OK to close the form.

Step #35. Add Columns to their Group.

Click on the icon <mark>О</mark> Colors	Categories		
and make only profiles	Sections	Color	Visible
HE320B visible by	HE320B	161	 Image: A start of the start of
unckecking all others.	IPE330	191	
	IPE120	1	
Dross OV	60x60x5	50	

Press OK.

	Select Group	×
Group		
Columns		v

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

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

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

Groups		
Inactive		Remove
Groups that M	1ember b	pelongs
Columns		

Step #36. Add Beams to their Group:

Click on the icon ^{Colors} and make only profiles IPE330 visible by unckecking all others.

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

Press OK.

Categories

Select Group ×	Click on the icon + Members and make
Group	sure that the Group Beams is selected as the active group.
Beams v	
	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 O Colors Categories Sections Color Visible and make only profiles 161 IPE120 visible by HE320B unckecking all others. 191 IPE330 ~ IPE120 50 60x60x5

Press OK.

	Select Group	×
Group		
Girders		v

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

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

Step #38. Add Bracing to their Group:

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

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

Press OK.

	Select Group	x
Group		
Bracing		¥

Click on the icon Herein 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 ^{IIII} Self Weight and the Self Weight dialog appears. Enter:

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

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

Self Weight or Dead Loads		x		
Load Information	on			
Name : Description :	SW 🗸			
Create Ne	ew Save / M	Modify	Delete	
Loads applie	ed parallel to this Glo	bal Axis		
○ Nega	ative Y	○ Positiv	еY	
Nega	ative Z	○ Positiv	e Z	
○ Nega	ative X	○ Positiv	e X	
Factor :		✓ Include	e Finite Elements	
		ОК	Cancel	

Step #40. Define Load Cases: Click on the icon ^I Load Cases</sup> and the Load dialog appears.

Enter:

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

and press Create New.

Load Case				×
Load Case Information				
Load ID (up to 8 chars) :	LL	Design variable :	L I V	Create New
Description :	Live Load			Save / Modify

Enter:

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

and press Create New.

Load Case				×
Load Case Information				
Load ID (up to 8 chars) :	PL	Design variable :	~	Create New
Description :	Point Load			Save / Modify
Load ID List :	LL			Delete

Press Exit to close the dialog.

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

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

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

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

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

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

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

Member Properties [Multiple Selection]		×
Model Section Properties Member Loads	Member Temperature Loads Member Distrortions	
Applied Load Cases	Loading Load Case : $\[Ll \] \] Create New \] Save / Modify \] Loads in this Load Case : \[\] \] Delete \]Load Distribution\[\] Concentrated \] \] \] \] \] \] \] \] \] \] \] \] \] $	
LL \$ Live Load \$[0] PL \$ Point Load \$[0]	Load Type and Direction Loading Magnitude Loading Location from Start Force X Local Y Moment Z Global Loading Magnitude Loading Location from Start Y Y O Absolute L2: 1.0 Absolute L2: I.0 Interview Interview Intervi	
	OK Cancel Apply Help	,

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

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

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

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

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

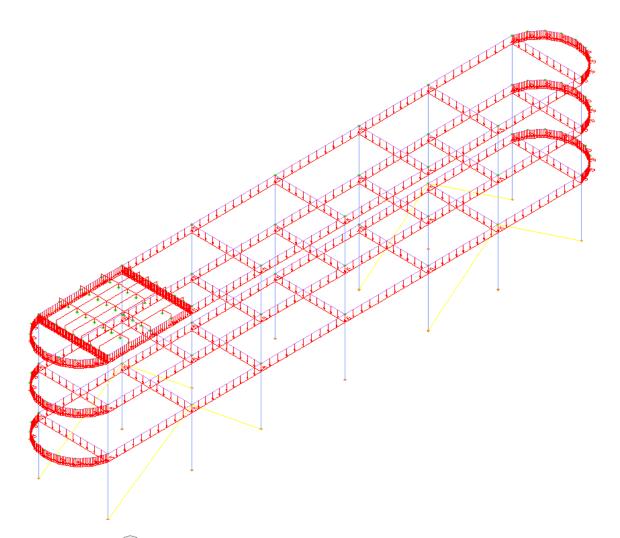
On the Menu Bar, click on GTS Display>Display>Member Loads and the Display Loads form appears:

Display Loads	×
Load Case LL \$ Live Load	~
Display Loads Applied To	
Joints Members	Elements
Display Options	
Scale Factor Concentrated (pts)	30
Scale Factor Distributed (pts) :	30
Arrowhead Size (pts) :	10
Font Size (pts) :	10
Show Clear	Close

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

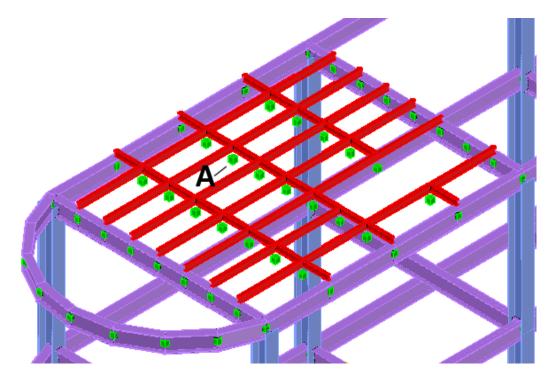
Press Show and the loading arrows are displayed.

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



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

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

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

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

Joint Properties		×
Model Joint Generalized Loads		
Applied Load Cases	Loading Load Case : PL Create New Description : Point Load Save / Modify Loads in this Load Case : Applied Load Values	
	Force X : Force Y : Force Z : -3 Moment X : Moment Y : Moment Z :	
Empty Load Cases LL \$ Live Load \$[0] PL \$ Point Load \$[0]	Tran X : Tran Y : Tran Z : Rot X : Rot Y : Rot Z :	
	OK Cancel Apply Help	

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

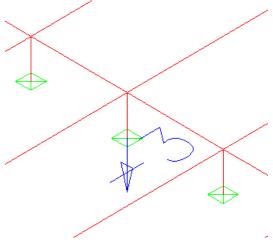
On the Menu Bar, click on GTS Display>Jont Loads and the Display Loads form appears:

Display Loads	x
Load Case	
PL	~
Display Loads Applied To	
Joints Members	Elements
Display Options	
Scale Factor Concentrated (pts)	30
Scale Factor Distributed (pts) :	30
Arrowhead Size (pts) :	10
Font Size (pts) :	10
Show Clear	Close

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 is to display the 3D solid view.

Step #45. Define Area Load for Level 1: An area load equal to 1.0kN/m² along the vertical direction will be applied to whole level. Switch to Level 1 by clicking on the icon **V** Lower Level untill "Level 1" is displayed.

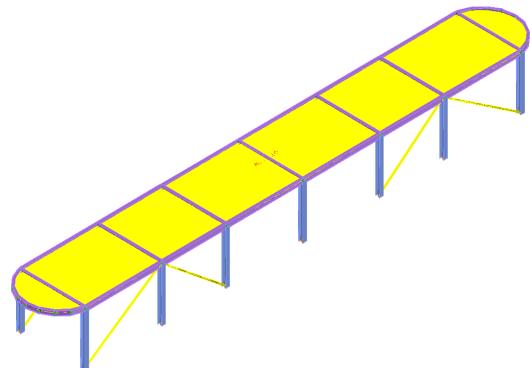
Click on the icon Area Load in the Loads panel of the ribbon.

	Area Load ×
Generate	
Name :	AL1 V
Description :	Area Load Level 1
Create Nev	v Save / Modify Delete
Load - Directior	1
Load Value :	1.0
	on Perpendicular to the Loading Plane :
○x	⊖Y ®z
Plane Toleran	ce : 0.0508
Elevation	
Plane Perpend	dicular at :
Value (coo	rdinate) 4.0
) Joint	~
Distribution	
Two way	⊙x
One way	ΟY
	Custom x 0.0000 y 0.0000
	Custom X

Type:

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

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



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

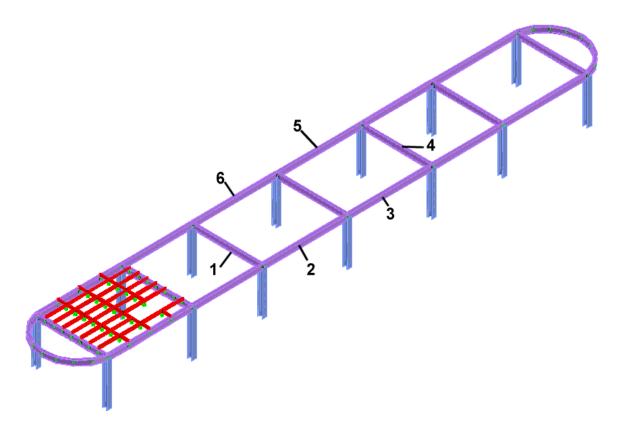
Step #46. Define Area Load for Level 3: An area load equal to 1.0kN/m² along the vertical direction will be applied only to the two middle openings. Switch to Level 3 by clicking on the

icon Higher Level untill "Level 3" is displayed.

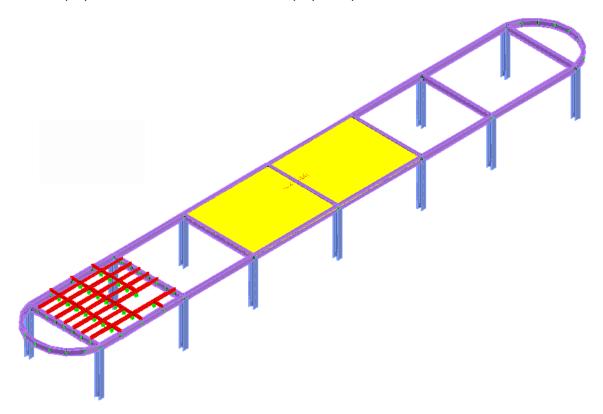
Click on the icon 📕 Area Load in the Loads pa	anel of the ribbon.
Area Load ×	Туре:
Generate Name : AL3 v Description : Area Load Level 3 Create New Save / Modify Delete	 AL3 as Area La 1.0 as Z as Gli 10.0 a
Load - Direction Load Value : 1.0000	level 3 - <i>Two w</i>
Global Direction Perpendicular to the Loading Plane : X Y Image: Z Plane Tolerance : 0.0508	Press "Define (prompted to s the outline reg
Elevation Plane Perpendicular at : Value (coordinate) Joint	the 6 member middle openir below.
Distribution Two way X One way Y Custom X 0.0000 Y 0.0000	
Advanced Features	
Define Outline Region >> Reset	
Exclude Area >> Reset Ignore Members >> Reset	
Display >> Clear OK Cancel	

- -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 \underline{OK} to store the area load AL3.

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

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

New Form Load or Load Combination	×
Load Information Name : Design variable :	Type Load Combination Form Load
Combine SW (Self Weight) LL (Live Load) PL (Point Load) AL1 (Area Load Level 1) AL3 (Area Load Level 3) CB1 (Load Combination 1) 'SW' 1.35000 'LL' 1.50000 'PL' 1.50000	ADD >> Factor : 1.5 Delete Item
All Formed Loads or Combinations CB1 (Load Combination 1) 'SW' 1.35000 'LL' 1.50000 'PL' 1.50000 Edit Delete	Done

3.14. GT STRUDL Input File

Step #48. Create GTI: Save your model as "Tutorial_Example_1.dwg" and click on the icon GTI 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	F.STRUDL In	put File	×
Export to GTI				
Create GTI File :	ocumentation\Versio	on2019\Example	e01\Tutorial_Examp	le_1.gti
• Export Whole	Model		Portion of the Mode prompted for Selec	
Perform Stiffn	ess Analysis			
Append Other GTI F	Files/Macros			
				+
				Up
Copy Command	ds from GTI Files/Mac	ros (not CINPU	Г)	Down
Contraction of the	Dec d Dec dia			
- Create Commands t				
✓ Read Joint Dis	placements			
Read Member	Forces			
Read Section F	Forces	Nur	nber of Sections:	10
Read Section [Displacements	Nur	nber of Sections:	10
Read Finite Ele	ment Results			
Read Code Ch	eck Results			
			ОК	Cancel

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



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

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

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

PARAMETERS

CODE EC3 ALL MEMBERS

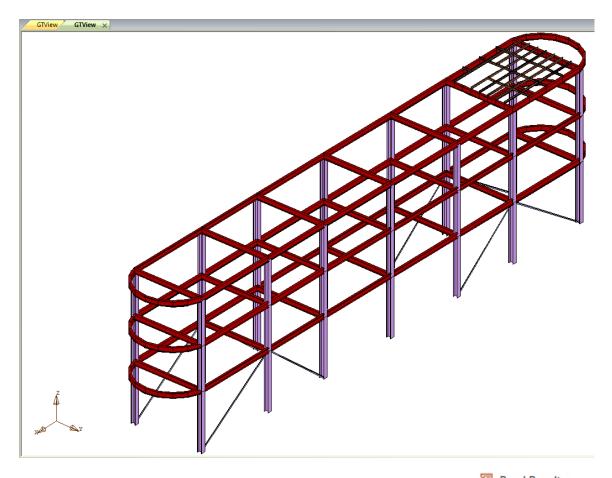
CHECK ALL MEMBERS AS BEAM

DBX BINARY 'Tutorial Example 1.25' REPLACE

WRITE REPLACE CODE 'Tutorial Example 1.25' MEMBERS EXISTING

The result of the CHECK ALL MEMBERS command shown above is that members 148149150151152153154155FAILED CODE CHECKS

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



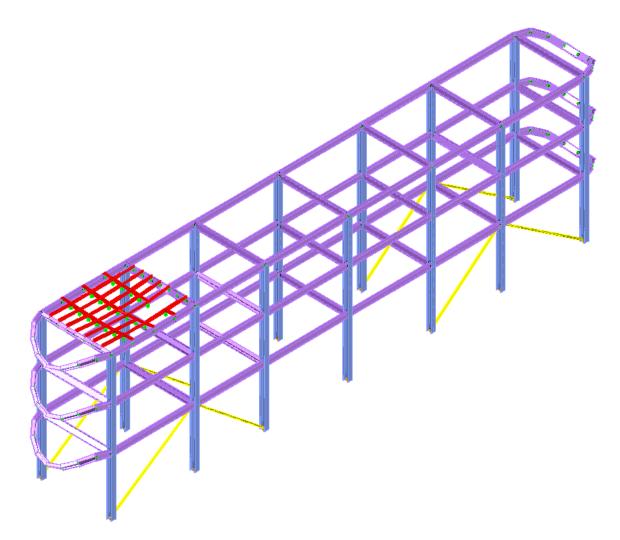
Step #51. Read Results from GT STRUDL: In CAD Modeler, click on the icon Read Results and the Read GT.STRUDL Results dialog appears. Check all options except "Read Finite Element Results" as shown below and press OK.

Read GT.STRUDL Results		×	
Read DBX Results			
GTI Directory : D:\Examples\GTSTRUDL\Example01			
Read Member Forces			
Read Section Forces	Number of Sections:	10	
Read Section Displacements	Number of Sections:	10	
Read Finite Element Results			
Type of element result :	<all></all>	\sim	
Surface :	<all></all>	\sim	
Read Code Check Results			
GTI Commands			
Corresponding GT STRUDL commands:			
DBX BINARY WRITE REPLACE JOINT RESULTS JOINTS EXISTING WRITE REPLACE MEMBER RESULTS MEMBERS EXISTING WRITE REPLACE SECTION FORCES NS 10 AUTOMATIC MEMBERS EXISTING WRITE REPLACE SECTION DISPLACEMENTS GLOBAL NS 10 MEMBERS EXISTING WRITE REPLACE CODE MEMBERS EXISTING			
	ОК	Cancel	

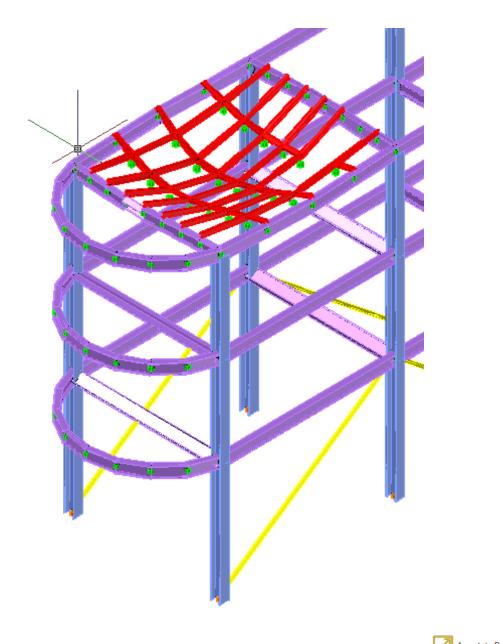
Note: You should get an information message "Results Loaded Successfully"

3.15. Display Results

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



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



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

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

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

Select: - SW as Load Case

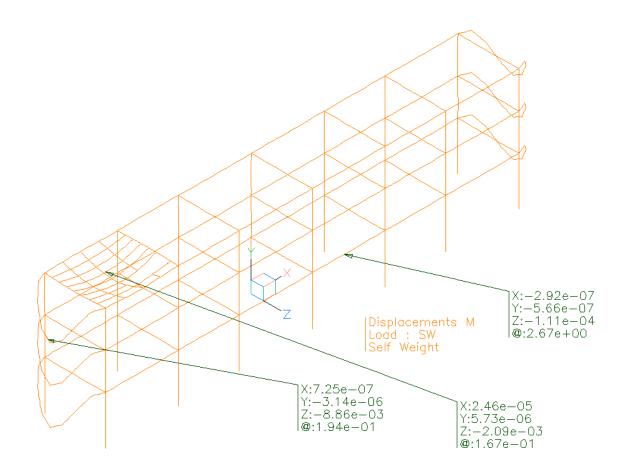
Displacements			
Load Case / Load Combination			
SW \$ Self Weight	~		
Display Options Scale Factor (Values) : Font Size (pt) : Annotation Format: Decimal Places :	0.100 10.00 Exponential V 2 V		
✓ Hide Model			
Display :	Display >>		
Annotate >	Legend >		
Animation Options Frames : Animation Speed % :	7 1 ×		
Generate Animation Frames	Animate >>		
Clear	Close		

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

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

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

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



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

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

Step #54. Display Member Diagrams: Click on ^{w Diagrams} (ribbon tab "GTS Display").

Member Diagrams ×			
Load Case / Load Combination			
SW \$ Self Weight	~		
Envelope			
Values			
MZ Moment	~		
Display Options			
Scale Factor (Values)	0.100		
Font Size (pt)	10.00		
Annotation Format:	Decimal 🗸		
Decimal Places :	2 🗸		
Label Max & Min Positive Sign			
Display >>			
Annotate >	Legend >		
Clear	Close		

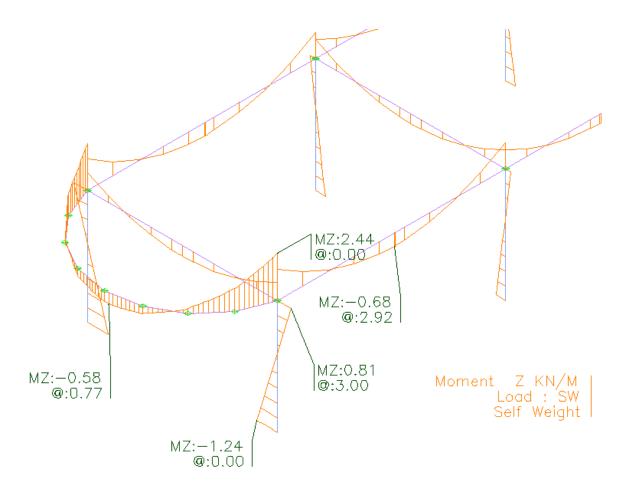
Select:

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

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

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

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



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

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

Code Check Results		
Display Text		
○ None		
Actual/Allowable Stress Ratios		
O Controlling Stress Code Provision		
O Actual/Allowable KL/r Ratios		
O Controling KL/r Code Provision		
Values		
O Greater Than	0.00	
O Less Than	0.00	
Display Options		
Font Size (pt)	4	
Display >>		
Legend >		
Clear	Close	

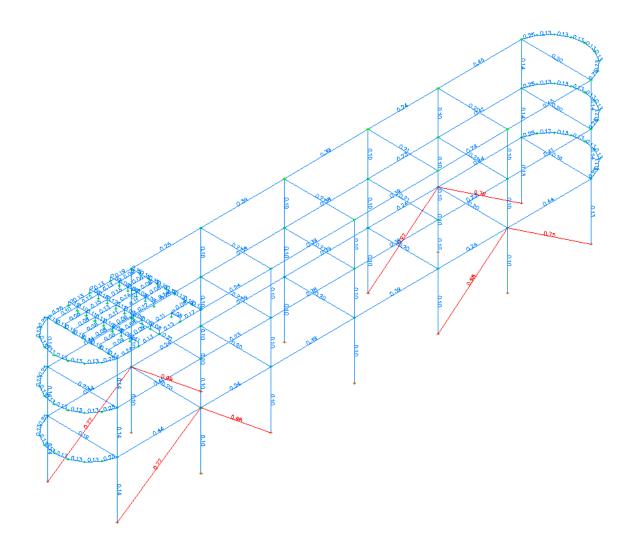
.

Select:

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

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

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



3.16. Results Datasheets

Step #56. Except from the graphical display of results there is an option to view them in datasheets from the item "Results Datasheets" of the menu/ribbon tab "GTS Display".

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

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

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

i <u>le E</u> dit <u>C</u> olumns <u>Fi</u> lter <u>S</u> ort Units <u>H</u> elp						
oint	Load	Force X	Force Y	Force Z	Moment X	Mome
1	SW	0.000	-0.086	24.790	-0.000	
3	SW	0.206	-0.085	24.199	-0.000	
5	SW	0.020	-0.085	23.735	-0.000	
7	SW	-0.162	-0.177	25.863	0.000	
9	SW	0.000	0.083	24.777	0.000	
11	SW	0.267	0.085	24.261	0.000	
13	SW	0.020	0.085	23.717	-0.000	
15	SW	-0.103	0.182	25.788	0.000	
26	SW	-0.316	-0.103	24.309	0.000	
28	SW	-0.015	-0.118	26.112	0.000	
30	SW	0.148	-0.140	27.442	-0.000	
32	SW	-0.247	0.067	24.146	-0.000	
34	SW	-0.014	0.064	25.249	-0.000	
36	SW	0.195	0.227	27.381	-0.000	
1	LL	0.000	-0.881	128.774	-0.000	
3	LL	1.222	-0.881	120.588	-0.000	
5	LL	0.200	-0.881	116.237	-0.000	
7	LL	-0.476	-1.865	137.305	-0.000	
9	LL	0.000	0.881	128.774	0.000	
11	LL	1.223	0.881	120.589	-0.000	
13	LL	0.200	0.881	116.237	0.000	
15	LL	-0.475	1.865	137.303	0.000	

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

le Edit (Column	s Filter So	ort Not Checked	l Help						
Member	P/F	Load	Section Loc	Crit Ratio	Crit Prov	Stress Ratio	Stress Prov	KL/r Ratio	KL/r Prov	Properti
148	FAIL	CB1	7.211	3.395	KL/r	0.766	5.5.1	3.395	KL/r	60x60x5
149	FAIL	CB1	6.403	3.015	KL/r	0.981	5.5.1	3.015	KL/r	60x60x5
150	FAIL	CB1	6.403	3.015	KL/r	0.977	5.5.1	3.015	KL/r	60x60x5
151	FAIL	CB1	7.211	3.395	KL/r	0.751	5.5.1	3.395	KL/r	60x60x5
152	FAIL	CB1	7.211	3.395	KL/r	0.768	5.5.1	3.395	KL/r	60x60x5
153	FAIL	CB1	6.403	3.015	KL/r	0.992	5.5.1	3.015	KL/r	60x60x5
154	FAIL	CB1	6.403	3.015	KL/r	0.973	5.5.1	3.015	KL/r	60x60x5
155	FAIL	CB1	7.211	3.395	KL/r	0.755	5.5.1	3.395	KL/r	60x60x5
1	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
2	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
3	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
4	Pass	CB1	4.000	0.293	KL/r	0.134	5.5.4(2)	0.293	KL/r	HE320B
5	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
6	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
7	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
8	Pass	CB1	4.000	0.293	KL/r	0.134	5.5.4(2)	0.293	KL/r	HE320B
9	Pass	CB1	0.000	0.939	KL/r	0.391	5.5.4(2)	0.939	KL/r	IPE330
10	Pass	CB1	0.000	0.782	KL/r	0.243	5.5.4(2)	0.782	KL/r	IPE330
11	Pass	CB1	6.000	0.939	KL/r	0.439	5.5.4(2)	0.939	KL/r	IPE330
12	Pass	CB1	0.000	0.939	KL/r	0.391	5.5.4(2)	0.939	KL/r	IPE330
13	Pass	CB1	0.000	0.782	KL/r	0.243	5.5.4(2)	0.782	KL/r	IPE330
14	Pass	CB1	6.000	0.939	KL/r	0.439	5.5.4(2)	0.939	KL/r	IPE330
15	Pass	CB1	0.000	0.782	KL/r	0.205	5.5.2	0.782	KL/r	IPE330
16	Pass	CB1	0.000	0.782	KL/r	0.205	5.5.2	0.782	KL/r	IPE330
17	Pass	CB1	5.000	0.205	5.5.2	0.205	5.5.2	0.000	*******	IPE330
18	Pass	CB1	5.000	0.188	5.5.2	0.188	5.5.2	0.000	*******	IPE330
19	Pass	CB1	0.000	0.249	5.5.3	0.249	5.5.3	0.000	*******	IPE330
20	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	*******	IPE330
21	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	*******	IPE330
22	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	*******	IPE330
23	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	*******	IPE330
24	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	*******	IPE330
25	Pass	CB1	0.871	0.127	5.7.7	0.127	5.7.7	0.000	*******	IPE330
26	Pass	CB1	0.871	0.249	5.5.3	0.249	5.5.3	0.000	*******	IPE330
27	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
28	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
29	Pass	CB1	4.000	0.293	KL/r	0.135	5.5.4(2)	0.293	KL/r	HE320B
30	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B
31	Pass	CB1	4.000	0.293	KL/r	0.102	5.7.7	0.293	KL/r	HE320B

3.17. Report Builder



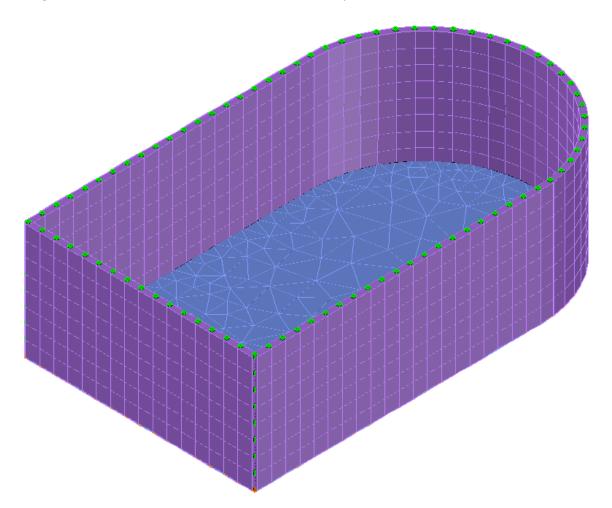
Step #57. Click on the icon Builder to generate the report of this structure by calling Report Builder. Report Builder is launched in a separate window. You can find more information on how creating your report in the GT STRUDL[®] Report Builder Getting Started Guide (Help Icon on the right top corner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

File Overall Report File File Insert H	Insert Remove eader and Footer Print Print Print Print Print	Joint Name: ALL Member Name: ALL Element Name ALL		 Loading: ALL Level: ALL Surface: ALL Filters	 Apply Filters to Checked Items	©.00 .0E Format Options Format	(?) Help Help	- □ × Style * @
	Load Data Summary of Loadings Length: M. Force: KN, Angle: DEG,	Temperature: DEGC , Tr	ine: SEC					
			Name SW LL PL AL1 AL3 CB1	Description Self Weight Live Load Point Load Area Load Level 1 Load Combination 1				
Count Average Element Results Average Element Results Member Results Graphs Origin of Steel Members								
Ready								CAP NUM SCRL

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.

Welco	me to GT STRL	DL									_ = >
	Open	Learn more about GT	STRUDL			Hide Details	~	\rightarrow	Ê	0	8
⊘ ∺≘	Help Options	GT Menu Perform structural analysis & design using a graphical use interface.		Command Mode Learn the STRUDL design language to create computer models for structural analysis and design.	Model Wizard Quickly model simple structures using templates.	Base Plate Wizard Analyze & design complex structural itsel base plates using a finite element analysis approach.		HEX	AGON		CADWorx® Analysis Sc
		Learn More Start a New Project	* CADModeler is not installed. Learn More	Learn More	Learn More	* Advanced license required Learn More	Introduc View ar	n informativ presented	IDL? STRUDL e webinar of on January 2	the nev	
		R		 	E Q		Now Av A limited	ailable: GT version of the on to the mo	STRUDL 201 a product that of st frequently us	an be us	ed as an
		GT Menu Working Directory	CAD Modeler	Command Mode	Model Wizard	Base Plate Wizard		AD-Based Mo bruary 20, 21	odeling for Stru 020	ctural Eng	<u>gineers</u>
			20-04-27_UTC-03-22-51\GT	5 2020 Shell\MO069_gti		Ē	Ci Si	onnecting De	roject Executions sign to Engine Piping Recorde	ering — U	niting
		Recent					M		ebinar: GT STF ay, with GT ST ar 13, 2018		
							S	ructural Worl	ebinar: GT STF kflow with Inter : October 18, 2	graph Sm	

4.3. Define the basic geometry of the model

Step #2. Define the correct Units by pressing the icon $\overset{\text{W Units}}{\text{ and select Meters (m) and KiloNewtons in the Units Form.}$

Units		×
Length	Force	Angles
O Inches (in)	O Pounds (lbs)	Degrees
O Feet (ft)	⊖ Kips	○ Radians
Meters (m)	◯ Tons	◯ Cycles
O Centimeters (cm)	◯ Kilograms	
O Millimeters (mm)	O Metric Tons	Time
Temperature	○ Newtons	 Seconds
○ Fahrenheit	KiloNewtons	◯ Minutes
Centigrade	OMegaNewtons	OHours
Scale non-structura (grids, structural line		OK Cancel

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

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

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

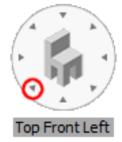
Step #4. Create a Line along the height of the tank: Switch to the isometric view of the structure by pressing the small house icon in AutoCAD's Cube or BricsCAD's Chair.



WCS 🗢

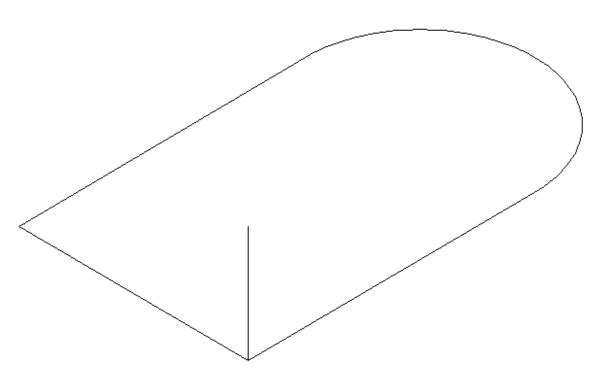
Type the following commands at the command prompt (each command is followed by an <ENTER>):

LINE
0,0,0
@0,0,4
<enter></enter>



The line shown at the picture below is created.

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



Create the bottom of the tank 4.4.

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

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

Select Mesh Properties ×					
Generate					
Material Concrete V					
Element Attributes					
Type SBHT6 ♥ Thickness 0.20					
Mesh Geometry					
External Boundary obj-563					
Boundary Maximum Edge Size 0.50					
✓ Do not split boundary more than Max					
Element Maximum Area 10.513274					
Mesh Quality High 🗸 🗸					
Internal Boundaries Internal Joints Add + Remove - Add Remove					
Spacing Extrude Direction					
● Uniform 4 V					
Variable Defined by Curve, Size: 3.242418					
Labeling More >>					
Preview Clear Create Close					

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

- Material : Concrete
- Type: *SBHT6*, meaning triangular elements having 6 degrees of freedom per node
- Thickness: 0.20
- Boundary Maximum 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.

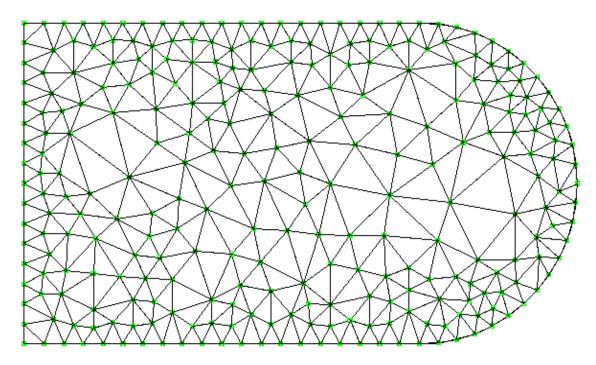
Step #6. Turn OFF labeling and view mesh:



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

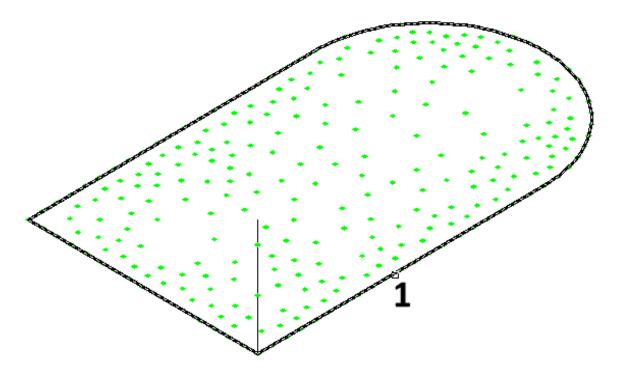
Now labeling is turned off and it is easier and faster to control the model.

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

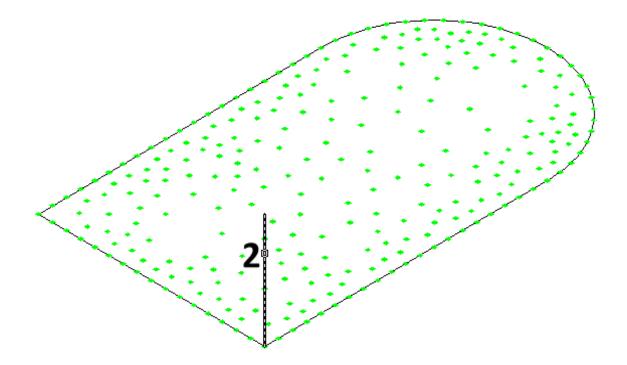


4.5. Create the walls of the tank

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 Button, located in Meshing at Ribbon Area (Note: the display of joints and elements previously created is automatically turned off to make selection of the polyline and extrude line easier). When the prompt message *Select Line, Arc, Circle or PolyLine to be Extruded* appears, click on the Polyline that you have created in a previous step, as shown in the following picture (Click #1).



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



Select Mesh Properties						
Generate						
Material Concrete V						
Element Attributes						
Type SBHQ6 Y Thickness 0.20						
Mesh Geometry						
External Boundary obj-218						
Boundary Maximum Edge Size 0.50						
✔ Do not split boundary more than Max						
Element Maximum Area 10.513274						
Mesh Quality Very Low 🗸						
Internal Boundaries Internal Joints						
Add + Remove -						
Multi+ Multi - Add Remove						
Spacing Extrude Direction						
● Uniform 8						
🔿 Variable 🛛 📖						
O Defined by Curve, Size: 3.242418						
Sweep Function						
Labeling More >>						
Preview Clear Create Close						

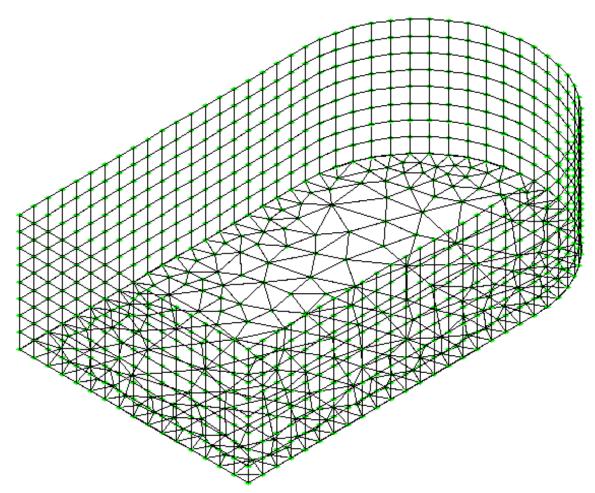
The Select Mesh Properties form appears where you have to enter:

- Material : Concrete
- Type: *SBHQ6*, meaning quad elements having 6 degrees of freedom per node
- Thickness: 0.2
- 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 ^{Joints Duplicates}, under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area.

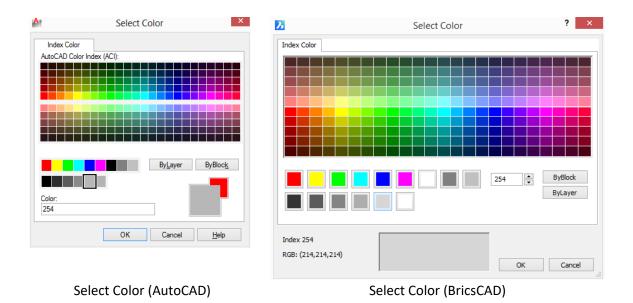
For the *Merge Tolerance* <0.001000>, just press <ENTER> to accept the default value.

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

st			Select All
Joint	Duplicate	Merge	<u>^</u>
1	281	Image: A start of the start	Unselect All
1	363		
2	282		ОК
3	283	✓	
4	284	✓	Cancel
5	285		
6	286		
7	287	✓	
8	288	✓	
9	289	✓	
10	290	✓	
11	291	✓	
12	292	✓	
13	293	✓	
14	294		
15	295		
16	296	✓	
17	297	✓	
18	298	✓	
19	299		
20	300		
21	301	Image: A start of the start	
22	302	Image: A start of the start	
23	303		
24	304		
25	305	Image: A start of the start	
26	306	Image: A start of the start	_

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

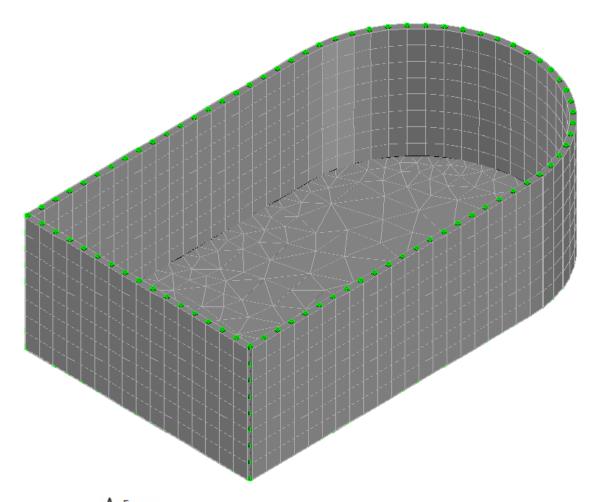
Step #9. Switch to 3D View: Press the house icon (AutoCAD cube) or Top Front Left (BricsCAD chair) to change the view to Isometric, and type Z and E (Zoom, Extents). Click on the icon Options to set a different color for 2D finite elements. When the Display Options form appears, click on the white button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



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

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

Display O	ptions ×
Visible Objects	Visible Labels
✓ Joints	Joints
✓ Members	Members
✓ 2D Elements	2D Elements
✓ 3D Elements	✓ 3D Elements
Label Settings - Font Sizes	Object Sizes
Joints :	Joint :
Members : 0.250000	Load Arrowhead (pts): 10.00000
2D Elements : 0.250000 3D Elements : 0.250000	Display Members / Elements
Annotation (pts) : 10.000000	Shrink Factor : 1.0 ¥
Annotation Format: Decimal 🗸	Do Not Display Thickness in 3D
Decimal Places : 2 V	Members As: Analytical V
Scale Factors	
Concentrated Load (pts) : 72.000000 Distributed Load (pts) : 72.000000	OK Cancel



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

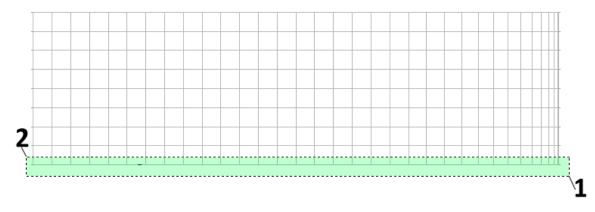
Step #10. Save your Model: In order to save your model, use AutoCAD's/BricsCAD's save command and store the DWG using any filename that you want.

4.6. Create Supports

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

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

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



-Restraints & Spring va	lues
Quick Selection :	Pin 🗸
Restraint Spring	Restraint Spring
√ Fx	Mx
√ Fy	My
✓ Fz	Mz

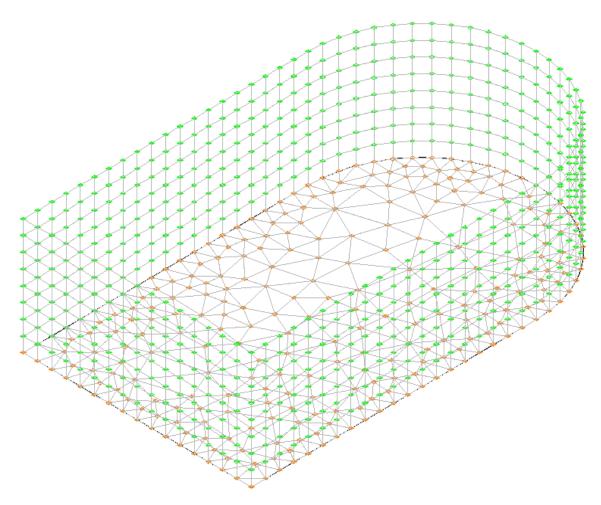
The Joint Properties [Multiple Selection] form appears.

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

Press OK.

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

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



4.7. Check the model

Step #12. Check for duplicate joints: In order to check for joints having the same coordinates,

click on the icon ^{Joints Duplicates}, under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area.

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

You should get the notification that O duplicate joints found .

Step #13. Check for floating joints: In order to check for joints not connected to the model, click

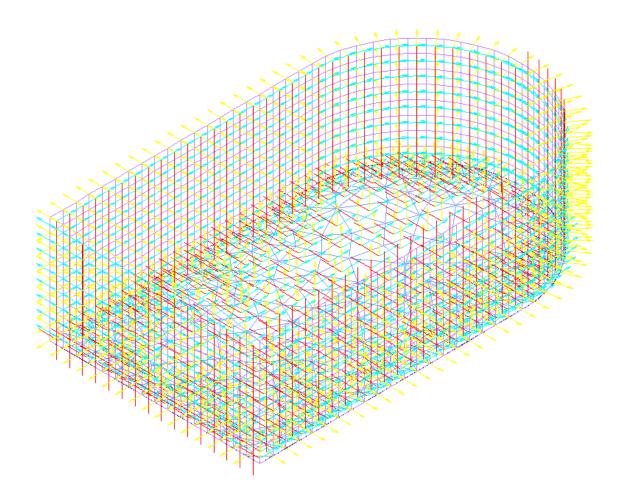
on the icon ^{Joints Floatings}, under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area. If your model was created as described so far, you should get a notification that *O floating joints found*.

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

Step #14. View Planar Axes for Finite Elements: In order to check the direction of the planar axes

of the Finite elements, click on the icon $\stackrel{\text{IP} \text{ Shell Planar Axes}}{\text{ in the "GTS Display" Ribbon area}}$ and then *Enter Legend Coordinates(x,y,z):* or click at the point where you want the legend to be displayed.

In the legend, the X axis is displayed in cyan, Y axis in red and Z axis in yellow (AutoCAD) or orange (BricsCAD). The size of the arrow and its arrowhead is controlled by the value given in Display Options > Object Sizes > Load Arrowhead and the size of the legend font is controlled by the value given in Display Options > Label Settings – Font Sizes > Annotation (pts) (shown in <u>2.6.64</u>).



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

4.8. Define Groups

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

٠	_	-
٠	_	-
	_	-
	_	-

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

		Groups	×
Group Proper	ties		Add Group
ID	Name	Physical	
1	Bottom		Delete Group
2	Wall		
			OK
			Cancel

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

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

Press OK to close the form.

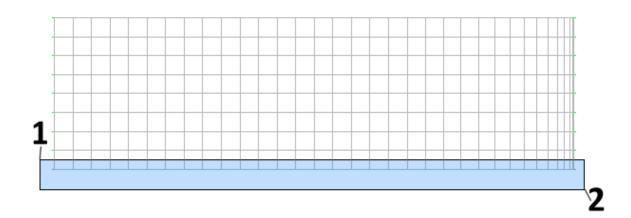
Step #16. Add the elements in the bottom of the tank to the Group Bottom:

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

-		
Group		
Bottom	- V	

Click on the icon + Shells 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.



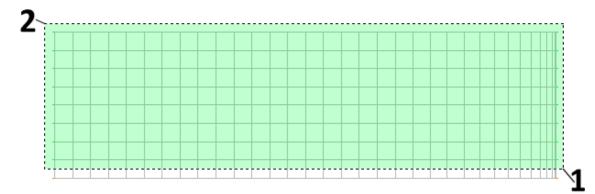
All the shell elements located in the bottom of the tank are now selected and added to group "Bottom".

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

	Select Group		
Group			
Wall		~	

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

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

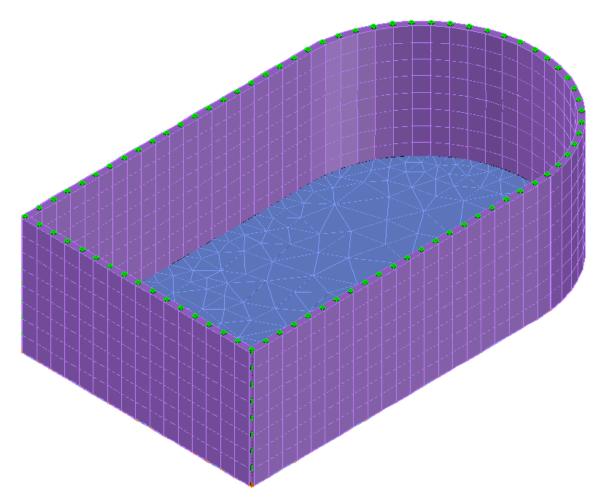


All shell elements located in the wall of the tank are now selected and added to group "Wall".

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

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

Click at the icon is 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 ^{IIII} Self Weight and the Self Weight dialog appears. Enter:

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

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

	Self Weight or	Dead Loads ×
Load Informatio	n	
Name : Description :	SW 🗸	
Create Net	w Save / Mo	odify Delete
-Loads applied	d parallel to this Globa	al Axis
🔿 Nega	tive Y	O Positive Y
🖲 Nega	tive Z	O Positive Z
🔿 Nega	tive X	○ Positive X
Factor :		✓ Include Finite Elements
		OK Cancel

Step #19. Define Load Cases: Click on the icon ¹ Load Cases</sup> and the Load dialog appears.

Enter:

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

and press Create New.

Load Case				×
Load Case Information				
Load ID (up to 8 chars) :	LL	Design variable :	L I V	Create New
Description :	Live Load			Save / Modify

Enter:

- PL as Name
- Pressure Load as the Load Description

and press Create New.

Load Case		×
Load Case Information		
Load ID (up to 8 chars) :	PL Design variable : V	Create New
Description :	Pressure Load	Save / Modify
Load ID List :	LL	Delete

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 \bigcirc Colors. Select the 2nd Tab in order to colorize elements by their group and make only the Group Bottom visible and press OK.

Sections Groups		
Categories		
Groups	Color	Visible
Bottom	161	
Wall	191	
UnGrouped data	256	

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

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

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

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

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

Applied Load Cases Loading Load Case : Load Case : Load Case : Description : Load Case : Porce Type Orection Ox	lodel Element Loads			Itiple Selection]		
Empty Load Cases Save / Modify Loads in this Load Case Image: Construction of the second	Applied Load Cases					Create New
Loads in this toad Case : Load Distribution Values (B or S) Force Type Oirection System Load Distribution Values (B or S) Body X O Local O Uniform v1 : -40 Surface Y O Global Variable v3 : - Empty Load Cases Z Projected v4 : -		Line	e Load			Save / Modify
Body X Local Uniform v1: -40 Surface Y Planar v2: - Empty Load Cases Z Projected v4:		Loads in this Load (Case :	¥		Delete
Image: Surface Image: Surfac		Force Type	Direction	System	Load Distribution	Values (B or S)
Image: Surface Y Image: Surface Y Image: Su		OBody	⊖x		 Uniform 	v1: -40
Empty Load Cases Image: Constraint of the second		Surface	OY	O Planar		v2:
Empty Load Cases				Global	○ Variable	v3 :
LL \$[0]		⊖ Edge ∨	⊙ z	O Projected		v4:
	LL \$[0]					

Press OK to close the dialog.

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

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

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

Sections Groups		
Categories		
Groups	Color	Visible
Bottom	161	
Wall	191	V
UnGrouped data	256	

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

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

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

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

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

Applied Load Cases	Loading Load Case : PL Description : Pre Loads in this Load (essure Load	~		Create New Save / Modify Delete
	Force Type	Direction	System	Load Distribution	Values (B or S)
	OBody	Ox	Local	 Uniform 	v1: 5
	Surface	ОY	🔾 Planar	🔿 Variable	v2:
	⊖ Edge ∨	• z	O Projected		v4:
Empty Load Cases [LL \$[476] [PL \$[656]					

Press OK to close the dialog.

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

Click on the icon ^{O Colors}. Select the 2nd Tab and make everything visible.

Step #22. Define a Load Combination: Click on the icon ^{IIII} Combinations</sup> and the Load Combination dialog appears. Enter:

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

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

New Form Load or Load Combination		×
Load Information Name : CB1 Design variable : Description : Load Combination 1	Type © Load Combination ○ Form Load	
Combine SW (Self Weight) LL (Live Load) PL (Pressure Load)	SW 1.30000 LL 1.5000 PL 1.10000	Delete Item
All Formed Loads or Combinations		Done

4.10. Create GT STRUDL Input File

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

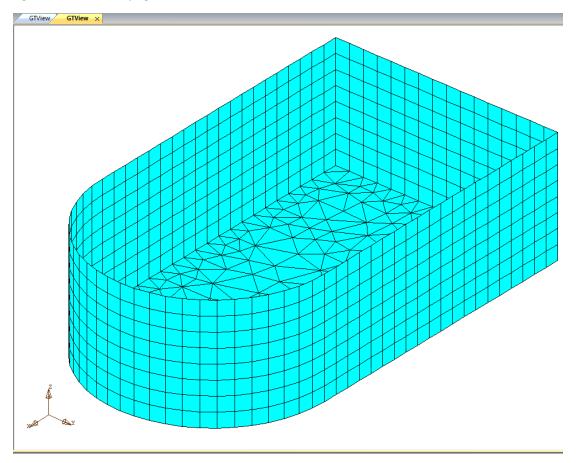
	Create GT.STRUDL Input File		
Export to GTI			
Create GTI File :	Documentation\Versio	n2019\Example02\Example02Ste	ep22.GTI
• Export Whole	Model	 Export Portion of the Moo (will be prompted for Sele 	
 Perform Stiffn 	✓ Perform Stiffness Analysis		
Append Other GTI	Files/Macros		
			+ - Up Down
Copy Comman	ds from GTI Files/Macro	s (not CINPUT)	
Create Commands	to Read Results		
Read Joint Dis	placements		
Read Member	Forces		
Read Section	Forces	Number of Sections:	10
Read Section I	Displacements	Number of Sections:	10
Read Finite Ele	ement Results		
Read Code Ch	neck Results		
		ОК	Cancel

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

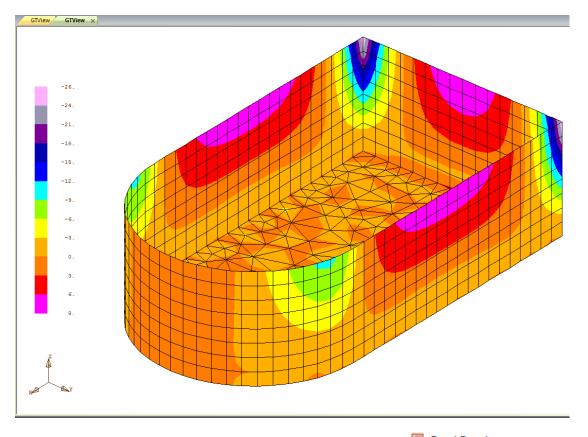
Step #25. Execute GT STRUDL: Click on the icon **GTS** and the GTI file created in the previous step will be sent to GT STRUDL main program that is waiting in the background.

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

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



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



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

Rea	d GT.STRUDL Results	×
Read DBX Results		
GTI Directory : F:\00.STRUDL	Europe	
Read Joint Displacements		
Read Member Forces		
Read Section Forces	Number of Sections:	10
Read Section Displacements	Number of Sections:	10
Read Finite Element Results		
Type of element result :	<all></all>	~
Surface :	<all></all>	*
Read Code Check Results		
GTI Commands		
Corresponding GT STRUDL comma	ands:	
WRITE REPLACE AVERAGE ELEM WRITE REPLACE AVERAGE RESL WRITE REPLACE AVERAGE ELEM WRITE REPLACE AVERAGE ELEM	S JOINTS EXISTING IENT STRESSES TOP MIDDLE BOTOM ELE IENT STRAINS TOP MIDDLE BOTOM ELEM JLTANTS TOP MIDDLE BOTOM ELEMENTS IENT PRINCIPAL STRESSES TOP MIDDLE B IENT PRINCIPAL STRAINS TOP MIDDLE B ICIPAL MEMBRANE RESULTANTS TOP MID	ENTS EXIST EXISTING BOTOM ELEN OTOM ELEM
	OK	Cancel

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

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

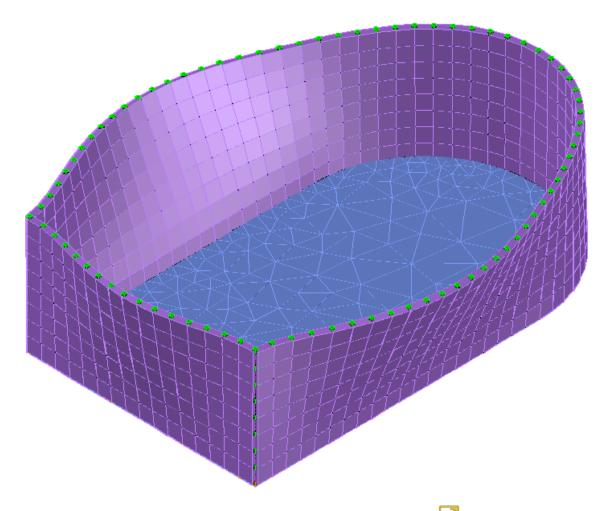
```
STDBX34 - Strains
```

STDBX37 - Principal Strains

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

4.11. Display Results

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



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

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

Another alternative is to click on *Probability* Displacements (ribbon tab "GTS Display").

Displacements				
Load Case / Load Combination				
PL \$ Pressure load	~			
Display Options Scale Factor (Values) : Font Size (pt) : Annotation Format: Decimal Places : I Hide Model	0.100 10.00 Exponential V 2 V			
Display >>				
Annotate >	Legend >			
Animation Options Frames : Animation Speed % : Generate Animation Frames	7 10 ¥ Animate >>			
Clear	Close			

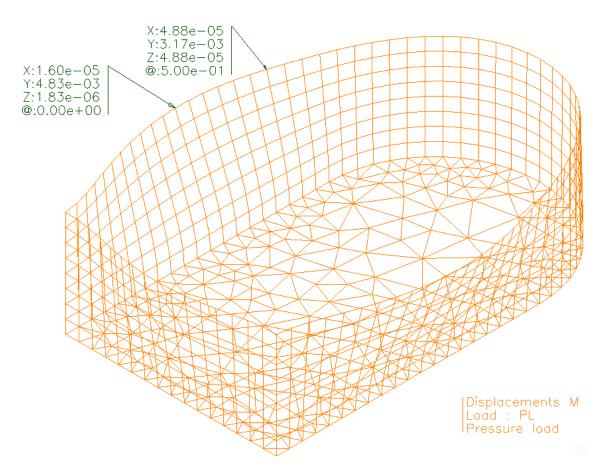
Select:

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

Press "Display >>" to view the deformed shape.

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

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



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

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

Step #28. Show Finite Element Results: Click on ^{Elements} (ribbon tab "GTS Display") and the Element Results Form appears.

Element Results ×			
Load Case			
PL \$ Pressure load	*		
Туре			
Resultants	~		
Mxx	~		
Middle	~		
Display Options			
Scale Factor (Values)	0.100		
Font Size (pt)	10.00		
Annotation Format:	Decimal 🗸		
Decimal Places :	2 🗸		
Display >>			
Annotate >	Legend >		
Animation Options			
Frames :	7		
Animation Speed % :	10 🗸		
Generate Animation Frames	Animate >>		
Clear	Close		

Select:

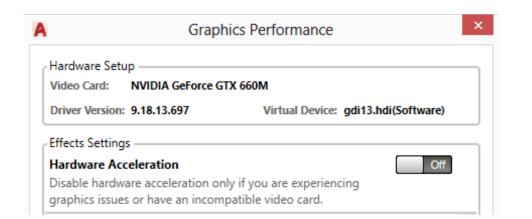
- PL as Load Case
- *Resultants* as the Type of element result
- *Mxx* as the Moment Resultant to display
- *Middle* as position (Resultants are only available for the middle surface of a 2D finite element)

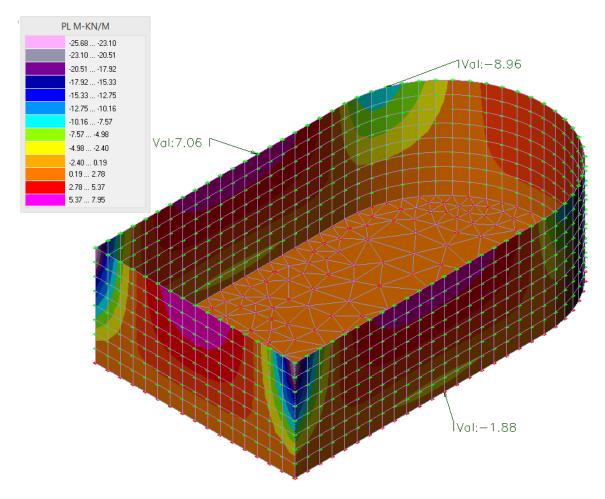
and press "Display >>"

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

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

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





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

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

4.12. Results Datasheets

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

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

oint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y	Γ
1	SW	0.0000	0.0000	0.0000	0.0000106	-0.0000138	
2	SW	0.0000	0.0000	0.0000	-0.0000232	0.0000019	
3	SW	0.0000	0.0000	0.0000	-0.0000178	0.000063	
4	SW	0.0000	0.0000	0.0000	-0.0000217	-0.0000060	
5	SW	0.0000	0.0000	0.0000	-0.0000299	0.0000074	
6	SW	0.0000	0.0000	0.0000	-0.0000240	-0.0000108	
7	SW	0.0000	0.0000	0.0000	-0.0000245	0.0000075	
8	SW	0.0000	0.0000	0.0000	-0.0000541	0.0000093	
9	SW	0.0000	0.0000	0.0000	-0.0000309	-0.0000440	
10	SW	0.0000	0.0000	0.0000	-0.0000478	0.0000193	
11	SW	0.0000	0.0000	0.0000	-0.0000426	-0.0000289	
12	SW	0.0000	0.0000	0.0000	-0.0000393	0.0000310	
13	SW	0.0000	0.0000	0.0000	-0.0000511	-0.0000206	
14	SW	0.0000	0.0000	0.0000	-0.0000470	0.0000164	
15	SW	0.0000	0.0000	0.0000	-0.0000420	-0.0000298	
16	SW	0.0000	0.0000	0.0000	-0.0000307	0.0000099	
17	SW	0.0000	0.0000	0.0000	-0.0000394	0.0000016	
18	SW	0.0000	0.0000	0.0000	-0.0000175	-0.0000173	
19	SW	0.0000	0.0000	0.0000	-0.0000129	0.000066	
20	SW	0.0000	0.0000	0.0000	-0.0000321	0.0000080	
21	SW	0.0000	0.0000	0.0000	-0.0000101	-0.0000294	
22	SW	0.0000	0.0000	0.0000	0.0000268	0.000065	
23	SW	0.0000	0.0000	0.0000	0.0000194	0.000086	
24	SW	0.0000	0.0000	0.0000	0.0000164	0.0000071	
25	SW	0.0000	0.0000	0.0000	0.0000193	0.0000075	
26	SW	0.0000	0.0000	0.0000	0.0000132	0.0000096	
27	SW	0.0000	0.0000	0.0000	0.000088	0.000062	
28	SW	0.0000	0.0000	0.0000	-0.0000073	0.0000003	

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

Joint I 1 1 1 1 1 1 1 1 1 1	Load SW SW LL LL LL LL PL	Surface Top Middle Bottom Top Middle Bottom Top	SXX -9.072 -10.035 -10.999 12.447 -0.211 -12.869	SYY -79.302 -68.864 -58.427 -9.762 -0.052 9.658	SXY 1.757 -0.187 -2.131 16.378 0.225	VonMises 43.439 37.202 31.122 19.803	5
1 1 1 1 1 1 1	SW SW LL LL LL PL	Middle Bottom Top Middle Bottom	-10.035 -10.998 12.447 -0.211 -12.868	-68.864 -58.427 -9.762 -0.052	-0.187 -2.131 16.378	37.202 31.122 19.803	
1 1 1 1 1 1	SW LL LL LL PL	Bottom Top Middle Bottom	-10.998 12.447 -0.211 -12.868	-58.427 -9.762 -0.052	-2.131 16.378	31.122 19.803	
1 1 1 1 1	LL LL LL PL	Top Middle Bottom	12.447 -0.211 -12.868	-9.762 -0.052	16.378	19.803	
1 1 1 1	LL LL PL	Middle Bottom	-0.211	-0.052			
1 1 1	LL PL	Bottom	-12.868		0.225		
1	PL			9 659		0.250	
1		Top	1 000	2.000	-15.928	19.530	
_	DT.		1.229	-32.917	-13.717	23.734	
1	5.7	Middle	-1.770	-39.660	3.729	22.713	
-	PL	Bottom	-4.768	-46.403	21.175	33.166	
2	SW	Top	-6.494	-41.199	-3.870	22.486	
2	SW	Middle	-6.716	-36.815	-3.170	19.860	
2	SW	Bottom	-6.939	-32.431	-2.470	17.255	
2	LL	Top	-2.938	-40.908	-8.212	24.250	
2	LL	Middle	-0.010	-0.434	1.038	1.067	
2	LL	Bottom	2.918	40.039	10.288	24.578	
2	PL	Top	26.386	-11.697	-30.198	35.951	
2	PL	Middle	-2.172	-8.818	0.775	4.659	
2	PL	Bottom	-30.731	-5.938	31.748	35.689	
3	SW	Top	-2.867	-41.443	-2.248	23.253	

4.13. Report Builder



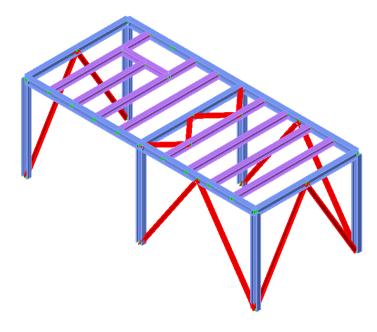
Step #57. Click on the icon Builder to generate the report of this structure by calling Report Builder. Report Builder is launched in a separate window. You can find more information on how creating your report in the GT STRUDL[®] Report Builder Getting Started Guide (Help Icon on the right top corner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

Home										Style		
		Joint	Name: ALL		Loadii	ng: ALL			🦅	00.00		
🔛 🔼 🚮 🖻		Men	ber Name: ALL		Level:	ALL			- V	@.00 .0E		
Generate Image Text RTF Overall Report File	Insert Remove	Print Print Preview Elem	ent Name ALL		Surfac	e: ALL			Apply Filters ··· Checked Iter			
	eader and Footer	Print			Filter				··· Checked iter	Format		
TSTRUDL Reports												
Contents	Load Data											
Model Data	Loud Data											
🔲 Groups												
Joint Coordinates	Element Loa	ds										
Joint Support Restraints												
Member Incidences	Length: M, Force: KN, Angle: DEG, Temperature: DEGC , Time: SEC											
Element Incidences												
Element Properties	Element Loa	ds (Body Force)										
Load Data		Element	Load Case	Coordinate System	Distribution	Joint	BX	BY	BZ			
🔽 Summary of Loadings 🗔 Loading Combinations		1	SW	GLOBAL	UNIFORM		0.000e+00	0.000e+00	-2.356e+01			
Joint Loads		2	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Member Loads		3	SW	GLOBAL GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Element Loads		4	SW	GLOBAL	UNIFORM UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
Analysis Results		6	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
🛄 Joint Displacements		7	SW	GLOBAL	UNIFORM	* *	0.000e+00	0.000e+00	-2.356e+01			
Support Joint Reactions		8	SW	GLOBAL GLOBAL	UNIFORM UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
Member Forces		10	SW	GLOBAL	UNIFORM		0.000e+00	0.000e+00	-2.356e+01			
Average Element Results		11	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Stresses		12	SW	GLOBAL GLOBAL	UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
- Stresses Envelope		13	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
🔲 Stresses Summary		15	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Resultants		16	SW	GLOBAL GLOBAL	UNIFORM UNIFORM	* *	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
Resultants Envelope		17	SW	GLOBAL	UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01			
Resultants Summary Principal Stresses		19	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
- Principal Stresses Envelope		20	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
- Principal Stresses Summary		21 22	SW	GLOBAL GLOBAL	UNIFORM UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
		23	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
🔄 Principal Membrane Resultants Er		24	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Principal Membrane Resultants Su		25	SW	GLOBAL GLOBAL	UNIFORM UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
Principal Bending Resultants Principal Bending Resultants Enve		20	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Principal Bending Resultants Enve		28	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Von Mises		29	SW	GLOBAL GLOBAL	UNIFORM UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
		30	SW	GLOBAL	UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
Von Mises Summary		32	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Member Results Graphs		33	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
Design of Steel Members		34	SW	GLOBAL GLOBAL	UNIFORM UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			
		36	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
		37	SW	GLOBAL	UNIFORM	*	0.000e+00	0.000e+00	-2.356e+01			
		38	SW	GLOBAL GLOBAL	UNIFORM UNIFORM	*	0.000e+00 0.000e+00	0.000e+00 0.000e+00	-2.356e+01 -2.356e+01			

5. Tutorial Example #3

5.1. Introduction

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



5.2. Open CAD Modeler and start working

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

c	ome to GT STRU	UDL									
	Open	Learn more about GT S	STRUDL			Hide Details	←	\rightarrow	ê	0	8
0	Help Options	GT Menu Perform structural analysis & design using a graphical user interface.		Command Mode Learn the STRUDL design language to create computer models for structural analysis and design.	Model Wizard Quicldy model simple structures using templates.	Base Plate Wizard Analyze & design complex structural itel base plates using a finite element analysis approach.		HEX PPM	AGON		CADWorx® Analysis Sc
		Learn More	* CADModeler is not installed. Learn More	Learn More	Learn More	* Advanced license required Learn More	Introduc View an	informativ presented	DL? STRUDL e webinar or on January 2	the new	
		(A)		 			Now Av	ailable: GT version of the on to the mo:	STRUDL 201 product that c st frequently us	an be use	
		GT Menu Working Directory	CAD Modeler	Command Mode	Model Wizard	Base Plate Wizard		D-Based Mo bruary 20, 20	deling for Strue	<u>tural Engi:</u>	neers
		D:\GTQA\Testing\202	20-04-27_UTC-03-22-51\GTS	5 2020 Shell\MO069_gti			Co	nnecting Des ructural and I	roject Executio sign to Engines Piping Recorde	ring — Uni	iting
		Recent					M		ebinar: GT STR y. with GT STR r 13, 2018		
							St	ructural Work	abinar: GT STF flow with Interg October 18, 2	raph Smar	
							<				

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



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

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

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

5.3. Define the basic geometry of the model

Step #3. Define the correct **Units** by pressing the icon $\overset{\text{W}}{\overset{\text{Units}}}$ and select *Feet (m)* and *Kips* in the *Units Form*.

Units	×	
Length	Force	Angles
O Inches (in)	O Pounds (lbs)	Degrees
Feet (ft)	Kips	◯ Radians
O Meters (m)	◯ Tons	○ Cycles
O Centimeters (cm)	◯ Kilograms	
O Millimeters (mm)	O Metric Tons	Time
Temperature	ONewtons	 Seconds
○ Fahrenheit	◯ KiloNewtons	○ Minutes
 Centigrade 		OHours
Scale non-structur (grids, structural lin	OK Cancel	
		Ē

Step #4. Define the one level of the model by pressing the icon ^{Levels}. Press the *Add Level* button one time to add a level to your model. Modify the height of the 1^{st} level by selecting the *Height* cell of the 1^{st} Level and entering 15.

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

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

Le	vel Properties					Х
	Level Heights				Options	
	Levels	Height	Elevation	Visible	Add Level	
	1	15.000000	15.000000	V	Delete Level	
					Detect Levels Automaticaly	
					Merge Levels	
					Base Elevation 0.0000]
					Z Vertical Axis (else Y)	
					Update Levels for All Entities	s
					OK Cancel	



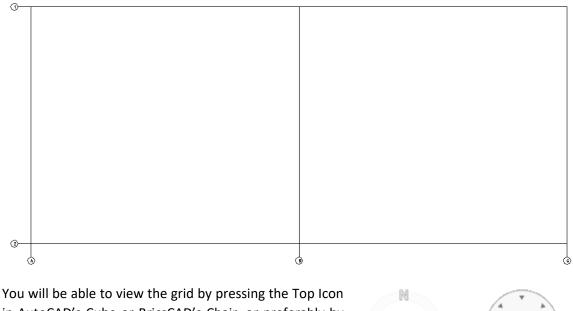
Step #5. Enter a Grid that will help you enter the columns quickly by clicking on the icon Grid . The grid is going to have 2 (20.4167 ft) spaces in the horizontal direction (X) and 1 space (18 ft) in the sidelong direction (Y). Enter 20 ft-5 in in the *Distance* text box and press the *Add* button 2 times. The program will automatically convert the distance of 20ft -5in to 20.4167 ft.

Grid	
Placement	
 Horizontal 	◯ Sidelong
Spacing	
20.4167 20.4167	Distance: 20.4167
	Add
	Edit
	Delete

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

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

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



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



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

✔ Object Snap On (F3)	Object Snap Tracking On (F11)	Endpoint
Object Snap modes		Midpoint
Endpoint	Select All	Center Node
		Ouadrant
	Perpendicular Clear All	✓ Intersection
○ □ Center	🕤 🗌 Tangent	Insertion
⊠ ∏ Node	∑	Perpendicular
		Tangent Nearest
🔷 🗌 Quadrant	Apparent intersection	Geometric center
X Intersection		Apparent intersection
		Extension
Extension		Parallel
To track from an Osnap p	point, pause over the point while in a	✓ On
command. A tracking ve	ctor appears when you move the cursor.	Off
To stop tracking, pause of	over the point again.	Settings
		ESNAP STRACK LWT TILE DUCS DYN

AutoCAD's Snap Setting

BricsCAD's Snap Settings

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





AutoCAD's View Cube for Isometric View

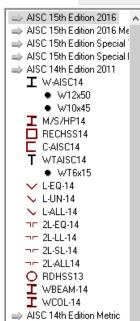
BricsCAD's View Cube for Isometric View

I

Enter the cross-section profiles that will be used at the model by pressing the icon $\frac{\text{Sections}}{\text{Sections}}$. Click on the *AISC 14th* Edition 2011 list and then on the *W-AISC14* table of profiles. Select the profile *W12x50* that will be used for the columns, by double clicking on it.

AISC 15th Edition 2016 AISC 15th Edition 2016 Me						W	-AISC	15					
AISC 15th Edition Special AISC 15th Edition Special I	NAME1	ND	AX	YD	WBTK	ZD	FLTK	KDES	KDET	К1	WEIGHT	BF 2TF	H T₩ /
AISC 14th Edition 2011	W12x136	1	0.2771	1,117	0.06583	1.033	0.1042	0.1542	0.1771	0.1042	0.4462	4.96	12.3
T W-AISC14	W12x120	1	0.2444	1.092	0.05917	1.025	0.0925	0.1417	0.1667	0.09896	0.3937	5.57	13.7
T M/S/HP14	W12x106	1	0.2167	1.075	0.05083	1.017	0.0825	0.1325	0.1563	0.09375	0.3478	6.17	15.9
RECHSS14	W12x96	. 1	0.1958	1.058	0.04583	1.017	0.075	0.125	0.151	0.09375	0.315	6.76	17.7
C-AISC14	W12x87	1	0.1778	1.042	0.04292	1.008	0.0675	0.1175	0.1406	0.08854	0.2854	7.48	18.9
T WTAISC14	W12x07	1	0.1611	1.042	0.03917	1.008	0.06125	0.1108	0.1354	0.08854	0.2592	8.22	20.7
✓ L-EQ-14 ↓ L-UN-14	W12x73	1	0.1465	1.035	0.03583	1.000	0.05583	0.1058	0.1304	0.08854	0.2352	8.99	20.7
✓ L-ON-14 ✓ L-ALL-14	W12x72	1	0.1465	1.025	0.03565		0.05042	0.1058		0.08333		9.92	24.5
⊐r 2L-EQ-14						1			0.125		0.2133		
⊐⊏ 2L-LL-14	W12x58	1	0.1181	1.017	0.03	0.8333	0.05333	0.1033	0.125	0.07813	0.1903	7.82	27
⊐⊏ 2L-SL-14	W12x53	1	0.1083	1.008	0.02875	0.8333	0.04792	0.09833	0.1146	0.07813	0.1739	8.69	28.1
nr 2L-ALL14	W12x50	1	0.1014	1.017	0.03083	0.6733	0.05333	0.095	0.125	0.07813	0.164	6.31	26.)
O RDHSS13	W12x45	1	0.09097	1.008	0.02792	0.6708	0.04792	0.09	0.1146	0.07813	0.1476	7	29.6
WBEAM-14	W12x40	1	0.08125	0.9917	0.02458	0.6675	0.04292	0.085	0.1146	0.07292	0.1312	7.77	33.6
AISC 14th Edition Metric	W12x35	1	0.07153	1.042	0.025	0.5467	0.04333	0.06833	0.09896	0.0625	0.1148	6.31	36.2
AISC 14th Edition Metric	W12x30	1	0.06104	1.025	0.02167	0.5433	0.03667	0.06167	0.09375	0.0625	0.09843	7.41	41.8
AISC 13th Edition 2005 Me	W12x26	1	0.05312	1.017	0.01917	0.5408	0.03167	0.05667	0.08854	0.0625	0.0853	8.54	47.2
AISC 9th Edition 1989	W12x22	1	0.045	1.025	0.02167	0.3358	0.03542	0.06042	0.07813	0.05208	0.07218	4.74	41.8
AISC LRFD 3rd Edition	W12x19	1	0.03868	1.017	0.01958	0.3342	0.02917	0.05417	0.07292	0.04688	0.06234	5.72	46.2
AISC 9th Edition Metric	W12x16	1	0.03271	1	0.01833	0.3325	0.02208	0.04708	0.06771	0.04688	0.05249	7.53	49.4
AISC 8th Edition 1978	W12x14	1	0.02889	0.9917	0.01667	0.3308	0.01875	0.04375	0.0625	0.04688	0.04593	8.82	54.3
AISC 7th Edition 1969	W10x112	0.8333	0.2285	0.95	0.06292	0.8667	0.1042	0.1458	0.1615	0.08333	0.3675	4.17	10.4
AISC 6th Edition 1963 ANSI Pipe	W10x100	0.8333	0.2035	0.925	0.05667	0.8583	0.09333	0.135	0.151	0.08333	0.3281	4.62	11.6
ANSI Pipe Brazilan Standard, NBR 58	W10x100	0.8333	0.1806	0.525	0.05042	0.8583	0.0825	0.1242	0.1406	0.07813	0.2887	5.18	13
British Standard 5950	W10x00	0.8333	0.1576	0.8833	0.04417	0.0000	0.0725	0.1142	0.1302	0.07292	0.2526	5.86	14.8
European	<	0.0000	0.1570	3.0033	0.04417	0.00	0.0723	0.1142	0.1502	0.012.02	0.2320	5.00	>
Intergraph Smart 3D tables	`												

Member Sections



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

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

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

5.4. Create Columns

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

Cross Section	
Table Section:	
W12x50 W-AISC14 AISC 14th Edition 2011 V	
or Member Dimensions:	
<select shape=""></select>	~
or Same as Member:	
<select existing="" member=""></select>	~
Material Steel V	
Releases Beta (o)	
Fix-Fix V 90	~
Place Member(s) >>	

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

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

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

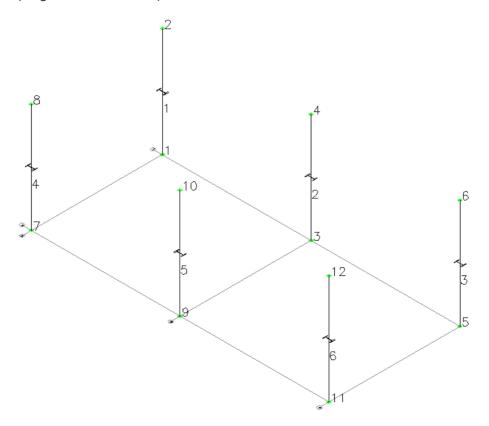
Repeat the same procedure by clicking at the points B-1, C-1, A-2, B-2 and C-2.

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



Note: Each time you create a member, the orientation of the cross section will appear in the middle of the element, unless you clear it with command "Clear" (see 2.6.84).

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



5.5. Create beams and girders

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

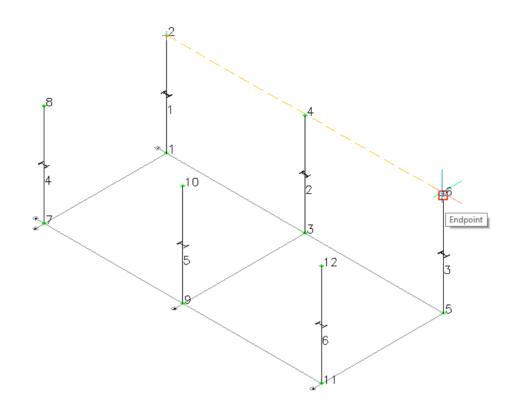
Cross Section	
Table Section:	
W12x50 W-AISC14 AISC 14th Editi	on 2011 🛛 🗸
or Member Dimensions:	
<select shape=""></select>	~
or Same as Member:	
<select existing="" member=""></select>	*
Material Steel	v
Releases	Beta (o)
Fix-Fix ¥	90 🗸
 Split Intersecting Members Split Ending Members 	✓ Physical Member
Place Member(s) >>

Select W12x50 as the cross section and make sure that Material is set to Steel, Releases to Fix-Fix and the Beta angle is 90. You have to set the Beta angle equal to 90 degrees in order to orientate the local Y axis of the W12x50 cross section along the Z global axis. Make sure that the option Split Intersecting Members is checked and then uncheck Split Ending Members. Moreover, check the "Physical Member" option, so that you also define the physical member and not only the analytical members.

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

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

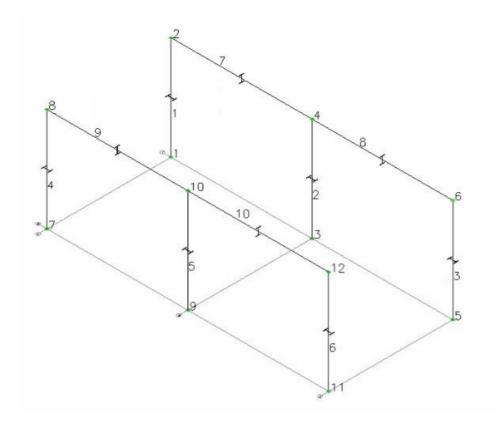
Then you have to define the *Ending Point* (x,y,z) so click on Joint 6, as shown in the picture below.



Members 7 and 8 will be created.

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

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

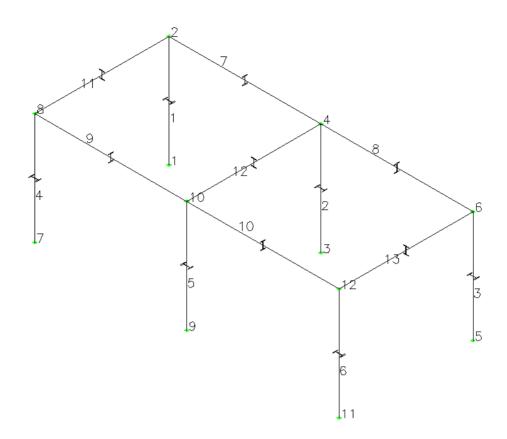


Step #10. Enter the beams, along Y axis. The command Generate Beams should be still active,

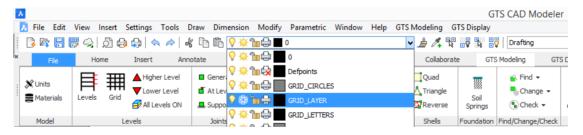
else you can call it again by clicking again on the icon Generate. Keep the same settings at the Place Member Form, as in the previous step, regarding the cross section and Beta angle, but do NOT click on *Split Intersecting Members*. Press the "Place Member(s) >>" button.

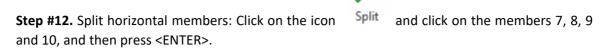
The prompt message *Starting Point* (x,y,z) appears, asking you to enter the coordinates or click on a specific point on screen. Click on the Joint 8, that is the top of column at position A-2. In order to define the *Ending Point* (x,y,z), click at Joint 2 (top of column at position A-1). Member 11 is generated.

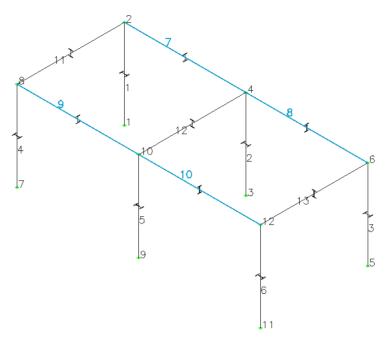
Repeat the same procedure by clicking on the Joints 10 and 4 to generate Member 12, Joints 12 and 6 to generate Member 13. Then, press ESC to terminate the command.



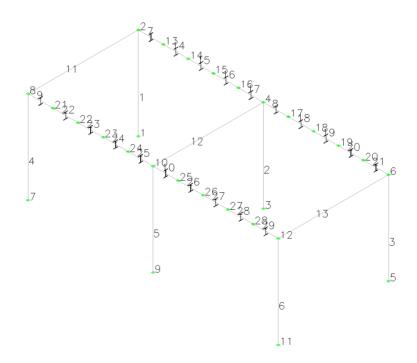
Step #11. Hide Grid: Since the Grid is no longer needed it can be hidden using AutoCAD/BricsCAD commands to freeze the layer GRID_LAYER. This can be done by clicking on the Home Tab and then selecting the GRID_LAYER from the drop down list of layers and clicking on its freeze icon.





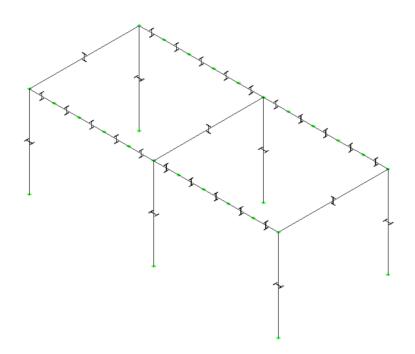


In order to define the Distance for spliting the member or the number of parts (negative number), enter -5, so that the beams will be split into 5 equal parts.

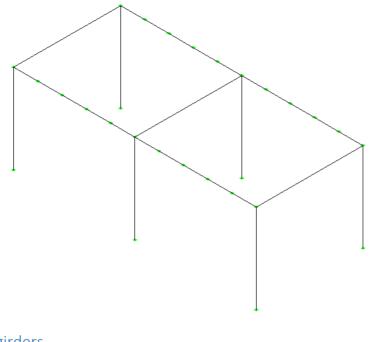


Step #13. Turn OFF labeling:





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



5.6. Create girders

Step #14. Place girder members along Y axis: Click on the icon Generate and Place Member form appears.

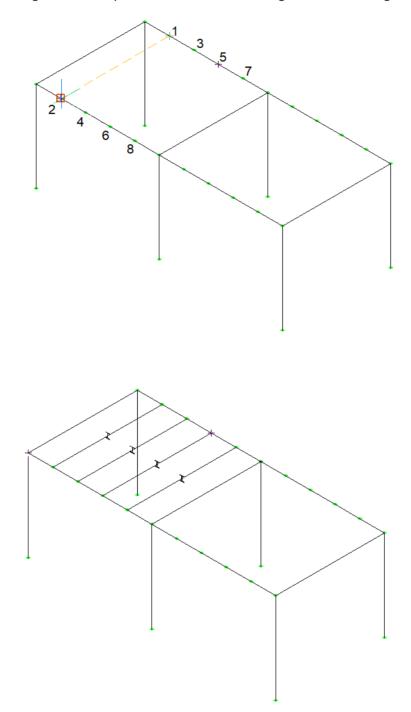
Cross Section						
Table Section:						
W10x45 W-AISC14 AISC 14th Edit	tion 2011 🛛 🗸					
or Member Dimensions:						
<select shape=""></select>	~					
or Same as Member:						
<select existing="" member=""></select>	~					
Material Steel	¥					
Releases	Beta (o)					
Pin-Pin 🗸	90 🗸					
Split Intersecting Members	✓ Physical Member					
Split Ending Members						
Place Member(s) >>						

Select W10x45 W-AISC14 14th Edition 2011 as the cross section and make sure that Material is set to Steel, Releases to Pin-Pin and Beta angle is 90. Uncheck Split Intersecting Members, Uncheck Split Ending Members and Check Physical Member

Press the "Place Member >>" button.

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

Having the command still active, click on the Joints at points 3 and 4 and another girder member is generated. Continue by clicking on Joints at points 5 and 6 and another girder member is generated. Continue by clicking on Joints at points 7 and 8 and another girder member is generated.



Press *ESC* to terminate the Generate Beam Command.

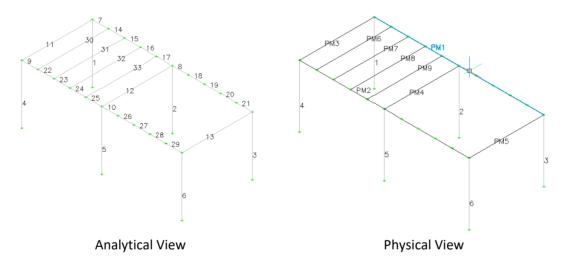
Press the icon ^{Clear} to remove the informative cross section shapes in the middle of each member.

Step #15. Switch between analytical and physical view.

Turn ON labeling:



Press the icon Analytical/Physical located in Display at Ribbon Area to switch from Analytical to Physical View as displayed in the picture bellow.



Physical Members, with the prefix PM, were generated while placing the analytical member, since the "Physical Member" option was checked. By double clicking on a Physical Member, in example PM1, you are able to see its properties, including the set of analytical members that define it. In example for PM1 the set is: 7,14,15,16,17,8,18,19,20,21 as created by placing the beam, and splitting it afterwards.

Physical Member Properties		×
Model Section Properties Member Loads		
General	Section Properties	Releases & Elastic Connection spring values
Name :	Section : W12x50 W-AISC14 AISC 14th \checkmark	Quick Selection :
Level : 1	Ax : 0.101389 Ix : 8.24653e-005	Start Spring End Spring
	Ay: 0.0313472 Iy: 0.00271508	□Fx 0 □Fx 0
Type - Incidences	Az : 0.0478815 Iz : 0.018856	
Type : SPACE FRAME V	Sy: 0.00804397 Ey: 0	
Start 2	Sz : 0.0371528 Ez : 0	Mx 0Mx 0
End : 6	Yd: 1.01667 Yc: 0.508333	My 0My 0
Beta Angle : 90	Zd: 0.673333 Zc: 0.336667	Mz 0Mz 0
Physical Member PM1 V	Shape Code : 1.0	
		End Sizes OR Member Eccentricities (Offsets)
Groups	Material Properties	Start End
	Material : Steel \checkmark	Sizes : 0
Analytical Members 7	E: 4.176e+006 Density 0.489543	x: 0 x: 0
14 15	G: 1.58399e+006 CTE: 1.17e-005	Y: 0 Y: 0
16 17 v	Poisson 0.3	
		OK Cancel Apply

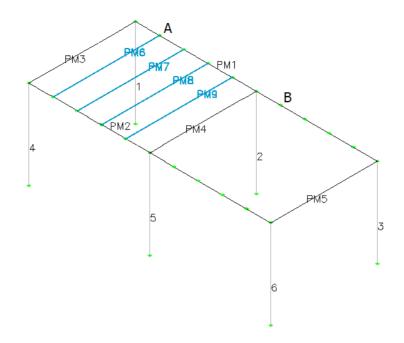
Step #16. Copy Physical Members

Having the physical member view ON the last 4 physical members created will be copied. If the COPY command is performed on physical members then the physical member is copied with its set of analytical members. If the COPY command is performed on analytical members, then only the analytical members are copied, without the physical member definition.

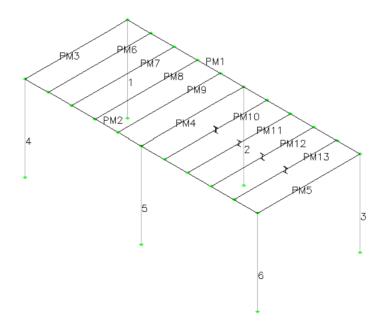
Type AutoCAD/BricsCAD COPY command and when asked to *Select objects/entites:* click on the members PM6, PM7, PM8 and PM9. You will get a notification about the selected entities and press <ENTER>.

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

In order to *Enter second point:* click on Point B of the following image.



Press ESC to exit from copy command. Another four girders, as physical members PM10, PM11, PM12 and PM13, are generated.

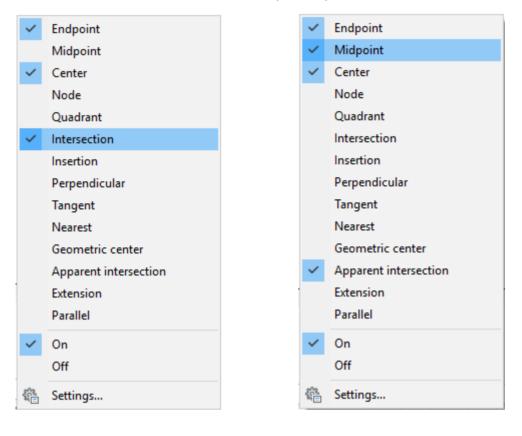


Switch back to analytical view by pressing the icon \Rightarrow Analytical/Physical and turn OFF member labeling (as explained in Step #13).

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

Step #17. Place bracing members.

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





Click on the icon Generate and Place Member form appears.

Cross Section	Select
Table Section:	Edition
WT6x15 WTAISC14 AISC 14th Edition 2011 V	and ma
or Member Dimensions:	to Stee Beta ar
<select shape=""></select>	
or Same as Member:	Make Interse
<select existing="" member=""> V</select>	that co
Material Steel V	along directio Membo
Releases Beta (o) Pin-Pin V	Press button
 ✓ Split Intersecting Members ✓ Split Ending Members 	Click o point Click o the mid
Place Member(s) >>	bracing

Select W16x15 WTAISC14 14th Edition 2011 as the cross section and make sure that Material is set to Steel, Releases to Pin-Pin and Beta angle is 0.

Make sure that the option "Split Intersecting Members" is ON, so that common joints will be created along the previously created Ydirection girders. Uncheck Physical Member option.

Press the "Place Member >>" putton.

Click on the Joint located at the point 1 of the following image. Click on point 3, that will snap to the middle of the member, and the bracing member is generated.

Having the command still active, click on the Joints at points 2 and 3 and another member is generated.

Continue by clicking on Joint at point 4 and midpoint 6 and another member is generated.

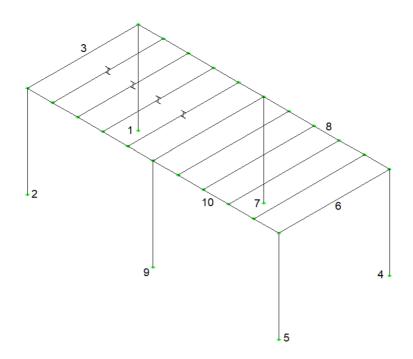
Continue by clicking on Joints at points 5 and 6 and another member is generated.

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

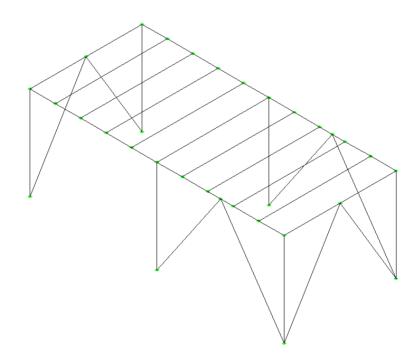
Continue by clicking on Joints at points 4 and 8 and another member is generated.

Continue by clicking on Joint at point 9 and midpoint 10 and another member is generated.

Continue by clicking on Joints at points 5 and 10 and another member is generated.



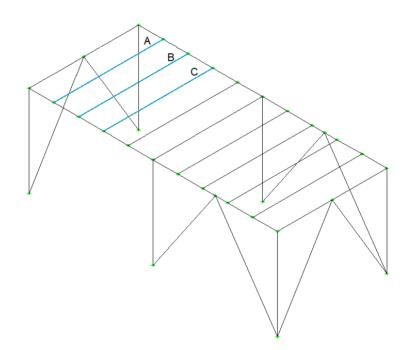
Press *ESC* to terminate the Generate Beam Command and click on the icon Clear to remove the informative cross section shapes in the middle of each member.

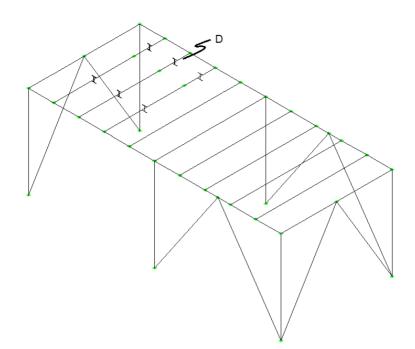


Step #18. Split Girders

Click on the icon **Split** and click on the members marked as A, B and C, and then press <ENTER>. In order to define the *Distance for spliting the member or the number of parts* (*negative number*), enter 5, so that the beams will be split at location 5 ft (current length units) from the start of the members.

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





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

Step #19. Add additional members along X



Click on the icon Generate and Place Member form appears.

Cross Section	
Table Section:	
W10x45 W-AISC14 AISC 14th Edit	tion 2011 🛛 🗸
or Member Dimensions:	
<select shape=""></select>	~
or Same as Member:	
<select existing="" member=""></select>	~
Material	
Steel	¥
Releases	Beta (o)
Pin-Pin 🗸	90 🗸
 Split Intersecting Members Split Ending Members 	Physical Member

Place Member(s) >>

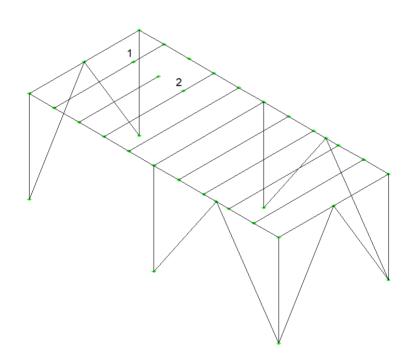
Select *W10x45 W-AISC14* 14th Edition 2011 as the cross section and make sure that Material is set to *Steel*, Releases to *Pin-Pin* and Beta angle is 90.

Make sure that the option "Split Intersecting Members" is ON, so that common joints will be created along the previously created Y-direction girders. Check Physical Member option.

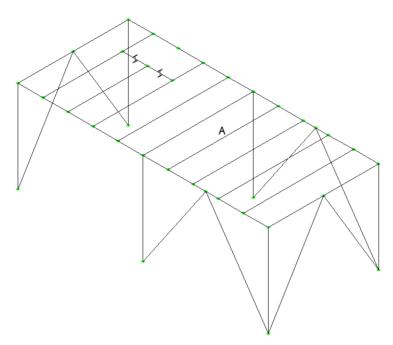
Press the "Place Member >>" button.

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

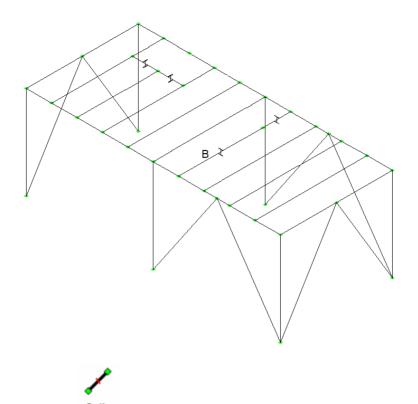
Press *ESC* to terminate the Generate Beam Command.



Step #20. Split Girders: Click on the icon Split and click on the member marked in A (of the following image) and then press <ENTER>.



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



Again, click on the icon Split and click on the member B (of the previous image) and then press <ENTER>. In order to define the *Distance for spliting the member or the number of parts (negative number)*, enter 9, so that the beam will be split at the location 9ft from the start of the member (see Notes of Step #18 for more information about the split command).

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

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

Cross Section	
Table Section:	
WT6x15 WTAISC14 AISC 14th Edi	tion 2011 🛛 🗸
or Member Dimensions:	
<select shape=""></select>	~
or Same as Member:	
<select existing="" member=""></select>	~
Material	
Steel	¥
Releases	Beta (o)
Pin-Pin 🗸	0 🗸
Split Intersecting Members	Physical Member
 Split Ending Members 	
Place Member(s	s) >>

Select WT16x15 WTAISC14 14th Edition 2011 as the cross section and make sure that Material is set to Steel, Releases to Pin-Pin and Beta angle is 0.

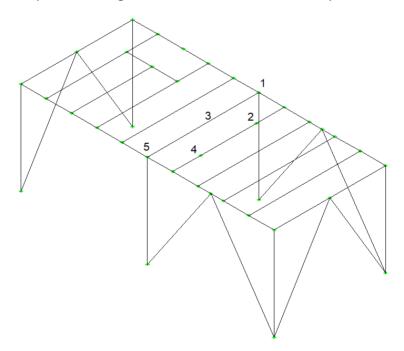
Make sure that the option "Split Intersecting Members" is ON and "Split Ending Members is also ON. Uncheck Physical Member option.

Press the "Place Member >>" button.

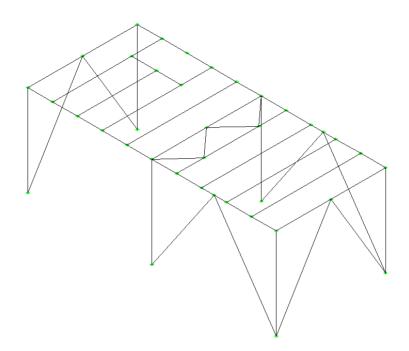
Click on the Joint located at the point 1 of the previous image. Click on the Joint at point 2 and the brace member is generated.

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

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



Press *ESC* to terminate the Generate Beam Command and click on the icon ^{Clear} to remove the informative cross section shapes in the middle of each member. The created model is displayed in the following image.

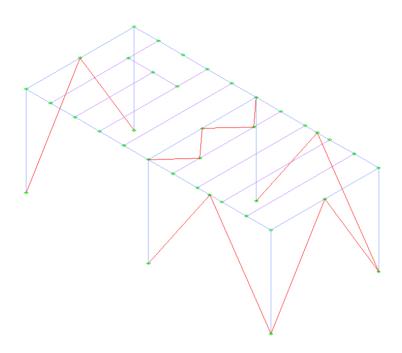


Step #22. Switch to 3D View

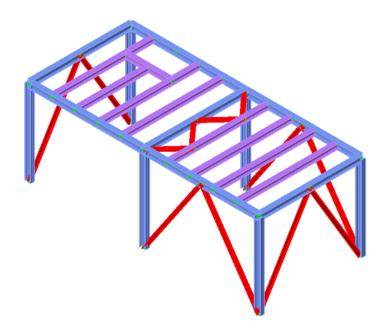
	Colors to set different colors for each profile.
Click on the icon	to set different colors for each profile.

Color Options					
Sections Groups					
Categories					
Sections		Color	Visible		
W12x50		161	\		
W10x45		191	✓		
WT6x15		1			

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



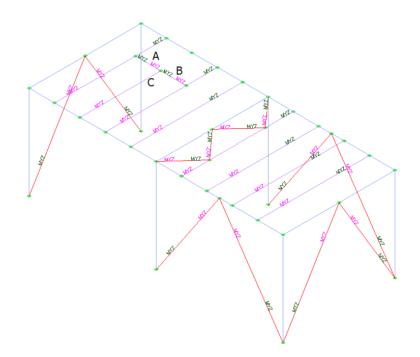
Press the icon 🗊 to display the 3D solid view of the model, replacing the wireframe view:



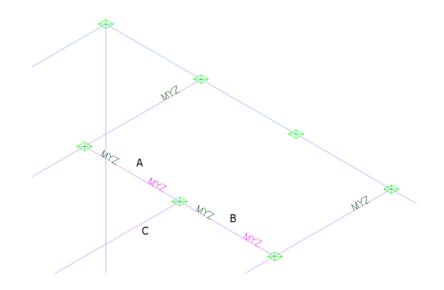
Press the icon \bigwedge Frame to switch back to wireframe view to be able to process CAD Modeler's and AutoCAD's/BricsCAD's commands faster.

Step #23. Save your Model: In order to save your model just use AutoCAD's or BricsCAD's save command and store the DWG using any filename that you want.

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



Click on the icon Clear to clear the annotations and zoom at the area of the following image. Again, press the Releases icon to show releases for each member. Click on the icon Clear to clear annotations.



el	Section Properties	Member Loads	Member	Temperature Lo	ads Me	mber Distrortions					
Gen	eral		Section	Properties			Releases & E	Elastic Connect	ion sprin	g values	
Nai	me:	53	Section	W 10x45 W-AI	SC14 AIS	SC 14th $\scriptstyle{\sim}$	Quick Selec	tion :		~	•
Le۱	/el:	1	Ax:	0.0923611	Ix:	7.28202e-005	Start Sp	ring	End	Spring	
			Ay:	0.0245486	Iy:	0.00257524	Fx	0	□ F x		0
Гуре	e - Incidences		Az:	0.0460408	Iz:	0.0119598	□Fy	0	□ Fy		0
Tvr	SPACE FF	RAME V	Sy:	0.00769676	Ey:	0	Fz	0	Fz		0
Sta		33	Sz:	0.0284144	Ez:	0	Мх	0	Mx	I	0
End	d:	34	Yd:	0.841667	Yc :	0.420833	✓му	0	Му		0
Bet	ta Angle :	90	Zd :	0.668333	Zc :	0.334167	Mz	0	Mz		0
Phy	ysical Member PM	14 ~	Shape C	ode :		1.0					
Grou	IDS							R Member Ecce art	entricities	s (Offsets) End	
	Inactive	Remove	Material	Properties				0			0
	oups that Member b	elonas	Material	: Steel		~	Sizes :	Ŭ			<u> </u>
_	114		E:	4.176e+006	Density	0.489543	x :	0	x :	1	0
			G:	1.58399e+006	CTE :	1.17e-005	Y:	0	Υ:		0
			Poisson	0.3]		z:	0	z:		0

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

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

Note: This error can also be located using the Check Physical Member command as well, as explained in 2.6.26 and this command will be used in a following step.

Error!!!	×
Physical Member PM14: Ending Member 54 has internal RELEASES at its start	
ОК]

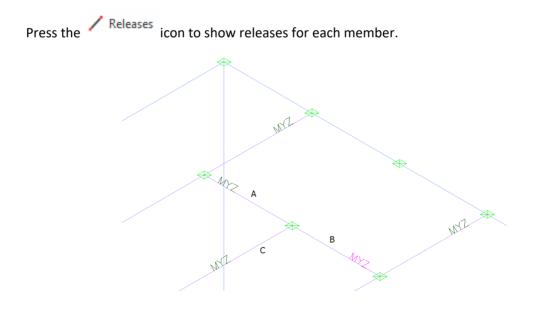
In order to fix the physical member PM14, select **member marked in B** by double clicking on it and uncheck the My and Mz Releases of the member at its Start and press OK.

Member Properties			×
Model Section Properties Member Loads	Member Temperature Loads Member Distrortions		
General	Section Properties	Releases & Elastic Connection spring v	alues
Name : 54	Section : W10x45 W-AISC14 AISC 14th \sim	Quick Selection :	~
Level :	Ax: 0.0923611 Ix: 7.28202e-005	Start Spring End Sp	ring
	Ay: 0.0245486 Iy: 0.00257524		0
Type - Incidences	Az : 0.0460408 Iz : 0.0119598	□Fy 0 □Fy	0
Type : SPACE FRAME V	Sy: 0.00769676 Ey: 0	□Fz 0 □Fz	0
Start 34	Sz: 0.0284144 Ez: 0	Mx 0Mx	0
End : 35	Yd: 0.841667 Yc: 0.420833	□ My 0 ☑ My	0
Beta Angle : 90	Zd: 0.668333 Zc: 0.334167		0
Physical Member PM14 V	Shape Code : 1.0		
		End Sizes OR Member Eccentricities (C	
Groups	Material Properties		nd
Inactive Remove	Material : Steel \checkmark	Sizes : 0	0
PM14	E: 4.176e+006 Density 0.489543	x: 0 x:	0
	G: 1.58399e+006 CTE: 1.17e-005	Y: 0 Y:	0
	Poisson 0.3	Z: 0 Z:	0
		2	
		OK Cancel	<u>A</u> pply

No errors are reported now.

Select **member marked in C** by double clicking on it and check the My and Mz Releases of the member at its Start. Note that this member does not belong to any Physical Member. It was belonging to PM7, but when a part of it was deleted (Step #18) the PM7 definition was deleted as well. Press OK to close the properties form.

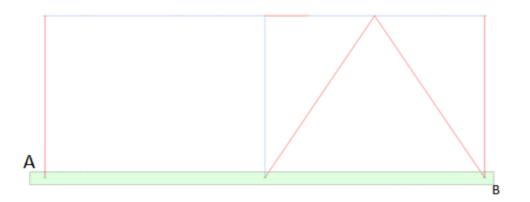
Member Properties			×
Model Section Properties Member Loads	Member Temperature Loads Member Distrortions		
General	Section Properties	Releases & Elastic Connection spring values	
Name : 51	Section : W10x45 W-AISC14 AISC 14th \checkmark	Quick Selection :	
Level : 1	Ax: 0.0923611 Ix: 7.28202e-005	Start Spring End Spring	
	Ay: 0.0245486 Iy: 0.00257524		D
Type - Incidences	Az : 0.0460408 Iz : 0.0119598	Fy 0 Fy	D
Type : SPACE FRAME V	Sy: 0.00769676 Ey: 0		D
Start 34	Sz: 0.0284144 Ez: 0		D
End : 18	Yd: 0.841667 Yc: 0.420833	Ø My 0 Ø My 0	D
Beta Angle : 90	Zd: 0.668333 Zc: 0.334167		0
Physical Member	Shape Code : 1.0		
Groups		End Sizes OR Member Eccentricities (Offsets) Start End	
Inactive Remove	Material Properties		0
Groups that Member belongs	Material : Steel 🗸 🗸 🗤	Sizes :	<u> </u>
	E: 4.176e+006 Density 0.489543	x: 0 x:	D
	G: 1.58399e+006 CTE: 1.17e-005	Y: 0 Y: 0	D
	Poisson 0.3	Z: 0 Z: 0	D
		OK Cancel A	pply



Click on the icon $\textcircled{\mbox{Clear}}$ to clear annotations.

5.7. Define supports

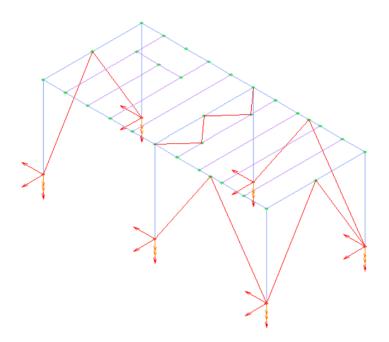
Step #25. Create supports: Use AutoCAD's Cube or BricsCAD's' Chair to change the view to Front. Press the icon ^{III} Support (in GTS Modeling Tab of the ribbon), select all Joints at the bottom, by clicking at points A and B of the following image, and press ENTER.



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

Joint Properties [Multiple Selection]		×
Model Joint Generalized Loads		
General Name :	Joint Coordinates Coordinates X :	
Rotation theta X : theta Y : theta Z :	Z : Restraints & Spring values Quick Selection :	
Groups Inactive Remove Groups that joint belongs	Fx Image: Mx Fy Image: My Fz Image: My	
	OK Cancel	Apply

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



Click on the icon 🙁 Clear to clear annotations and symbols.

5.8. Define Loads

Step #26. Change UCS: At the command window, type *UCS* and then give *w*, to change to World UCS.

Step #27. Define self-weight: Click on the icon Self Weight to define self-weight load case. Give a Name and a Description. Select Negative Z.

Make sure that the option "Include Finite Elements" is ON.

Self Weight or Dead Loads	×
Load Information	
Name : 1 Description : Self Weight	~
Create New Sa	ave / Modify Delete
Loads applied parallel to t	his Global Axis
O Negative Y	O Positive Y
Negative Z	O Positive Z
O Negative X	O Positive X
Load Information Name : Description : Self Weight Create New Save / Modify Delete Loads applied parallel to this Global Axis Negative Y Negative Y Negative Z Positive Z	
	OK Cancel

5.9. Perform analysis

Step #28. Overview the model and run checks.

Check for duplicate joints: Click on the icon ^{Joints Duplicates}, under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area. For the *Merge Tolerance* <0.001000>, just press <ENTER> to accept the default value. You should get the verification that:

0 duplicate joints found

Check for floating joints: In order to check for joints not connected to the model, click on the

icon Joints Floatings, under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area. If your model was created as described so far, you should get a notification:

0 floating joints found

Check Physical Members: Click on the icon , under the "Check" Drop Button, located in Find/Change/Check at Ribbon Area.). This check applies to Physical Members having two or more analytical members. The output should be (press F2 to extend the command line height and see all of it.

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

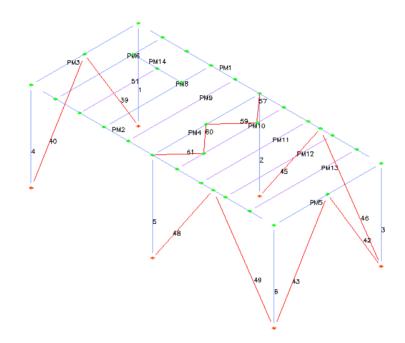
```
Physical Member PM14: OK
O Errors Found
```

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

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

Display Options		×
Visible Objects		Visible Labels
🗹 Joints		🗌 Joints
Members		Members
2D Elements		☑ 2D Elements
☑ 3D Elements		☑ 3D Elements
Label Settings - Font Sizes		Object Sizes
Joints :	0.820210	Joint : 0.164042
Members :	0.500000	Load Arrowhead (pts): 10.00000
2D Elements : 3D Elements :	0.820210	Display Members / Elements
Annotation (pts) : Annotation Format: Deci Decimal Places :	10.000000 mal ~ 2 ~	Shrink Factor : 0.9 Do Not Display Thickness in 3D Members As: Physical

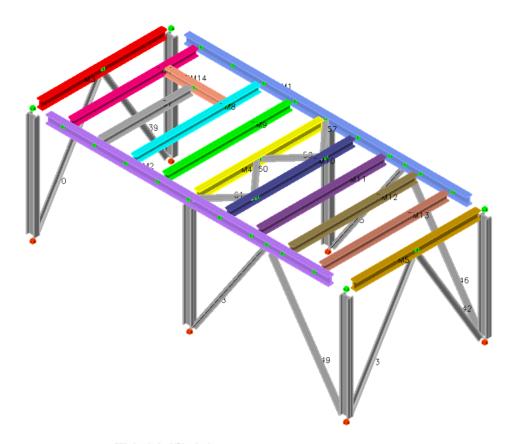
Press OK to close the form and to see the physical members in wireframe view.



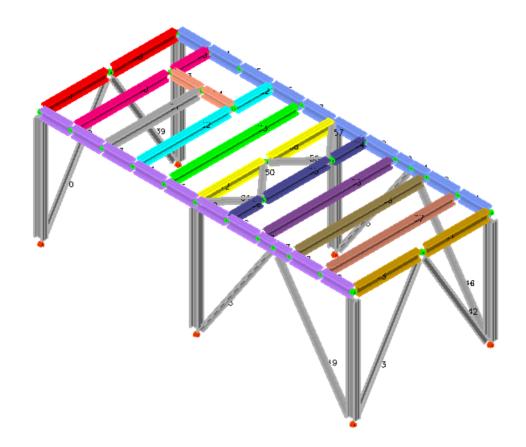
Click on the icon ^O Colors</sup> and select the 2nd Tab in order to colorize members by their group. Member do not belonging to any Physical Member are in the "Ungrouped data". Their default color is BYLAYER (256), meaning use the color from Display Options Form, but you can set any other color. Press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

Color Options			×	Display Options	×
Sections Groups				Visible Objects	Visible Labels
Categories				✓ Joints	Joints
Groups	Color	Visible	^		
PM4	50			Members	Members
PM5	42				
PM6	230	v		✓ 2D Elements	2D Elements
PM7	122			3D Elements	JD Elements
PM8	130	I			
PM9	3				
PM10	175				
PM11	205				
PM12	45				
PM13	23				
PM14	21				
UnGrouped data	256	>	~		
Note: If you set a color to 256 (BYLAYER) at View > Options Dialog If a member/shell/oint belongs to more the in this list is the one that defines its' color a Uncheck ALL Check ALL	an one groups, ti		s last		

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



By clicking on the icon Analytical/Physical, you can switch between physical view (image above) and analytical view (image below). Note the difference between the physical view, where each physical member is a single object, and the analytical view, where you can control each analytical member individually.



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

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

Press the icon ${\ensuremath{\mathbb A}}^{\ensuremath{\mathsf{Frame}}}$ to switch back to wireframe view.

Step #29. Create input file: Click on the icon **GTI** to create GTI input file. At the dialog box "Create GT.STRUDL Input File" check the following and click OK.

- Perform Stiffness Analysis
- Read Joint Displacements
- Read Member Forces
- Read Section Forces
- Read Section Displacements

Create GT.STRUDL Input File		×
Export to GTI		
Create GTI File : D:\GTSTRUDL\CAD_Mo	deler \Examples \Example3 \Example3.gti	
Export Whole Model	 Export Portion of the Model (will be prompted for Selection) 	
Perform Stiffness Analysis		
Append Other GTI Files/Macros		
		+
		-
		Up
Copy Commands from GTI Files/Macros		Down
Create Commands to Read Results		
Read Joint Displacements		
Read Member Forces		
Read Section Forces	Number of Sections:	10
Read Section Displacements	Number of Sections:	10
Read Finite Element Results		
Read Code Check Results		
	ОК Са	ancel

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

Example3_13	3.gti - Notepad		-	\times
File Edit Form	mat View Help			
STRUDL'' \$ \$ This \$\$	GTSTRUDL file created from GTS CAD	Modeler on Wednesday, 10 April, 2019		
ARGE PROBLEM				
JNITS FEET KI	PS DEGREES CENTIGRADE SECONDS			
JOINT COORDIN	ATES GLOBAL			
1	0.000000E+000 0.000000E+000			
2	0.0000000E+000 0.000000E+000			
3	2.0416700E+001 0.0000000E+000			
4	2.0416700E+001 0.0000000E+000			
5	4.0833399E+001 0.0000000E+000			
6	4.0833399E+001 0.0000000E+000			
7	0.000000E+000 -1.800000E+001			
8	0.0000000E+000 -1.8000000E+001 2.0416700E+001 -1.8000000E+001			
10	2.0416700E+001 -1.8000000E+001 2.0416700E+001 -1.8000000E+001			
10	4.0833399E+001 -1.8000000E+001			
12	4.0833399E+001 -1.8000000E+001 4.0833399E+001 -1.8000000E+001			
13	4.0833399E+000 0.000000E+000			
14	8.1666801E+000 0.0000000E+000			
15	1.2250020E+001 0.0000000E+000			
16	1.6333360E+001 0.0000000E+000			
17	4.0833399E+000 -1.8000000E+001			
18	8.1666801E+000 -1.8000000E+001			
19	1.2250020E+001 -1.8000000E+001			
20	1.6333360E+001 -1.8000000E+001	1.5000000E+001		
21	2.4500040E+001 0.0000000E+000	1.5000000E+001		
22	2.8583380E+001 0.0000000E+000	1.5000000E+001		
23	3.2666720E+001 0.0000000E+000	1.500000E+001		
24	3.6750059E+001 0.0000000E+000	1.5000000E+001		
25	2.4500040E+001 -1.8000000E+001	1.5000000E+001		
26	2.8583380E+001 -1.8000000E+001			
27	3.2666720E+001 -1.8000000E+001	1.5000000E+001		

6

Step #31. Execute GT.STRUDL: Click on the icon **GTS** to perform the stiffness analysis.

GT.STRUDL window will appear with information about analysis.

^a r _s GT STRUDL 2019 Beta (Not for commercial use)	-		×
File Edit Modeling Analysis Results SteelDesign RC_Design Tools View Help			
= 0.00 seconds.			^
<pre>{ 203) > { 204) > UNITS M RM { 205) > { 205) > { 206) > DEX BINARY 'D:\GTSTRUDL\CAD_Modeler\Examples\Example3.12' REPLACE { 207) > { 208) > WRITE REPLACE SECTION FORCES NS 10 AUTOMATIC 'D:\GTSTRUDL\CAD_Modeler\Examples\Examples\Example3.12' MEMBERS EXISTING</pre>			
<pre>**** INFO_WRSASF - SECTION FORCE records have been written to file D:biWhySBRCSTFRUDU.CAD_ModelerUExample3LExample3_13.12 for 60 members and 1 loads.</pre>			
**** INFO_WRSASF Time to write 628 SECTION FORCE records = 0.00 seconds.			
<pre>{ 209} > { 210} > UNITS M RM { 211) > { 212) > DEX BINARY 'D:\GTSTRUDL\CAD_Modeler\Examples\Example3\Example3.09' REFLACE { 212) > DEX BINARY 'D:\GTSTRUDL\CAD_Modeler\Examples\Example3.09' REFLACE { 213) > { 214) > WRITE REFLACE SECTION DISPLACEMENTS GLOBAL MS 10 'D\\GTSTRUDL\CAD Modeler\Examples\Examples\Example3\Example3.09' MEMBERS EXISTING</pre>			
****INFO_WRSASD SECTION DISFLACEMENT records have been written to file D:\GTSTRUDL\CAD_Modeler\Example3\Example3.09 for 60 members and 1 loads.			
****INFO_WRSASD Time to write 601 SECTION DISPLACEMENT records = 0.00 secs.			
<pre>{ 215) > { 216) > { 217) > { 216) > { 217) > { 216) > 216) > 216) > UNITS FEET KIPS DEGREES CENTIGRADE SECONDS { 216) > { 219) > { 219) > } </pre>			~
✓ <u>Command</u>			-
Ready Feet Kips Degrees Centigrade Seconds Apr	110, 2019	9 1:29 F	M /

5.10. Read analysis results

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

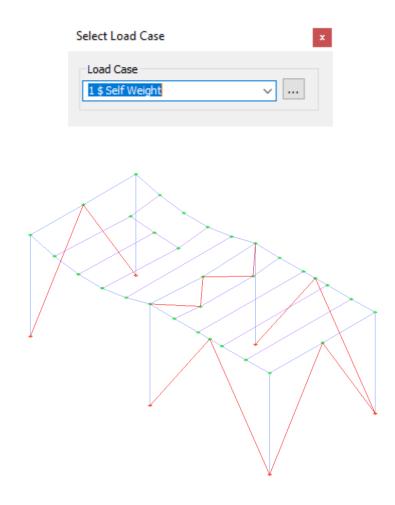
-Read DBX Result	LS		
GTI Directory :	D: \GTSTRUDL \CAD_Mod	deler\Examples\Example3	
🗹 Read Joint	Displacements		
Read Memb	per Forces		
Read Section	on Forces	Number of Sections:	10
Read Section	on Displacements	Number of Sections:	10
Read Finite	Element Results		
Type of ele	ement result :	<all></all>	\sim
Surface :		<all></all>	\sim
Read Code	Check Results		
GTI Commands			
	GT STRUDL commands:		
DBX BINARY WRITE REPLA WRITE REPLA WRITE REPLA			

: :_GTSResultsGTI Results Loaded Successfully :

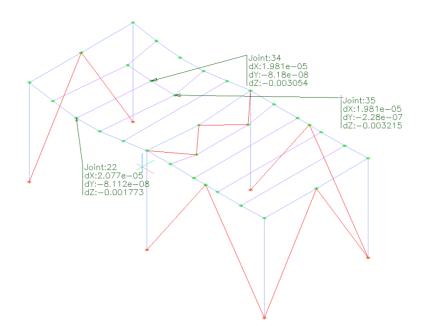
5.11. Display analysis results

:

Step #33. Graphical display of analysis results: Click on the icon ^{I/ Deformed}, under the "Deformation" Drop Button, located in "Display Results" at Ribbon Area, to display deformed shape of the model for Self Weight load case and press ENTER two times.



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



Click on the icon Clear to clear additional generated symbols. Click on the icon Undeformed to display undeformed model again.

Step #35. Display Section Displacements: Click on *P* Displacements (ribbon tab "GTS Display").

Displacem	ents ×
Load Case / Load Combin	nation
1 \$ Self Weight	~
Display Options Scale Factor (Values) :	0.100
Font Size (pt) :	10.00
Annotation Format:	Exponential 🗸
Decimal Places :	2 🗸
✓ Hide Model	
Display >	>
Annotate >	Legend >
Animation Options	
Frames :	7
Animation Speed % :	1 ~
Generate Animation Frames	Animate >>
Clear	Close

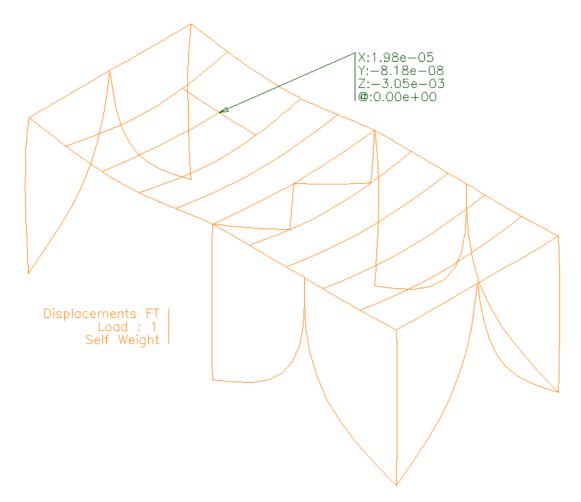
Select:

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

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

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

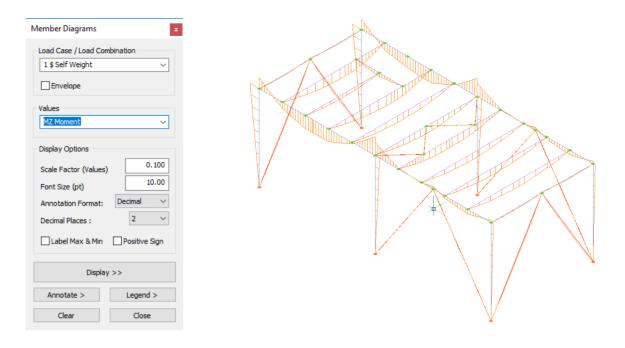
Press on "Legend >" and click at any part of the screen to place the legend of the diagram.



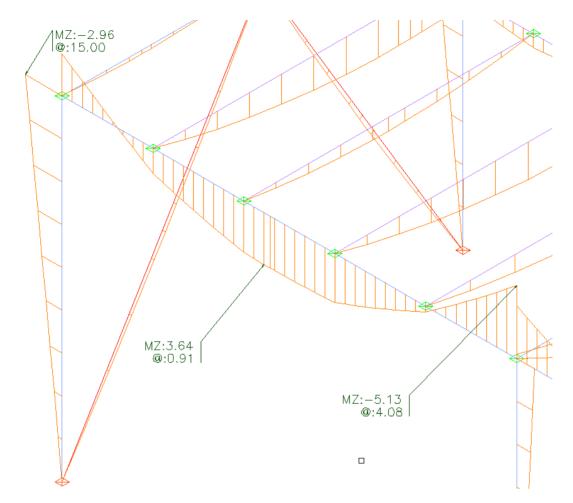
In order to display the animation, press "Generate Animation Frames" and then "Animate >>". To terminate the animation press "Stop" button.

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

Step #36. View member diagrams: Click on the icon ^{The Diagrams} to view member diagrams. In the dialog box "Member Diagrams", select *MZ Moment* as Values, *2* as Decimal Places and click on "Display".



You can also annotate moments at specific Joints by clicking "Annotate" in the dialog box "Member Diagrams" and select the desired precision (decimal places) as well as the annotation format (decimal, exponential, generic).



Select "Clear" and "Close" to clear the diagram/annotations and terminate the command.

Step #37. Display results in datasheets: Click on the icon **Displacements**, on Results Datasheets area, to view analysis results in datasheets.

e Edit	Columns	Filt	er Sort Units He	lp				
oint	Load		Trans X	Trans Y	Trans Z	Rotation X	Rotation Y	Rotation Z
1		1	0.0000	0.0000	0.0000	-0.0003399	-0.0048971	0.000000
2		1	0.0000	-0.0000	-0.0001	0.0006845	0.0107201	0.000026
3		1	0.0000	0.0000	0.0000	-0.0039857	0.0024911	0.000000
4		1	0.0000	-0.0000	-0.0001	0.0080282	-0.0051456	0.000035
5		1	0.0000	0.0000	0.0000	-0.0004337	0.0012434	0.000000
6		1	0.000	-0.0000	-0.0000	0.0008740	-0.0025264	0.000005
7		1	0.0000	0.0000	0.0000	0.0003399	-0.0050672	0.000000
8		1	0.0000	0.0000	-0.0001	-0.0006842	0.0111001	0.000014
9		1	0.0000	0.0000	0.0000	0.0039877	0.0025457	0.000000
10		1	0.0000	0.0000	-0.0001	-0.0080274	-0.0052484	0.0000022
11		1	0.0000	0.0000	0.0000	0.0004335	0.0012498	0.000000
12		1	0.0000	0.0000	-0.0000	-0.0008738	-0.0025280	0.000015
13		1	0.000	0.0000	-0.0011	0.0021532	0.0120130	-0.000000
14		1	0.0000	-0.0000	-0.0017	0.0036220	0.0044458	-0.0000022
15		1	0.000	-0.0000	-0.0017	0.0050907	-0.0056013	-0.000008
16		1	0.000	-0.0000	-0.0009	0.0065595	-0.0117479	-0.000049
17		1	0.0000	0.0000	-0.0001	0.0065356	0.0006018	0.000072
18		1	0.000	0.0000	-0.0001	0.0050429	0.0007610	-0.000000
19		1	0.000	0.0000	-0.0002	0.0036121	0.0013782	-0.000000
20		1	0.0000	0.0000	-0.0003	0.0022431	-0.0015496	-0.0000012
21		1	0.000	0.0000	-0.0011	-0.0021529	0.0126868	-0.0000010
22		1	0.000	-0.0000	-0.0018	-0.0036215	0.0045080	-0.000018
23		1	0.0000	-0.0000	-0.0017	-0.0050902	-0.0061676	-0.000028
24		1	0.000	-0.0000	-0.0009	-0.0065588	-0.0120713	0.000048
25		1	0.000	0.0000	-0.0001	-0.0065349	0.0005932	-0.000001
26		1	0.000	0.0000	-0.0001	-0.0050424	0.0007877	0.000000
27		1	0.000	0.0000	-0.0002	-0.0036117	0.0013902	-0.0000014
28		1	0.000	0.0000	-0.0003	-0.0022428	-0.0015499	-0.000004
29		1	0.000	-0.0000	-0.0001	0.000001	0.0106267	0.00005
30		1	0.000	0.0000	-0.0001	0.000001	-0.0024599	0.000042
31		1	0.0000	0.0000	-0.0002	0.0042966	0.0012625	-0.000000
32		1	0.0000	0.0000	-0.0002	-0.0042961	0.0012849	-0.000000
33		1	0.0000	0.0000	-0.0026	0.0116327	0.0122002	-0.000086
34		1	0.0000	-0.0000	-0.0031	0.0116465	0.0042854	-0.0000017
35		1	0.0000	-0.0000	-0.0032	0.0116604	-0.0057586	0.0000170
36		1	0.0000	0.0000	-0.0012	0.0107657	-0.0003520	0.0000168
37		1	0.0000	0.0000	-0.0012	-0.0107855	-0.0003590	-0.000098
38		1	0.0000	-0.0000	-0.0010	0.000003	-0.0037684	0.000039

Click on the icon Member Forces to view member forces results in datasheets.

									_
lember	Load	Joint	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z	
1	1	1	2.1786	-0.1908	0.0018	-0.0000	-0.0000	-0.0001	
1	1	2	-1.4341	0.1908	-0.0018	0.000	-0.0270	-2.8618	
2	1	3	3.6514	0.0933	0.0211	-0.0000	-0.0000	0.0000	
2	1	4	-2.9068	-0.0933	-0.0211	0.0000	-0.3170	1.3994	
3	1	5	1.4975	0.0461	0.0023	-0.0000	-0.0000	0.0000	
3	1	6	-0.7530	-0.0461	-0.0023	0.0000	-0.0345	0.6908	
4	1	7	2.1796	-0.1975	-0.0018	-0.0000	-0.0000	-0.0001	
4	1	8	-1.4351	0.1975	0.0018	0.0000	0.0270	-2.9626	
5	1	9	3.6544	0.0952	-0.0211	-0.0000	0.0000	-0.0000	
5	1	10	-2.9099	-0.0952	0.0211	0.0000	0.3171	1.4282	
6	1	11	1.4984	0.0462	-0.0023	-0.0000	0.0000	0.0000	
6	1	12	-0.7539	-0.0462	0.0023	0.000	0.0345	0.6923	
7	1	2	0.1908	1.2561	-0.0000	-0.0008	0.0002	2.8618	
7	1	13	-0.1908	-1.0535	0.0000	0.0008	-0.0002	1.8536	
14	1	13	0.1906	0.4071	-0.0000	-0.0008	0.0002	-1.8537	
14	1	14	-0.1906	-0.2044	0.0000	0.0008	0.0000	3.1021	
15	1	14	0.1906	0.2044	-0.0000	-0.0008	-0.0000	-3.1021	
15	1	15	-0.1906	-0.0017	0.0000	0.0008	0.0002	3.5230	
16	1	15	0.1908	-0.6447	0.0002	-0.0008	-0.0002	-3.5229	
16	1	16	-0.1908	0.8474	-0.0002	0.0008	-0.0006	0.4767	
17	1	16	0.1908	-1.2543	-0.0005	-0.0008	0.0006	-0.4766	
17	1	4	-0.1908	1.4570	0.0005	0.0008	0.0014	-5.0588	
8	1	4	0.0975	0.9123	0.0005	0.0008	-0.0011	3.6590	
8	1	17	-0.0975	-0.7096	-0.0005	-0.0008	-0.0008	-0.3476	
18	1	17	0.0976	0.2118	-0.0002	0.0008	0.0008	0.3481	
18	1	18	-0.0976	-0.0091	0.0002	-0.0008	0.0000	0.1030	
20	1	19	0.0460	0.2453	-0.0000	0.0008	0.0000	-0.6225	
20	1	20	-0.0460	-0.0427	0.000	-0.0008	0.0000	1.2104	
21	1	20	0.0460	-0.3643	-0.0000	0.0008	-0.0000	-1.2104	
21	1	6	-0.0460	0.5669	0.0000	-0.0008	0.0001	-0.6908	
9	1	8	0.1975	1.2571	-0.0000	0.0008	0.0002	2.9627	
9	1	21	-0.1975	-1.0545	0.0000	-0.0008	-0.0000	1.7568	

Click on the icon Section Forces to view section forces results in datasheets.

e Edit (Columns F	ilter Sort l	Jnits Help						
lember	Load	Section	Axial	Y shear	Z shear	Torsion	Y bending	Z bending	
1	1	0.000	-2.1786	0.1908	-0.0018	0.0000	0.0000	0.0001	
1	1	0.111	-2.0959	0.1908	-0.0018	0.0000	-0.0030	-0.3179	
1	1	0.222	-2.0132	0.1908	-0.0018	0.0000	-0.0060	-0.6359	
1	1	0.333	-1.9304	0.1908	-0.0018	0.0000	-0.0090	-0.9539	
1	1	0.444	-1.8477	0.1908	-0.0018	0.0000	-0.0120	-1.2719	
1	1	0.556	-1.7650	0.1908	-0.0018	0.0000	-0.0150	-1.5899	
1	1	0.667	-1.6823	0.1908	-0.0018	0.0000	-0.0180	-1.9079	
1	1	0.778	-1.5995	0.1908	-0.0018	0.0000	-0.0210	-2.2258	
1	1	0.889	-1.5168	0.1908	-0.0018	0.0000	-0.0240	-2.5438	
1	1	1.000	-1.4341	0.1908	-0.0018	0.0000	-0.0270	-2.8618	
2	1	0.000	-3.6514	-0.0933	-0.0211	0.0000	0.0000	-0.0000	
2	1	0.111	-3.5686	-0.0933	-0.0211	0.0000	-0.0352	0.1555	
2	1	0.222	-3.4859	-0.0933	-0.0211	0.0000	-0.0704	0.3110	
2	1	0.333	-3,4032	-0.0933	-0.0211	0.0000	-0.1056	0.4665	
2	1	0.444	-3,3205	-0.0933	-0.0211	0.0000	-0.1409	0.6219	
2	1	0.556	-3.2377	-0.0933	-0.0211	0.0000	-0.1761	0.7774	
2	1	0.667	-3,1550	-0.0933	-0.0211	0.0000	-0.2113	0,9329	
2	1	0.778	-3.0723	-0.0933	-0.0211	0.0000	-0.2466	1.0884	
2	1	0.889	-2.9896	-0.0933	-0.0211	0.0000	-0.2818	1.2439	
2	1	1,000	-2.9068	-0.0933	-0.0211	0.0000	-0.3170	1.3994	
3	1	0.000	-1.4975	-0.0461	-0.0023	0.0000	0.0000	-0.0000	
3	1	0.111	-1.4148	-0.0461	-0.0023	0.0000	-0.0038	0.0767	
3	1	0.222	-1.3321	-0.0461	-0.0023	0.0000	-0.0077	0.1535	
3	1	0.333	-1.2494	-0.0461	-0.0023	0.0000	-0.0115	0.2303	
3	1	0.444	-1.1666	-0.0461	-0.0023	0.0000	-0.0153	0.3070	
3	1	0.556	-1.0839	-0.0461	-0.0023	0.0000	-0.0192	0.3838	
3	1	0.667	-1.0012	-0.0461	-0.0023	0.0000	-0.0230	0.4605	
3	1		-0.9185	-0.0461	-0.0023	0.0000	-0.0269	0.5373	
3	1	0.889	-0.8357	-0.0461	-0.0023	0.0000	-0.0307	0.6141	
3	1		-0.7530	-0.0461	-0.0023	0.0000	-0.0345	0.6908	
4	1		-2.1796	0.1975	0.0018	0.0000	0.0000	0.0001	
4	1		-2.0969	0.1975	0.0018	0.0000	0.0030	-0.3291	
4	1		-2.0141	0.1975	0.0018	0.0000	0.0060	-0.6583	

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

E	GTSTRUDL - Joint Reactions Datasheet, display Units: Feet Kips						_ 🗆 ×
File Edit (Columns Fi	lter Sort Units He	lp				
Joint	Load	Force X	Force Y	Force Z	Moment X	Moment Y	Moment Z
1	1	0.191	-0.242	2.709	-0.000	0.000	0.000
3	1	0.400	-0.021	4.511	-0.000	-0.000	-0.00
5	1	-0.590	-0.237	2.955	-0.000	-0.000	0.00
7	1	0.198	0.242	2.710	0.000	-0.000	-0.00
9	1	0.393	0.021	4.508	0.000	-0.000	0.00
11	1	-0.591	0.237	2.957	0.000	-0.000	-0.00

Step #37. Display results in Report Builder: Click on the icon to view results in Report Builder.



											- 0	- ×
Home											S	ityle *
🖌 🛐 🔣 🚮 🖬		Joint	Name: ALL			Loading	ALL				@.00	(?)
		Memi	er Name: ALL			Level:	ALL			V		· · ·
Open Generate Image Text RTF		rint Eleme	nt Name ALL			Surface	ALL			oply Filters to	Format	Help
File Overall Report File File Insert	Header and Footer Print	creme	in the sec			Filters				hecked Items	Options Format	Help
						Tillers					Torniat	Tiel
GTSTRUDL Reports	An alteria Descrita											
⊡ Model Data	Analysis Results											
Groups												
Joint Coordinates	Joint Displacements											
Joint Coordinates	Some Dispincements											
Member Incidences												
Member Properties	Length: FEET, Force: KIP, Angle:	DEG, Tempe	rature: DEGC , 1	Time: SEC								
Member/Element Constants												
Element Incidences		Joint	Load	Trans X	Trans Y	Trans Z	Rotation X	Rotation Y	Rotation Z			
Element Properties		1	1	0.0000000	0.0000000	0.0000000	-0.0003399	-0.0048971	0.0000000			
Load Data		2	1	0.0000231	-0.0000000	-0.0000640	0.0006845	0.0107201	0.0000026			
Summary of Loadings		3	1	0.0000000	0.0000000	0.0000000	-0.0039857	0.0024911	0.0000000			
Loading Combinations		4	1	0.0000139	-0.0000006	-0.0001162	0.0080282	-0.0051456	0.0000039			
Joint Loads		5	1	0.0000000	0.0000000	0.0000000	-0.0004337	0.0012434	0.0000000			
Member Loads		6	1	0.0000105 0.0000000	-0.0000000	-0.0000399 0.0000000	0.0008740 0.0003399	-0.0025264 -0.0050672	0.0000005			
Element Loads		8	1	0.0000246	0.0000000	-0.0000640	-0.0006842	0.0111001	0.0000014			
Analysis Results		9	1	0.0000000	0.0000000	0.0000000	0.0039877	0.0025457	0.0000000			
Joint Displacements		10	1	0.0000151	0.0000002	-0.0001163	-0.0080274	-0.0052484	0.0000022			
Support Joint Reactions		11	1	0.0000000	0.0000000	0.0000000	0.0004335	0.0012498	0.0000000			
		12	1	0.0000115	0.0000001	-0.0000399	-0.0008738	-0.0025280	0.0000019			
Section Forces		13	1	0.0000213	0.0000000	-0.0010522	0.0021532	0.0120130	-0.0000007			
Average Element Results		14	1	0.0000194	-0.0000001	-0.0016859	0.0036220	0.0044458	-0.0000022			
Member Results Graphs		15	1	0.0000176	-0.0000002	-0.0016606	0.0050907	-0.0056013	-0.0000005			
Design of Steel Members		16	1	0.0000158 0.0000130	-0.0000003 0.0000000	-0.0009273 -0.0000794	0.0065595 0.0065356	-0.0117479 0.0006018	-0.0000049 0.0000072			
		18	1	0.0000130	0.0000000	-0.0001450	0.0050429	0.0007610	-0.0000009			
		19	1	0.00001120	0.0000001	-0.0002408	0.0036121	0.0013782	-0.0000009			
		20	1	0.0000109	0.0000000	-0.0002569	0.0022431	-0.0015496	-0.0000012			
		21	1	0.0000227	0.0000000	-0.0010900	-0.0021529	0.0126868	-0.0000010			
		22	1	0.0000208	-0.0000001	-0.0017728	-0.0036215	0.0045080	-0.0000018			
		23	1	0.0000189	-0.0000002	-0.0017071	-0.0050902	-0.0061676	-0.0000028			
	1	24	1	0.0000170	-0.0000003	-0.0009423	-0.0065588	-0.0120713	0.0000048			
			1 1	0.0000141	0.0000001	-0.0000763	-0.0065349	0.0005932	-0.0000016			
		25	-	0.0000121								
		26	1	0.0000131	0.0000001	-0.0001433	-0.0050424	0.0007877	0.0000009			
			1 1	0.0000131 0.0000124 0.0000119	0.0000001 0.0000001 0.0000000	-0.0001433 -0.0002408 -0.0002572	-0.0050424 -0.0036117 -0.0022428	0.0007877 0.0013902 -0.0015499	0.0000009 -0.0000014 -0.0000004			

6. Appendix – List of Commands

Command	lcon	Menu	Command Prompt	Link
Units	🗶 Units	GTS Modeling>Units	GTSUnits	<u>2.6.1</u>
Materials	Materials	GTS Modeling>Materials	GTSMaterials	<u>2.6.2</u>
Sections	Sections	GTS Modeling>Cross Sections>Table	GTSParams	<u>2.6.9</u>
Prismatic Sections	-	GTS Modeling>Cross Sections>Prismatic	GTSPrismatic	2.6.9
Levels	Levels	GTS Modeling>Levels	GTSLevels	<u>2.6.3</u>
Higher Level	Higher Level	-	GTSLevelUp	<u>2.6.3</u>
Lower Level	Lower Level	-	GTSLevelDown	<u>2.6.3</u>
All Levels ON	🞒 All Levels ON	GTS Display>All Levels ON	GTSSetAllVisible	<u>2.6.3</u>
Grid	Grid	GTS Modeling>Grid>Create	GTSGrid	<u>2.6.4</u>
Change Grid	🗱 Change Grid	GTS Modeling>Grid>Change	GTSGridChange	<u>2.6.4</u>
Generate Joint	Generate	GTS Modeling>Joint>Generate Joint	GTSJoint	<u>2.6.5</u>
At Level (Joint)	🕤 At Level	GTS Modeling>Joint>Generate Joint at Level	GTSJointLevel	<u>2.6.5</u>
Find (Joint)	🔒 Find Joint	GTS Modeling>Joint>Find	GTSFJID	<u>2.6.6</u>
Support	💻 Support	GTS Modeling>Joint>Support	GTSJointSupport	<u>2.6.7</u>
Change (Joint)	🌯 Change Joint	GTS Modeling>Joint>Change	GTSJointChange	<u>2.6.8</u>
Generate (Member)	Generate	GTS Modeling> Members>Generate Beam Members	GTSBeam	<u>2.6.10</u>

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

Physical Members	Physical Members	GTS Modeling>Checks>Physical Members	GTSCheckPhysicalMembers	<u>2.6.26</u>
Shells Duplicates	Shells Duplicates	GTS Modeling>Checks>Shells Duplicates	GTSCheckDuplicateShells	<u>2.6.27</u>
Names Duplicates	Names Duplicates	GTS Modeling>Checks>Names Duplicates	GTSCheckNames	<u>2.6.28</u>
Renumber Names	6.2.8.1 1.2.6.9 Renumber Names	GTS Modeling>Checks>Renumber Names	GTSRenumber	<u>2.6.29</u>
Database Integrity	Database Integrity	GTS Modeling>Checks>Database Integrity	GTSCheckDatabase	<u>2.6.30</u>
1D Curve (Meshing)	1D Curve	GTS Modeling>Mesh Generation>1D Along Line or Curve or Circle	GTSMesh1D	<u>2.6.31</u>
2D 2Curves (Meshing)	2D 2Curves	GTS Modeling>Mesh Generation>2D Between 2 Lines or Curves	GTSMesh2D2L	<u>2.6.32</u>
2D 4Curves (Meshing)	2D 4Curves	GTS Modeling>Mesh Generation>2D Between 4 Lines or Curves	GTSMesh2D4L	<u>2.6.33</u>
2D Area (Meshing)	2D Area	GTS Modeling>Mesh Generation>2D Between 4 Lines or Curves	GTSMesh2DPoly	<u>2.6.34</u>
3D Extrude (Meshing)	3D Extrude	GTS Modeling>Mesh Generation>3D Extrude PolyLine	GTSExtrudePoly	<u>2.6.35</u>
2D 3Curves (Meshing)	3D 3Curves	GTS Modeling>Mesh Generation>3D Between 3 Lines or Curves	GTSMesh3D3L	<u>2.6.36</u>
Array 3D Advanced	Array 3D Advanced	-	GTSArray3D	<u>2.6.37</u>
Soil Springs	Soil Springs	GTS Modeling>Soil Springs	GTSFoundationSprings	<u>2.6.38</u>
Export STR	C2	GTS Modeling> CAESAR II>Export STR	GTSExportSTR	<u>2.6.39</u>
Convert Lines/Polylines to Members/Shells	DXF Convert Lines to Members	GTS Modeling> Convert Lines/Polylines to Members/Shells	GTSDXFRead	<u>2.6.40</u>

Model Wizard	Model Wizard	GTS Modeling>Model Wizard	GTSModelWizard	<u>2.6.41</u>
List (Group)	List	GTS Modeling>Groups>Manage	GTSGroups	<u>2.6.42</u>
+Joints (Group)	+Joints	GTS Modeling>Groups>Add Joints	GTSGroupJoints	<u>2.6.42</u>
+Members (Group)	+Members	GTS Modeling>Groups>Add Members	GTSGroupMembers	<u>2.6.42</u>
+Shells (Group)	+Shells	GTS Modeling>Groups>Add Shells	GTSGroupShells	<u>2.6.42</u>
Self Weight	🕮 Self Weight	GTS Modeling>Loads>Self Weight	GTSSelfWeight	<u>2.6.43</u>
Load Cases	📑 Load Cases	GTS Modeling>Loads>Load Cases	GTSNewLoadCase	<u>2.6.44</u>
Load Combinations	Combinations	GTS Modeling>Loads>Load Combinations	GTSLoadCombination	<u>2.6.52</u>
Standardized Combinations	Standardized Combinations	GTS Modeling>Loads>Standardiz ed Combinations	GTSLoadCombinationStandardized	<u>2.6.53</u>
Joint Load	Joint Load	GTS Modeling>Loads>Joint Load	GTSJointLoad	<u>2.6.45</u>
Member Load	↓↓ Member Load	GTS Modeling>Loads>Member Load	GTSBeamLoad	<u>2.6.46</u>
Shell Load	Shell Load	GTS Modeling>Loads>Shell Load	GTSShellLoad	<u>2.6.47</u>
Area Load	📕 Area Load	GTS Modeling>Loads>Area Load	GTSAreaLoad	<u>2.6.48</u>
Wind Load ASCE 705	ार्क्स Wind Load ASCE 705	GTS Modeling>Loads>Wind Load ASCE 705	GTSWindLoadsASCE705	<u>2.6.49</u>
Wind Load ASCE 710	Wind Load ASCE 710	GTS Modeling>Loads>Wind Load ASCE 710	GTSWindLoadsASCE710	<u>2.6.50</u>
Seismic Load		GTS Modeling>Loads>Seismic Load	GTSSeismicLoading	<u>2.6.51</u>
Steel Design Parameters		GTS Modeling>Steel Design Parameters	GTSSteelDesignParameters	<u>2.6.54</u>
Create Input File	GT	GTS Modeling>Create Input File	GTSExportGTI	<u>2.6.55</u>
Edit GTI	🗎 Edit GTI	GTS Modeling>Edit GTI	GTSEditGTI	<u>2.6.56</u>

Execute GT STRUDL	GTS	GTS Modeling>Execute	GTSExecuteGTI	<u>2.6.57</u>
Read Results	Read Results	GTS Modeling>Read Results	GTSResultsGTI	<u>2.6.58</u>
Import GTI	强 Import GTI		GTSGTIRead	<u>2.6.59</u>
Set View	Set View	GTS Display>Set View	GTSSetView	<u>2.6.60</u>
3D View	30 3D	GTS Display>3D Sections	GTSSet3D	<u>2.6.61</u>
Analytical/Physical	🚆 Analytical/Physical	GTS Display>Analytical/Physical	GTSDisplayPhysicalMembers	<u>2.6.62</u>
Frame View	Frame	GTS Display>Frame	GTSSet1D	<u>2.6.61</u>
Options (View)	Options	GTS Display>Options	GTSDisplay	<u>2.6.64</u>
Colors	O Colors	GTS Display>Colors	GTSColorView	<u>2.6.63</u>
Annotate	🛃 Annotate	GTS Display>Annotate	GTSAnnotate	2.6.65
Select	🔍 Select	-	GTSSelect	<u>2.6.66</u>
Display Member Local Axes	➤ Member Local Axes	GTS Display>Member Local Axes	GTSDisplayLocalAxes	<u>2.6.67</u>
Display Member Releases	🖊 Releases	GTS Display> Member Releases	GTSDisplayReleases	<u>2.6.68</u>
Display Shell Planar Axes	💾 Shell Planar Axes	GTS Display> Shell Planar Axes	GTSDisplayPlanarAxes	<u>2.6.69</u>
Display Joint Supports	🛓 Joint Supports	GTS Display> Joint Supports	GTSDisplaySupports	<u>2.6.70</u>
Display Joint Loads	🋓 Joint Loads	GTS Display>Joint Loads	GTSDisplayJointLoads	<u>2.6.71</u>
Display Member Loads	Hember Loads	GTS Display>Member Loads	GTSDisplayMemberLoads	<u>2.6.72</u>
Display Shell Loads	📕 Area Loads	GTS Display>Shell Loads	GTSDisplayElementLoads	<u>2.6.73</u>
Display Area Loads	💆 Shell Loads	GTS Display>Area Loads	GTSDisplayAreaLoads	<u>2.6.74</u>
Deformed Structure	/ Deformed	GTS Display>Deformed Structure	GTSDisplayJointDisplacements	<u>2.6.75</u>
Undeformed Structure	Undeformed	GTS Display> Udeformed Structure	GTSDisplayJointDisplacements	<u>2.6.75</u>
Annotate Joint Displacements	Annotate Displacements		GTSAnnotateJointDisplacements	<u>2.6.76</u>

Displacements	Displacements	GTS Display> Displacements	GTSDisplaySectionDisplacements	2.6.77
Member Diagrams	🐨 Diagrams	GTS Display>Member Diagrams	GTSDisplayMemberForces	<u>2.6.78</u>
Finite Element Results	🎽 Elements	GTS Display>Element Results	GTSDisplayElementResults	<u>2.6.79</u>
Finite Element Results Selection	🎽 Selection	GTS Display>Element Results Selection	GTSDisplayElementResultsSel	<u>2.6.80</u>
Member Code Check Results	🔗 Code Check	GTS Display>Member Code Check Results	GTSColorCodeCheck	<u>2.6.81</u>
Displacements Datasheets	Displacements	GTS Display>Results Datasheets> Displacements	GTSDataSheetJointDisp	<u>2.6.82</u>
Member Forces Datasheets	Member Forces	GTS Display>Results Datasheets> Displacements	GTSDataSheetMemberForces	<u>2.6.82</u>
Section Forces Datasheets	Section Forces	GTS Display>Results Datasheets> Displacements	GTSDataSheetSectionForces	<u>2.6.82</u>
Reactions Datasheets	Reactions	GTS Display>Results Datasheets> Displacements	GTSDataSheetReactions	<u>2.6.82</u>
Stresses Datasheets	Stresses	GTS Display>Results Datasheets> Displacements	GTSDataSheetStresses	<u>2.6.82</u>
Code Check Datasheets	Code Check	GTS Display>Results Datasheets> Displacements	GTSDataSheetCodeCheck	<u>2.6.82</u>
Report Builder		-	GTSReportBuilder	<u>2.6.83</u>
Clear Results	🗷 Clear	GTS Display>Clear Results Layer	GTSDisplayResultsClear	<u>2.6.84</u>
Current Version	-	GTS Display>Version	GTSVersion	<u>2.6.85</u>