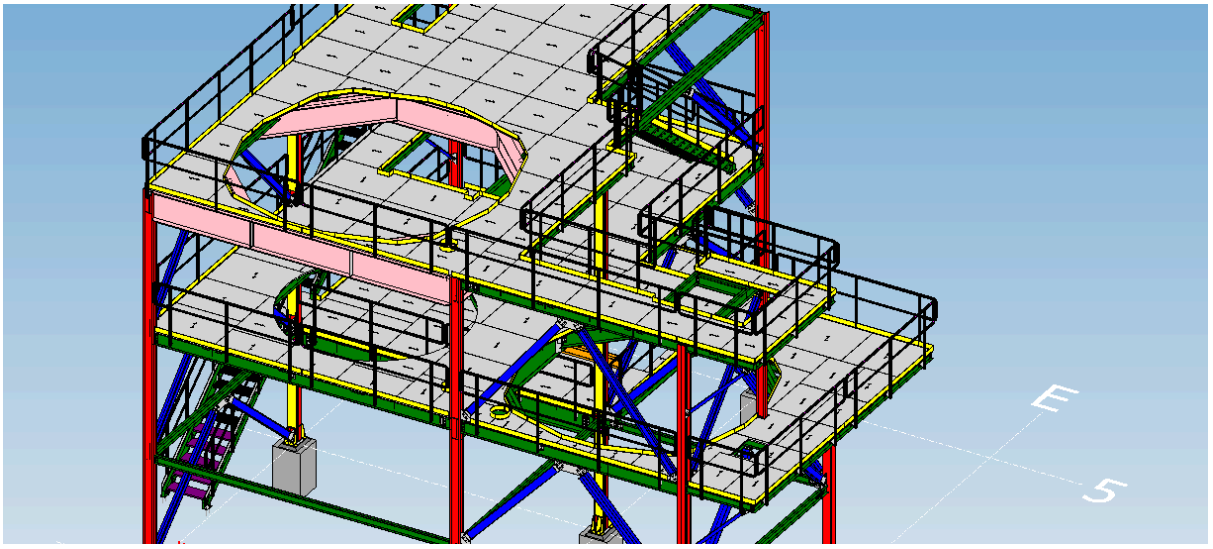
TSteel 3D - User Manual

Version: 2024-01-25

Link to Google Docs online version



Installation

How to download and install

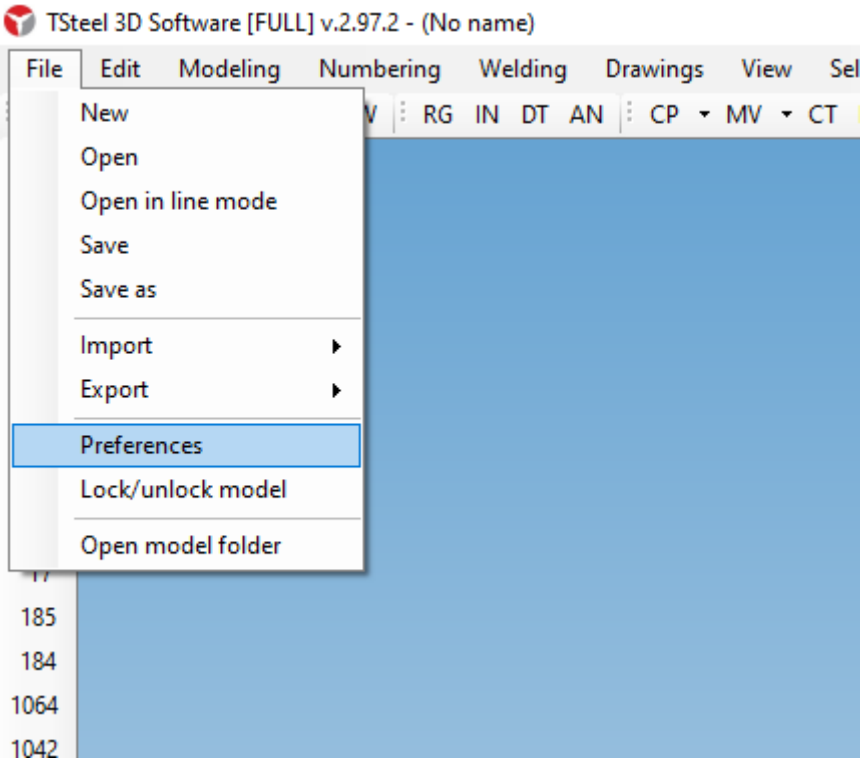
To download the installer, follow these steps:

1. Go to the TSteel website (www.tsteel3d.com), find the download form and fill in your details
2. You will receive an email with the download link
3. The download will download an executable program which is the installer. Depending on the security level of your windows, there may be warning messages and confirmation requests. TSteel is still a small piece of software and not recognized by Microsoft.
4. Follow the installer's steps and you're done!

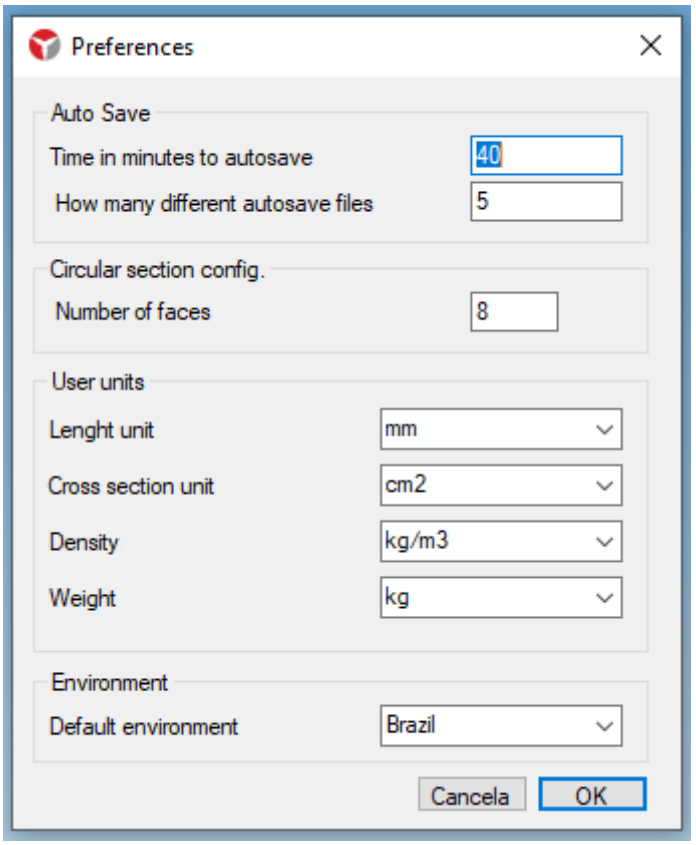
O TSteel,**when installed on a computer that it does not yet know**, releases a trial version (TRIAL), valid for 30 days. If your computer has already used the trial period previously, you will need to obtain a usage license.

Configure the work environment

When TSteel runs for the first time it will ask you which work environment you prefer. But this can be changed whenever necessary in the preferences menu:



The preferences window shown will be:



where you can change drives and working environment.

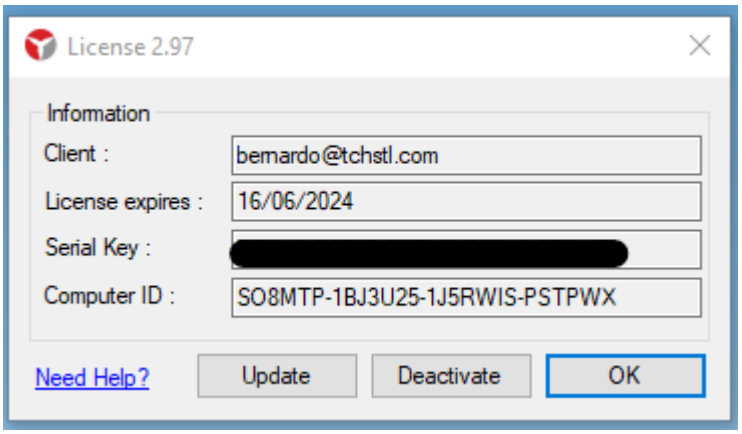
The work environment determines when creating a new model:

1. Grid inicial
2. Units of measurement
3. Profile catalogs

TSteel Licenses

There is only one program installer, and what gives access to the functions is the usage license. This license (**Serial Key**) is an alphanumeric code sent by email from the license server.

In the License menu, you will have access to the license screen:



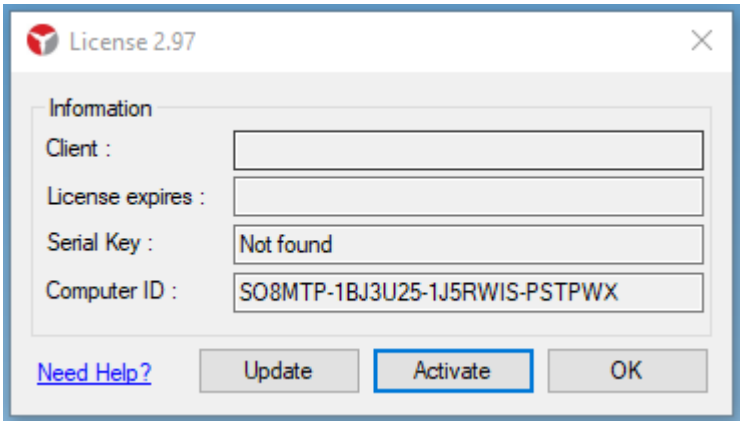
How to get your license

The types of licenses available and how to purchase them can be found on our website athttps://tsteel3d.com/licencas-e-precos/

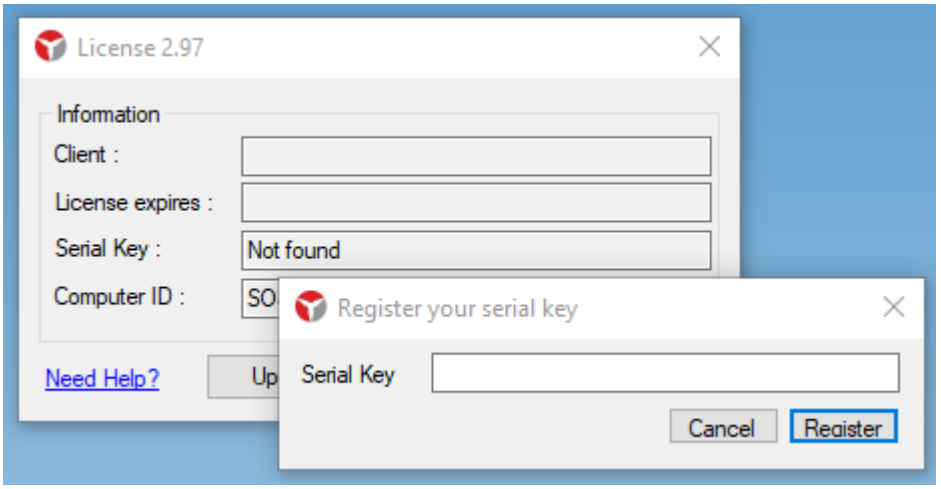
How to install your license

If your TRIAL license has already expired, TSteel will take you directly to the license window. Another way to access this window is through the “License” menu.

License window without a license installed:



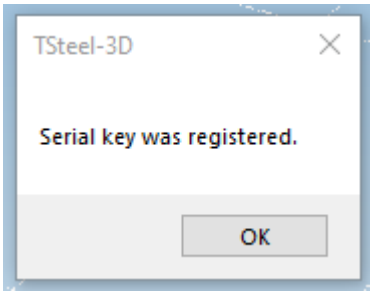
Click “Activate” to activate the license (Serial Key) you received by email.



Enter the license number and click Register.

TSteel will access the online license server, check the license number and register the activation on your machine. Therefore, during this process, internet access is necessary.

If everything went well, you will receive the message:



Ready! Log out of TSteel and log in again to update the activated license permissions.

What can go wrong

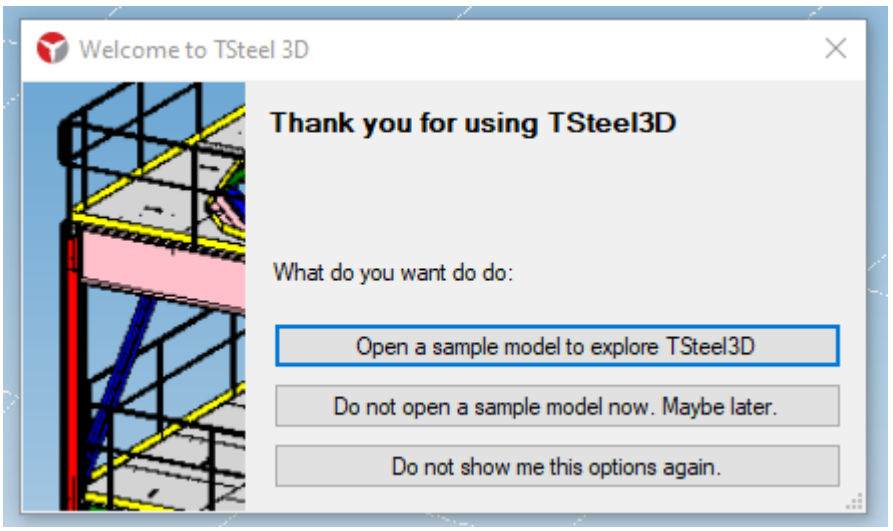
The most common errors when activating a license are:

1. License server access error. This may be due to a lack of internet connection or there is a blockage that prevents this access. This block can be a firewall or antivirus.
2. The license is invalid. Most likely, there was a typing error, or that the copy/paste operation included or suppressed something other than the license digits.
3. License notice already in use. The most common cause is that the user forgot to deactivate the license on another computer before activating it again. It could also be a writing error in the computer's license file (power outage, computer crashed, turning off the computer without leaving TSteel, etc...)

3d modeling

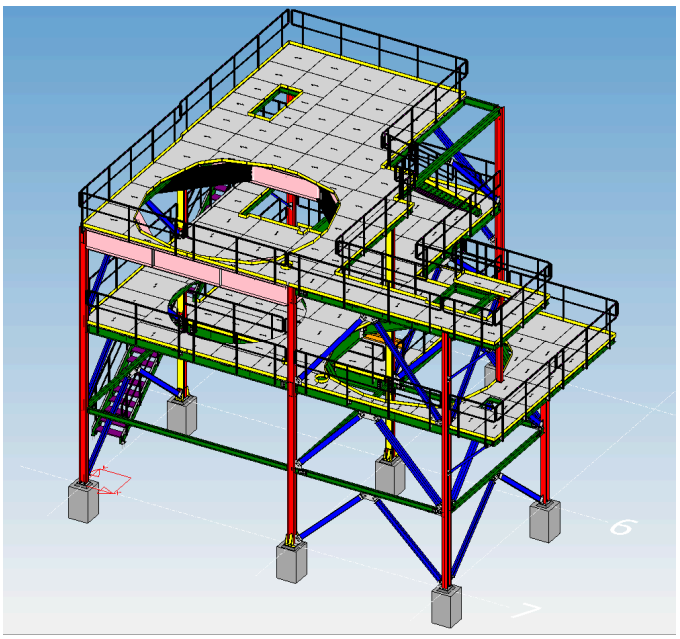
Open the example template

The first time you run TSteel, it offers you to view an example model:



Even if you have already disabled this window, the example model should be available at C:\TSteel3D\Samples\SampleModel.

The example model is as follows:



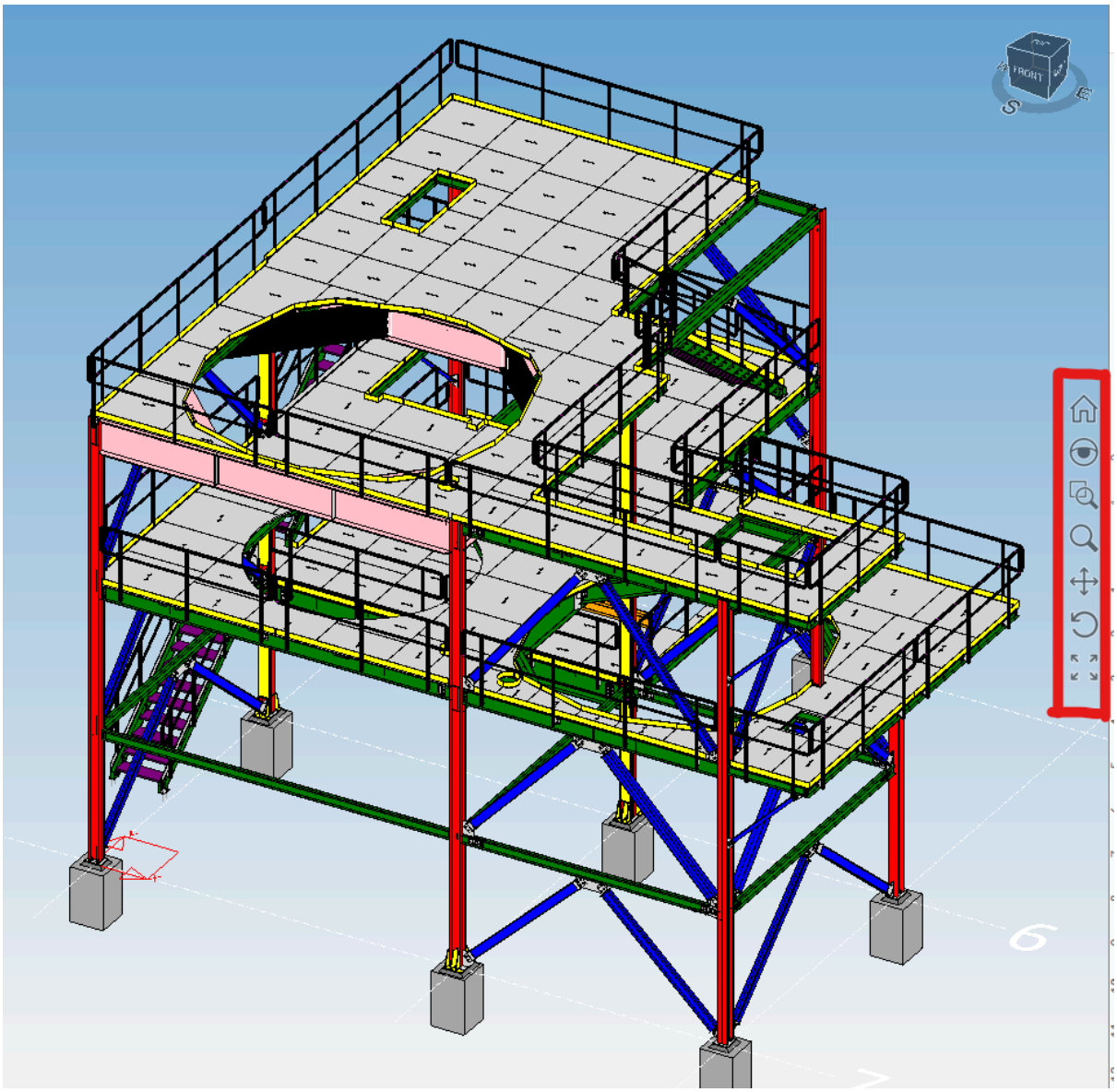
Let's spend a little time here on this model and see TSteel at work.

Mouse operation

1. With the mouse scroll, do**zoom** to see closer and further. Note that it makes a difference where the mouse pointer is when you rotate the scroll.
2. Hold down the mouse “scroll” and move (PAN) your model in any direction
3. With CTRL pressed, keep the mouse scroll pressed and rotate the model. Note that it makes a difference where you start the rotation movement

Screen menu for viewing

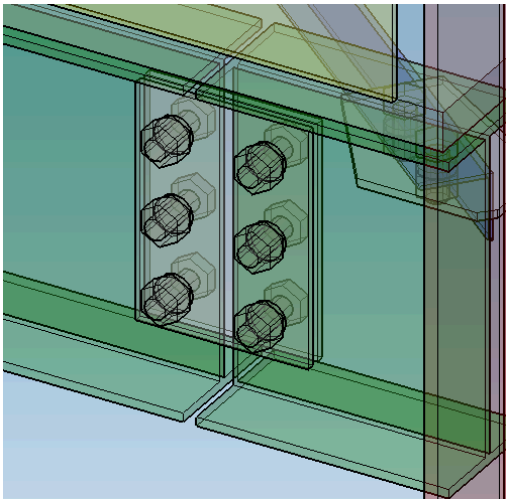
On the right side, within the model screen, notice 7 icons available to view your model.



Also note in the upper right corner, an orientation cube. Using the mouse, you can modify the rotation and view of your model with it.

Visualization types

Explore model view types by pressing CTRL+1, 2, 3, 4 and 5



Transparent view (CTRL+2)

Select

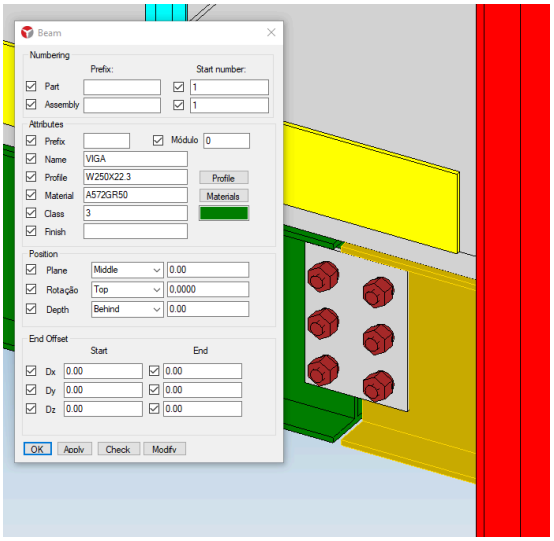
To select a piece, simply click on it (note that it changes color).
To select parts within a window, hold down the left mouse button and define the selection window, always**from left to right**. If you want to select the pieces that are in the window and the pieces that cross the edges of the window, create the**right to left**.

To select piece by piece, hold down**SHIFT** Press and click on the pieces you want. Do you want to exclude a part from the selection? Click on the part by pressing the**CTRL key**.

Double click on a part

Double click on the part you want to change. TSteel will identify which part was clicked and will open the correct editing window.

See the beam editing window:

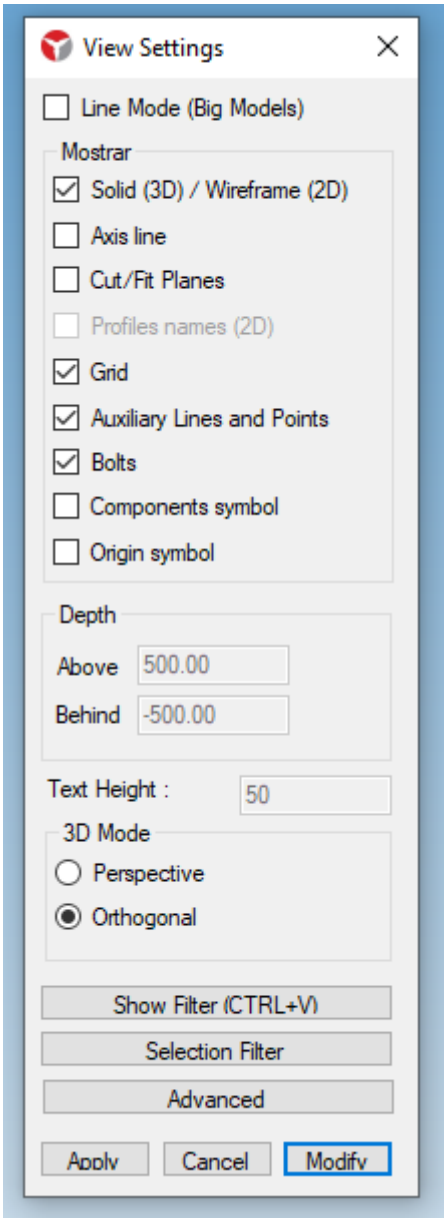


Take the opportunity to make changes and click**“Modify”.** See the changes portrayed in the model. Don't worry about “ruining” the model, just don't save it when you've finished doing your experiments.

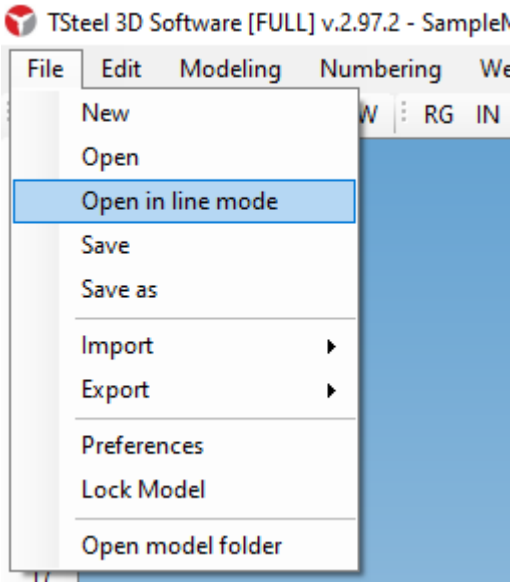
Open and see editing windows for beams, plates, columns, screws, etc…

Double click on the screen

Double clicking on the screen opens an important configuration window:



Line Mode: Represents the model in lines, aiming to save memory and time. May be useful for very large models. You can also open models in this line mode from the menu:



Turn on line mode and see it in operation.

Within the Show options, there are several options to turn on and off the display of model elements. Some of these elements may not yet be familiar to you, but for now, it's important to know that they can be shown or hidden here.

Some properties appear turned off, as they only make sense in 2D views, which we will see later.

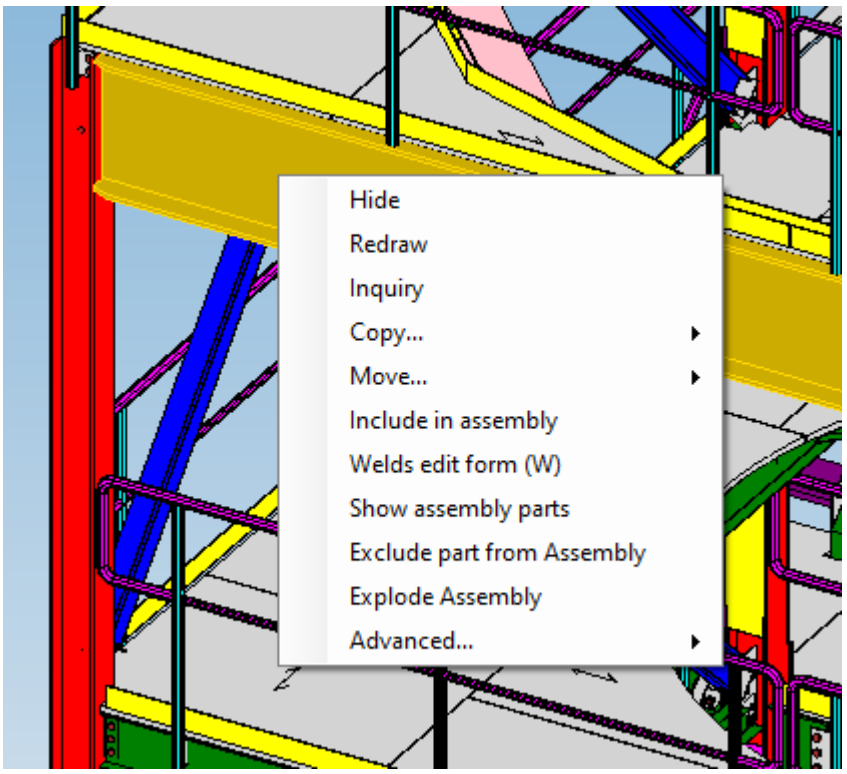
perspective or orthogonal viewing.

Filter of parts to be shown. We will look at filters later.

The advanced settings window, which can also be opened by ALT + double-clicking on an empty screen, configures the display of several important modeling tools. We will look at this in detail later.

Smart right-click menus

When you right-click on a part, TSteel will open a quick menu, as in the example below:



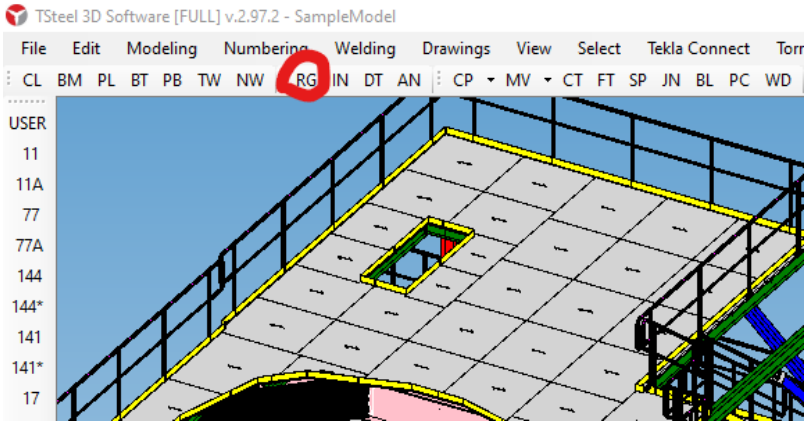
The menu options will vary depending on the element you clicked.

Hiding parts (Hide)

Note in the right-click menu that there is the HIDE command. Select the command and the piece will disappear from the screen. It was not deleted, just hidden to aid visualization.

Select a few pieces and hide them to test. To hide several pieces at the same time, select all the pieces you want to delete, and with SHIFT pressed, right-click on one of the pieces. Note that the menu appears and the pieces remain selected. Click Hide and hide them all.

To show the pieces again, just have the view regenerate. This command (RG) is in the toolbox at the top.

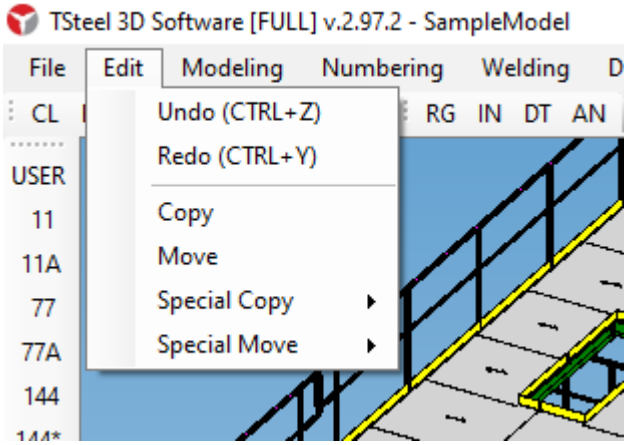


Deleting parts

To delete, press DEL. TSteel will delete all selected parts. Want to reverse the delete command? See below the UNDO and REDO commands

Undo/Redo

Many times we need to undo a command that was made by mistake. It is the UNDO command.



You can perform UNDO or REDO commands through the menu or using the shortcut screens.

TSteel is able to UNDO the vast majority of commands and stores the list of operations that can be undone while the program is active. In other words, after you save and exit TSteel, there is no more information to undo operations.

Grids, how to create and edit

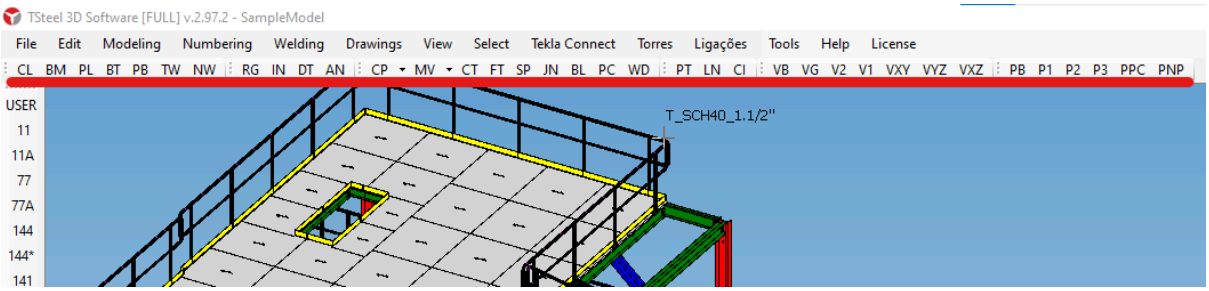
Every time you open TSteel 3D or create a new model, a basic Grid is created by TSteel 3D automatically. You can edit this grid by placing the distances and axis names you need.

To edit, simply double-click on any line in the grid.

See how to interpret the X coordinate line, for example: From the origin point (defined as 0,0,0) the first axis is at 0mm and I will have 4 more axes at 3000mm spacings. I will have a total of 5 axes, named A, B, C, D and E. If you enter “0 3000 3000 3000 3000” you will have the same axis definition.

Watch video about Grids

Toolbox, using right button and left button



Here we will see what Toolbox commands are and learn that they can be used with the right or left button.

If you want to know what one of the commands is, hover your mouse over it and a description will appear.

To execute the command, click on the command with the left button. To configure the command, right-click it.

We will learn how to use each of them throughout the manual, starting with creating columns and beams.

CL: (Column) Inserts a new column
BM: (Beam) Inserts a new beam
PL: (Plate) Inserts a plate by contour (defines the points of the contour traverse)
BT: (Bolt) Inserts screws
PB: (polybeam) Inserts a polybeam
TW: (Twin) Inserts double angles
NW: (Nuts and Washers) Inserts loose nuts and washers

RG: (Regen) Regenerates the 3D view, cleaning auxiliary entities and forcing the recreation of the solids of each part
IN: (Information) Provides information about selected parts
DT: (Distance) Provides the distance between two selected points
AN: (Angle) Provides the measurement of the selected angle

CP: (Copy) Allows you to copy parts and modeling objects
MV: (Move) `Allows you to move parts or modeling objects
CT: (Cut) Inserts cutting planes for parts
FT: (Fit) Inserts fit plans for parts
SP: (Split) Divides a bar into two
JN: (Join) Joins two bars into one
BL: (Boolean) Performs Boolean operations (solid subtractions), which allows you to add holes and/or cuts to parts
PC: (Polygonal cut) A way of cutting parts from a polygonal
WD: (Weld) Inserts welds

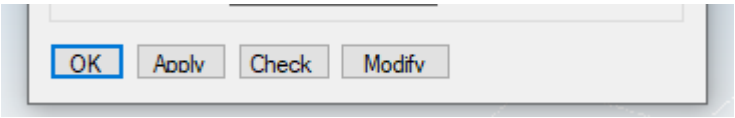
PT: (Point) Inserts an auxiliary point
LN: (Line) Inserts an auxiliary line
THERE: (Circle) Inserts an auxiliary circle

VB: (View - basic) Creates a basic 2D view (parallel to the XY, YZ, XZ planes)
VG: (View - Grid) Creates a 2D view from a grid line
V2: (View - 2 points) Creates a 2D view from 2 selected points
V1: (View - 1 point) Creates a 2D view from a selected point
VXY: Creates a 2D view in the XY plane of a selected part
VYZ: Creates a 2D view in the YZ plane of a selected part
VXZ: Creates a 2D view in the XZ plane of a selected part

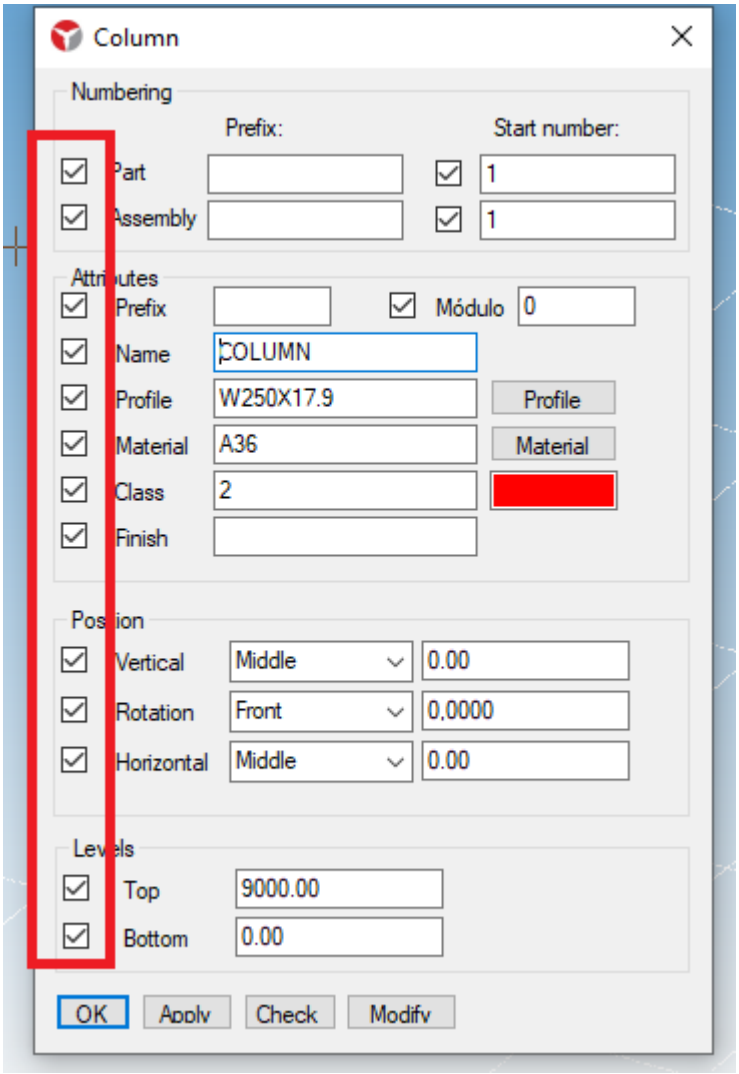
PB: Defines the work plane, parallel to one of the basic planes
Q1: Resets the work plane with 1 point selected
P2: Resets the work plane from 2 selected points
P3: Resets the work plane from 3 selected points
PPC: Resets the work plane to the principal plane of the selected part
PNC: Resets the work plane to the normal plane of the selected part

How edit forms work

All of the part editing forms we will see next have the following buttons at the bottom:



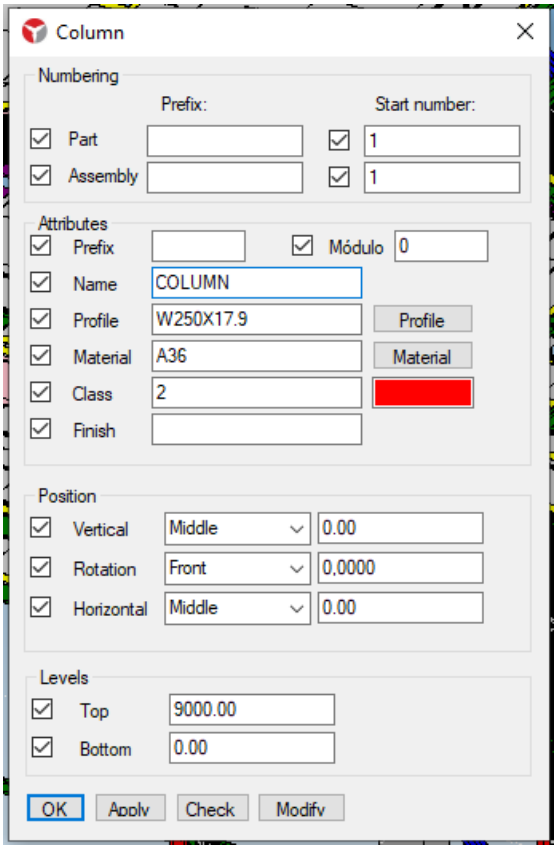
OK: Closes the window without executing any changes you have made.
APPLY: All information in the window becomes the new default. For example, the beam gauge is W250X17.9, all new beams created from now on will initially have this gauge. This applies to all properties: Class, Name, Materia, etc…
CHECK: Inverts CHECK selections for properties.



MODIFY: Modifies, in all selected parts, the properties that have CHECK turned on. This allows, for example, modifying the name (or class, material, etc.) of all beams in the model at once.

Create and edit a column

Columns are created from the CL button on the toolbar. Right click to open the settings window.



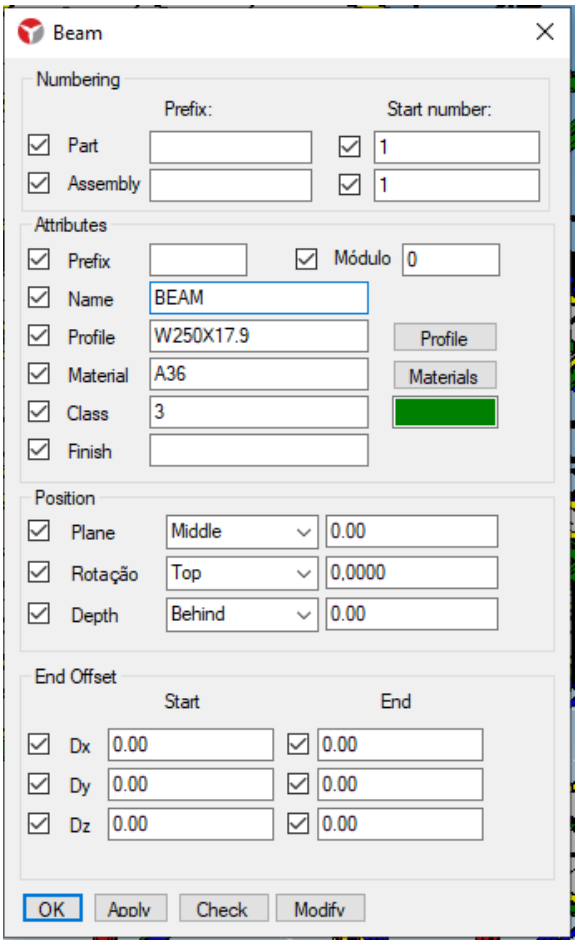
In this window we will define all the properties of a column, such as name, gauge (Profile), material, class, finish (Finish), positioning and the initial and final levels.

Numbering fields will be covered in another topic.

Watch the column creation video showing how to make changes and how to use the Apply (Mark changes as Default), Check (Which properties should be changed) and Modify (modify selected columns) functions

Create and edit a beam

Beams are created using the BM (Beam) button on the toolbar. The properties window is a little different from the columns, but the behaviors and concepts are the same.



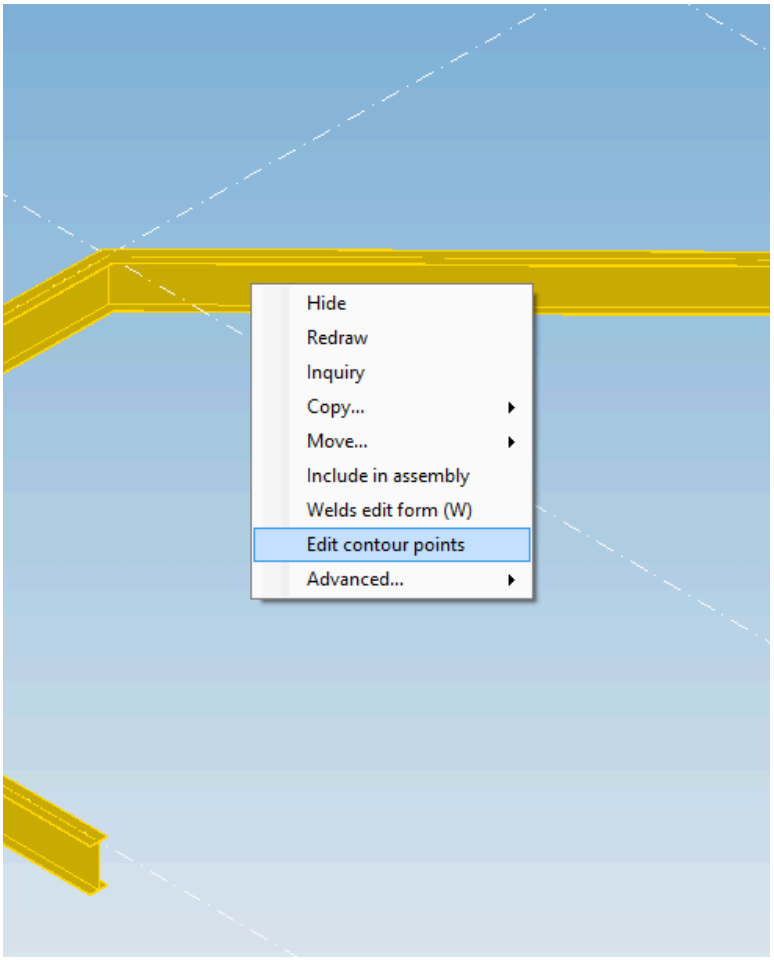
The difference in relation to columns is that we do not have start and end levels, but we have displacements that can be applied to the start and end points of the beam.

The operation for modification, application of default values and changes to gauge, material and class is the same as the column window.

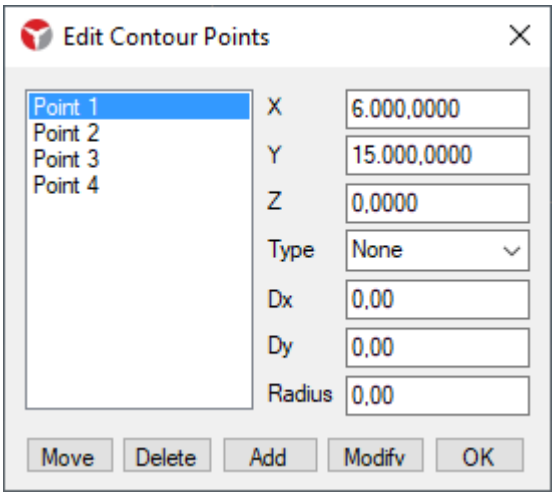
Watch YouTube video about beams

Create and edit polybeams

Polybeams, button PB (Polybeam), are defined with 3 or more points and their intermediate points (or contour points) have properties that change the geometry of the part. Once you have finished defining all the points, click the right mouse button.

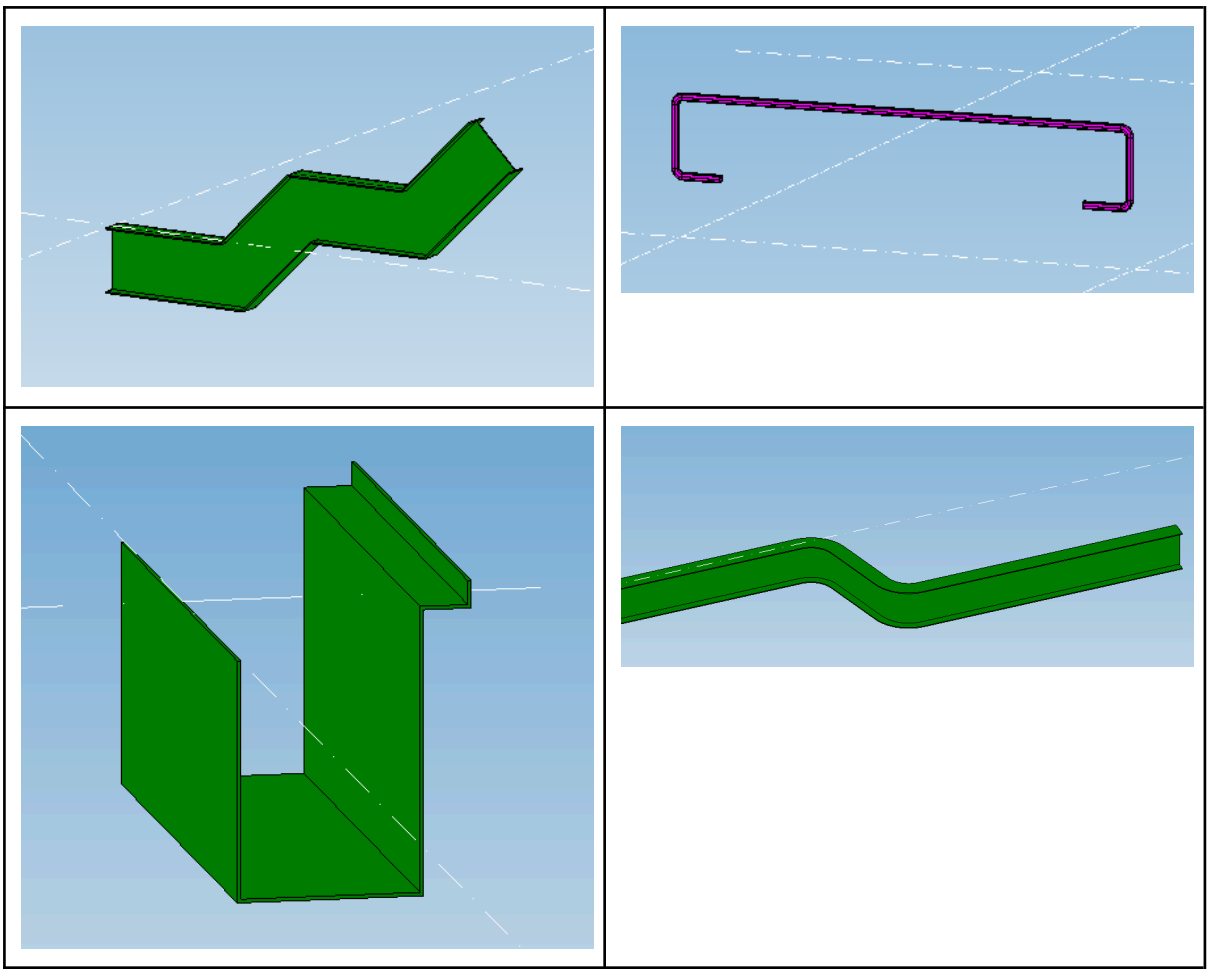


By right-clicking on a polybeam, you can call up the window that edits the contour points. For[see more about contour points](), see specific post.



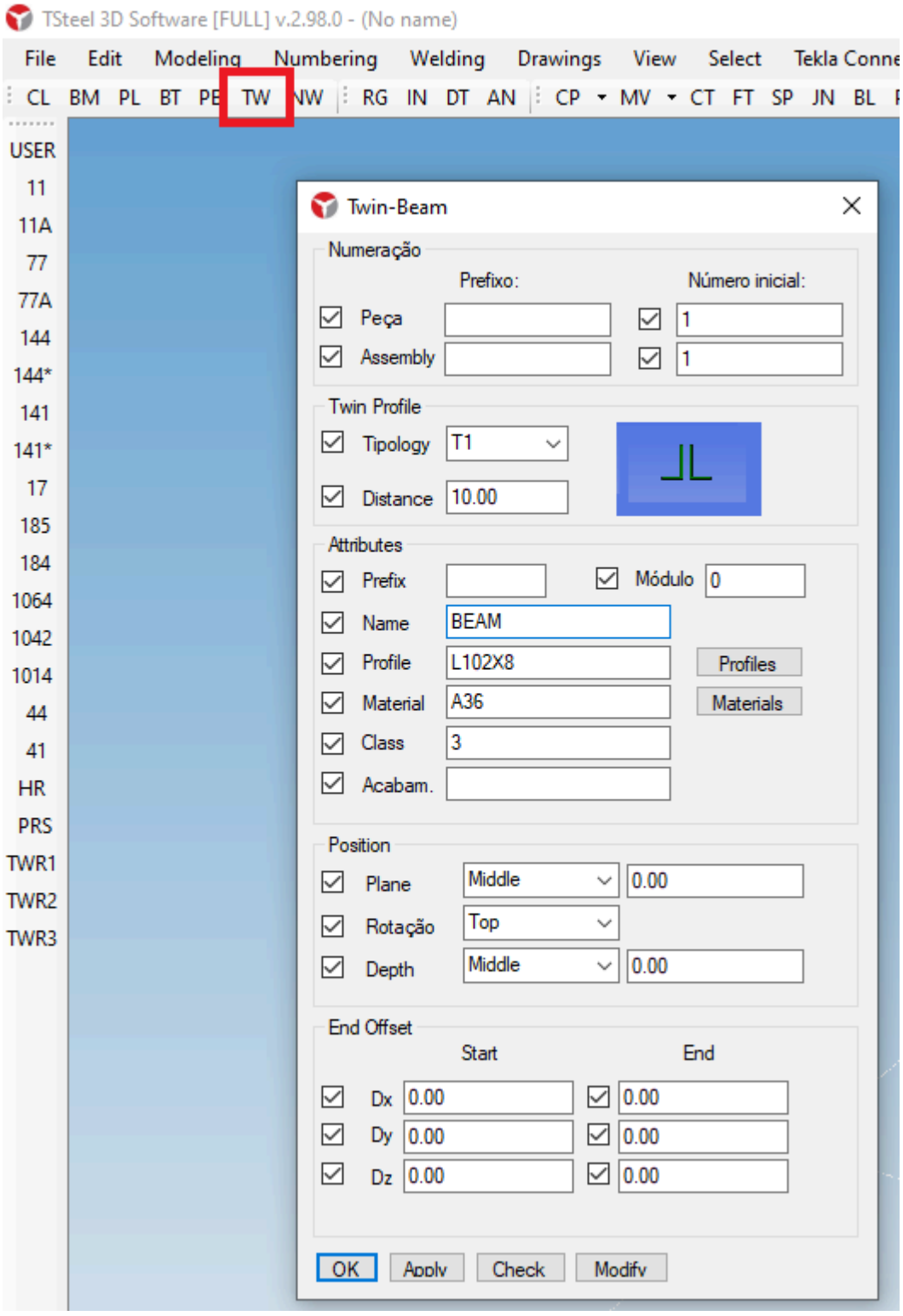
In this window you can edit points (coordinates or type), delete or create new ones.

Examples of polybeams:



How to create double angles (twin-beam)

TSteel 3D allows the creation of double angles by defining the working points and the type of angles. It's important to note that this is just a quicker way of casting two angles that will actually be separate beams for model purposes.



The double angles can be “exploded” (option accessible from the right-click menu), making them behave like two separate pieces.
Certain model editing operations automatically “explode” double gussets.

[Watch video]

How to create contour sheets

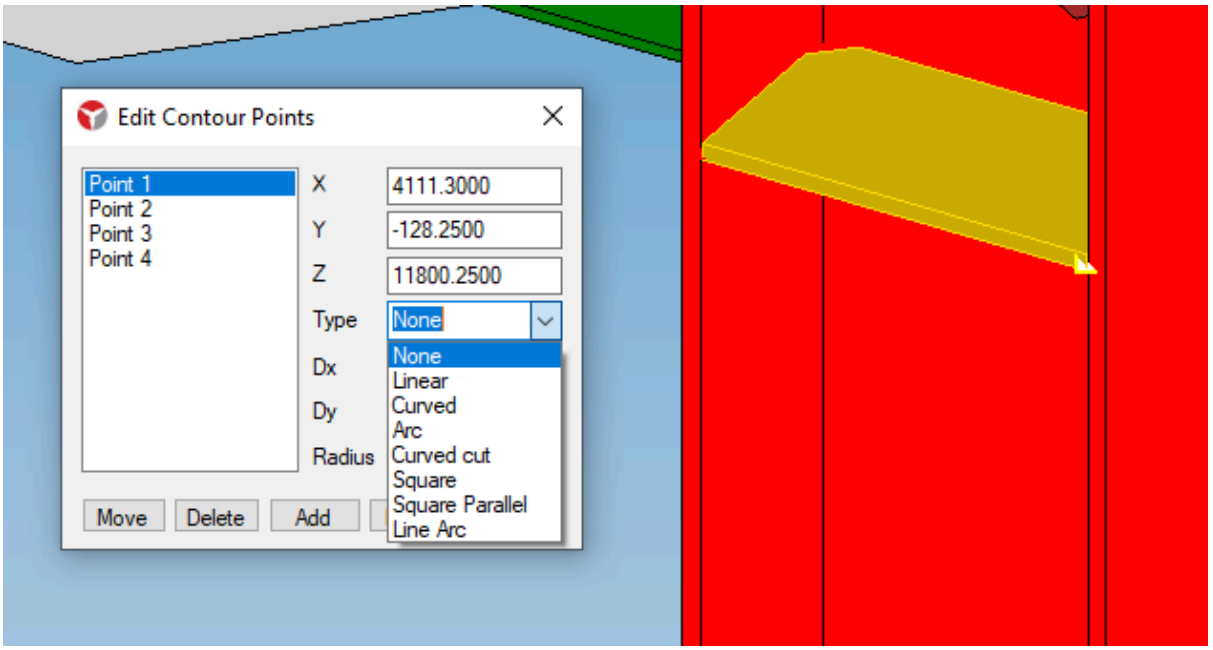
One of the ways to create a plate is by providing the contour polygon and the plate thickness. The contour points, just like on the polybeam, can be modified to create straight or curved chamfers.

To access the contour points, follow the same procedure as for the polybeams.

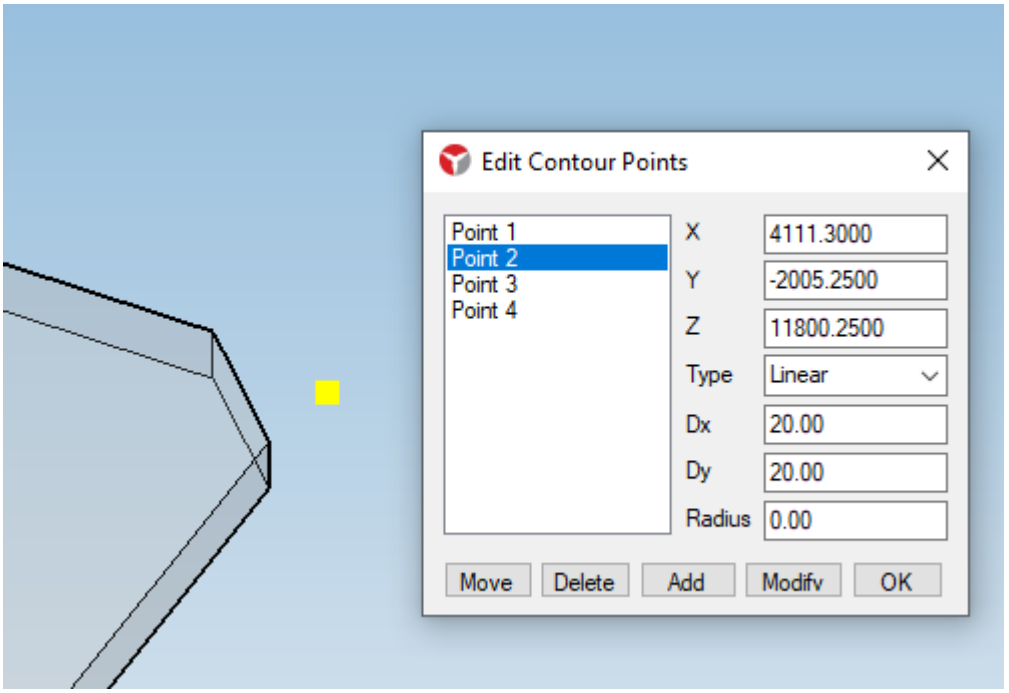
Watch video on YouTube.

Contour points

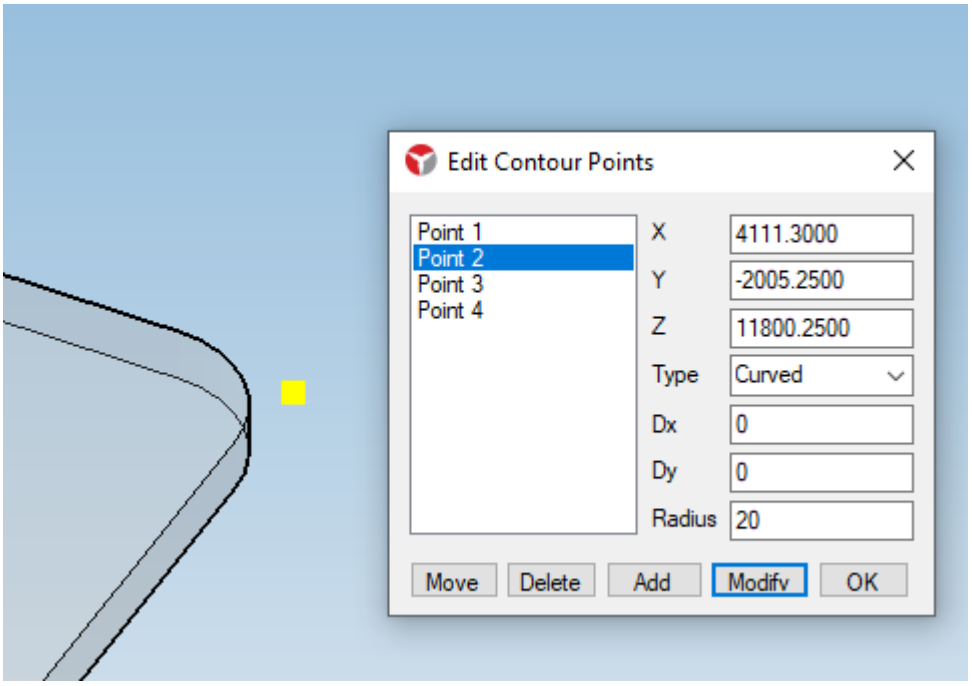
Contours are used to define polybeams and boundary plates. Each point on the traverse that defines the contour can have properties that help modify the contour.



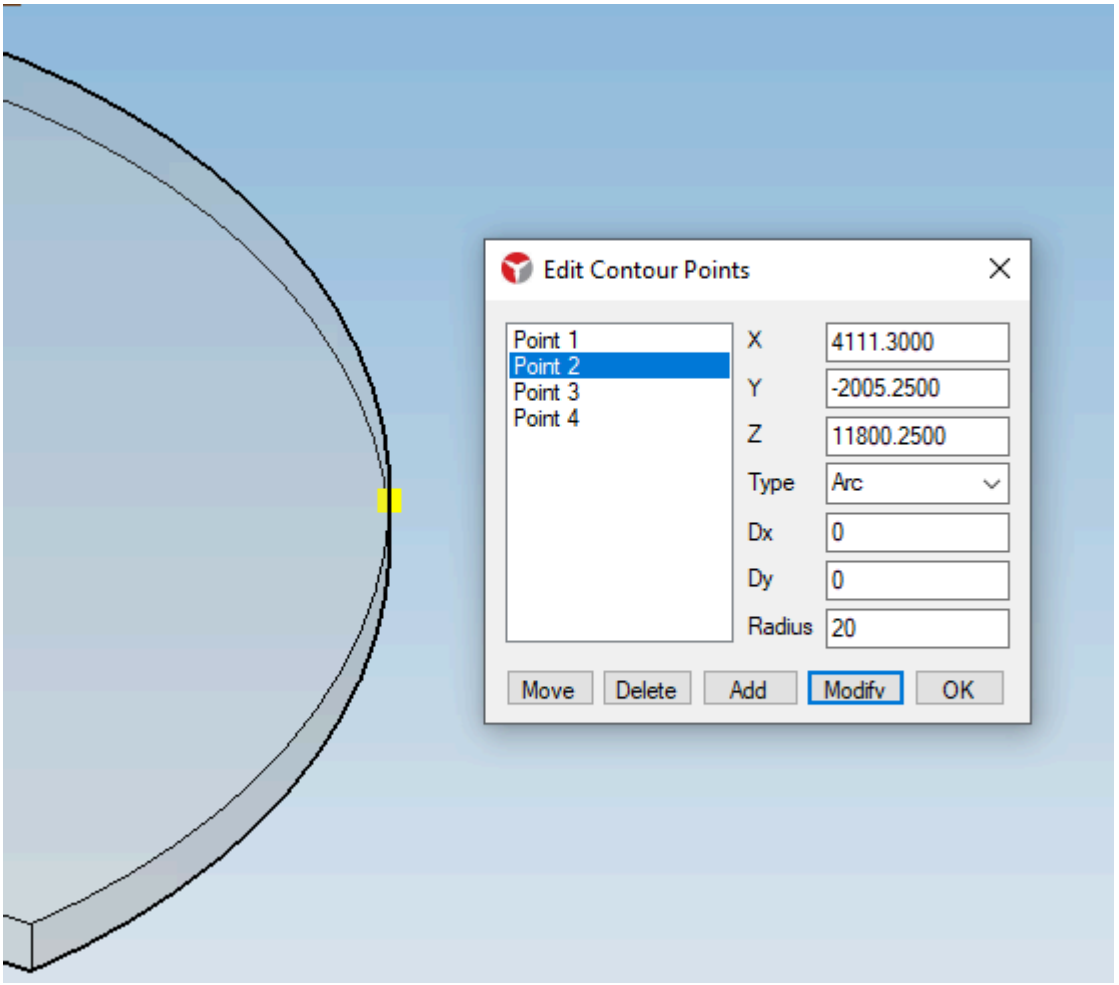
None : Does not modify the contour vertex. (Default stitch when creating the contour)



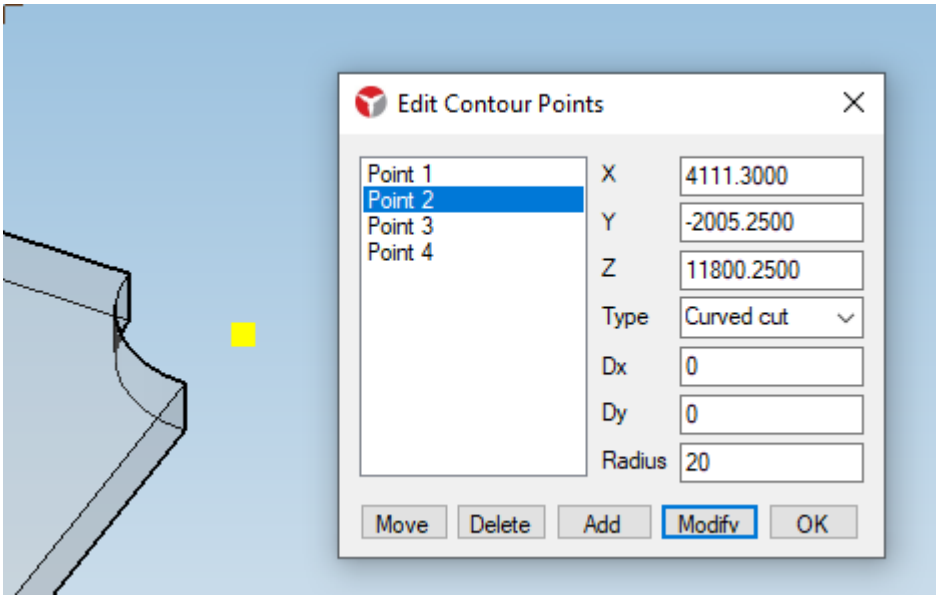
Linear: Makes a linear chamfer at the vertex. Two chamfer measurements are required (Dx, Dy)



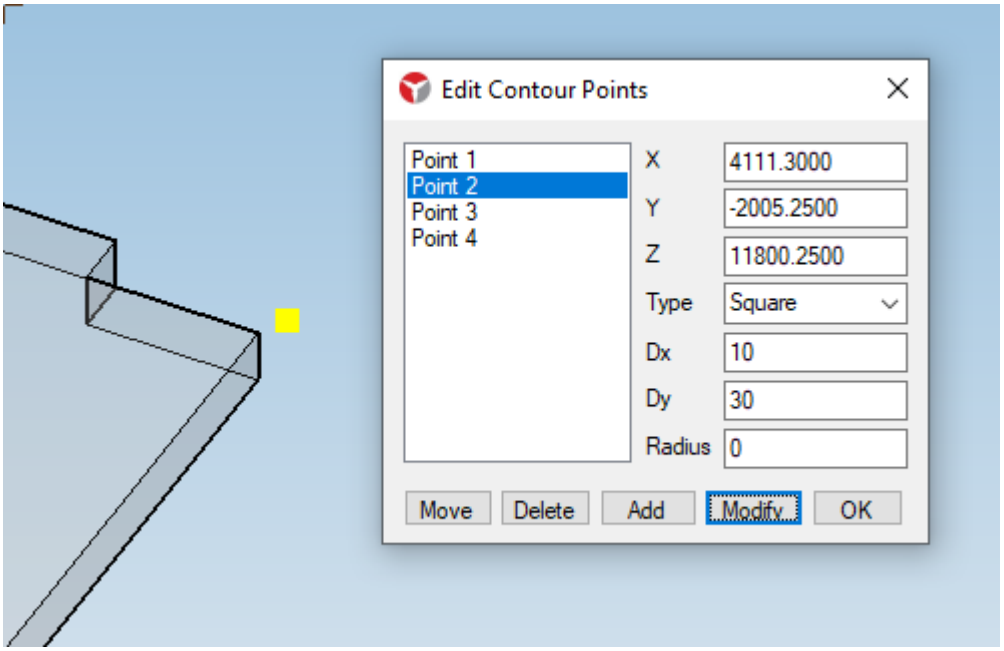
Curved: Rounds the vertex. It is necessary to define a value for the radius.



Arc: Makes an arc chamfer defined by 3 points: the previous point, the vertex we are modifying and the posterior point. As it is an arc defined by 3 points, you do not need to define the radius



Curved Cut: Makes an arc chamfer at the apex. You must define a value for the radius.



Square: Makes a chamfer with cuts orthogonal to the edges of the contour. It is necessary to define Dx and Dy values.

Square parallel: Makes a chamfer with cuts parallel to the edges of the contour. It is necessary to define Dx and Dy values.

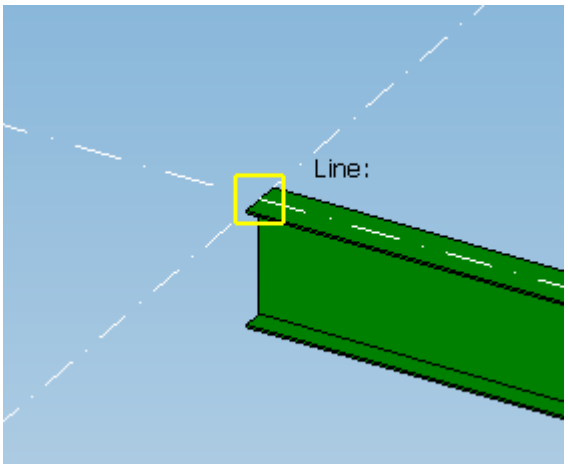
Line Arc: Similar to the chamfer with cutout, but with the radius defined by the smallest cutout side. It is necessary to define Dx and Dy values.

How to choose the right spots

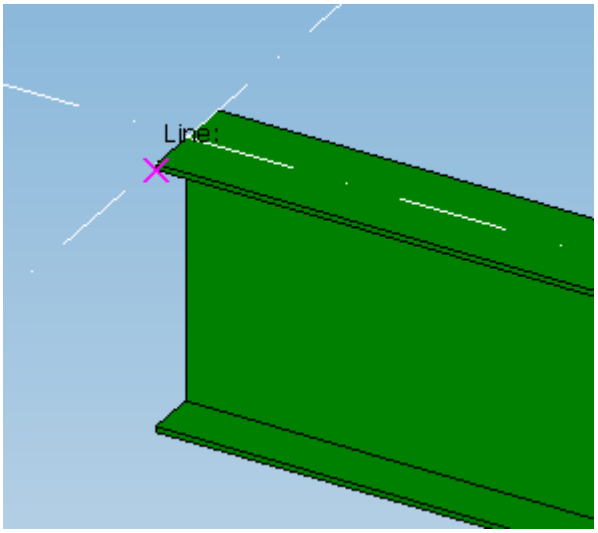
To be able to model, you need to know how to find and select the correct points, which will be used to define columns, beams, sheet contours, etc. The best way to understand is[watching the video from this link], but first let's see a little of the theory.

There are two categories of points:

1. Axis points. These points are those that the user uses to define a part, such as the starting and ending points of a beam. They are marked with a yellow rectangle.



2. Generic Points are the points formed by the edges of solids.



The type of point that TSteel 3D looks for depends on the options that are turned on. You will find, in the bottom bar, the buttons that define and help you during the process of choosing points. The types are:

- IN – Intersection between axes or edges
- EN – (End Point) Starting or ending point of an edge/axis
- PE – Perpendicular. It is the point that belongs to a line and is perpendicular to the previously selected point.
- MD – (MID) Midpoint of an edge or axis
- PT – (Point) Construction (or auxiliary) point
- CT – (Center Point) Center point of a circle
- SP – (Special Point) Special points
- QD – Quadrant of a circle
- NE – (Nearest) Point closest to the mouse position and belonging to the line or axis.
- ALL – Connects all types of points
- CLR – (Clear) Turns off all stitch types

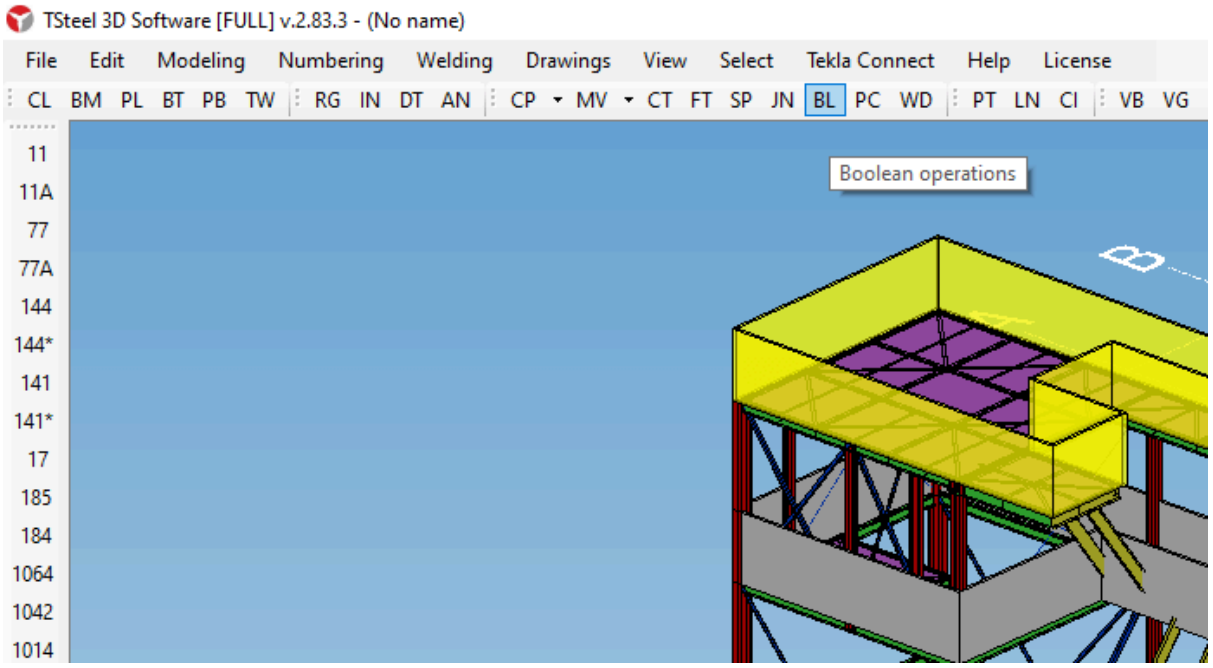
There are also options:

- ORTHO – The selected points will be orthogonal (X,Y axis orientation) in relation to the previous point
- AXIS – (F4) Turns axis point selection on or off
- PTOS – (F5) Turns the selection of generic points on or off

How to make holes and cutouts in parts

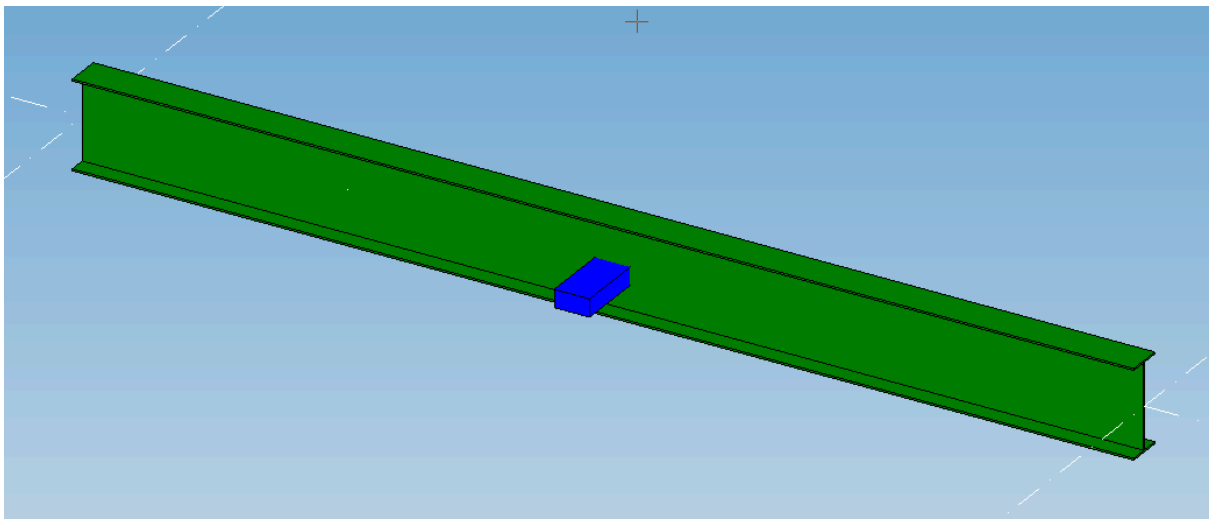
The holes and cuts in the parts are made using Boolean operations, which is nothing more than the subtraction of solids. The solid used to make the cutout or hole is called a Boolean part. The commands are in**BL** (Boolean operations) and PC (Polygonal cut) which is a faster and easier way to perform a Boolean operation. Let's see both ways working in the video.

Watch the video about holes and cutouts

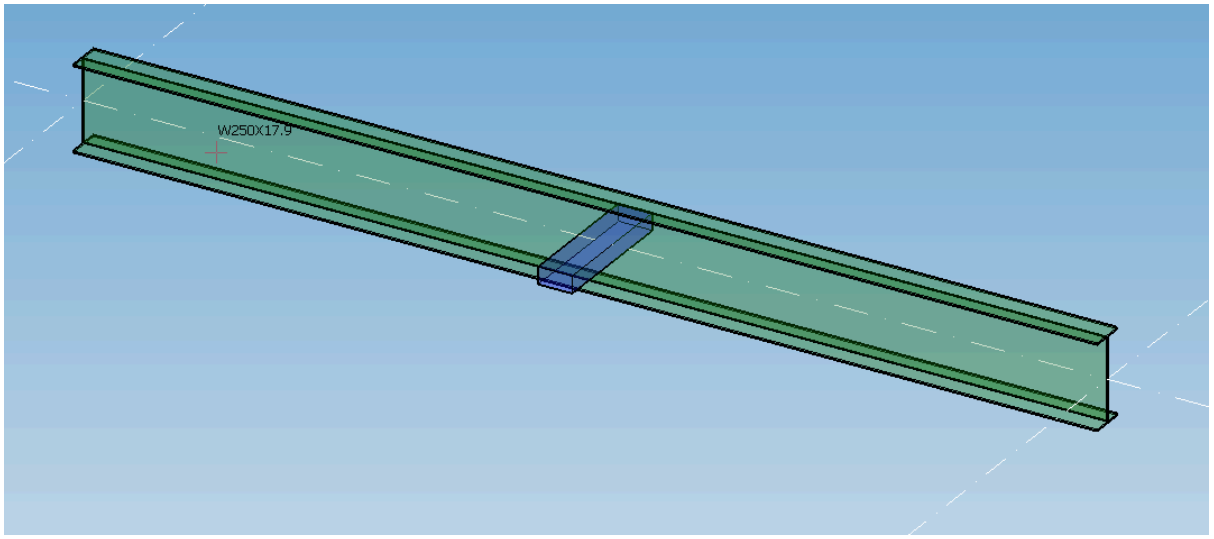


Let's drill a rectangular hole in the web of a beam

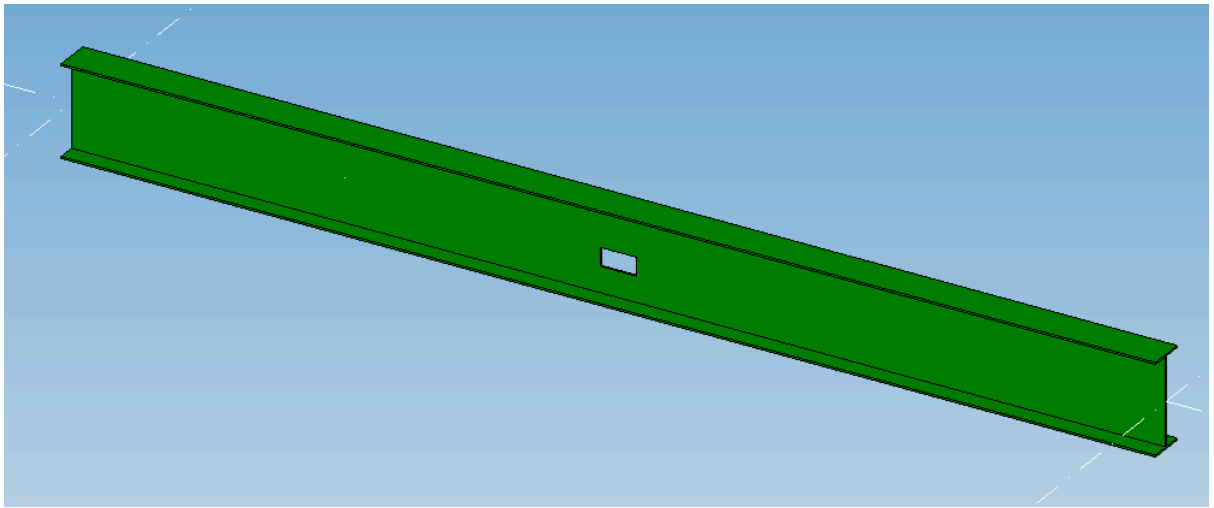
Keeping in mind that we will make the hole through a solid subtraction operation, create a part in the position and with the dimensions of the desired cutout, for example the blue plate below:



See in the transparent image that the sheet crosses the beam:

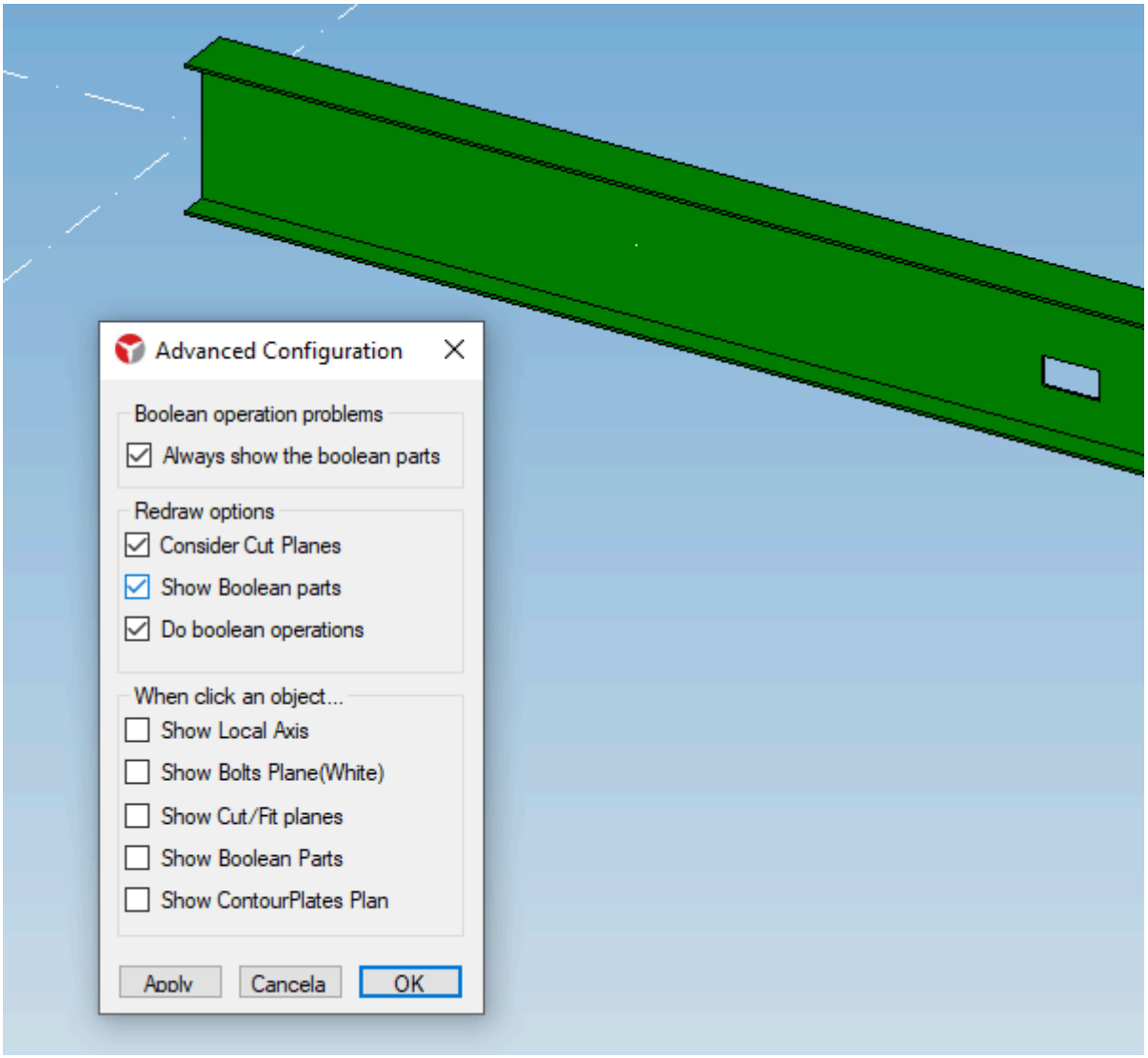


Now, use the BL command, selecting the main part and the cutout part. Once the command is finished, it will appear that nothing happened, since the blue plate is filling the place of our cutout. Erase the blue plate:

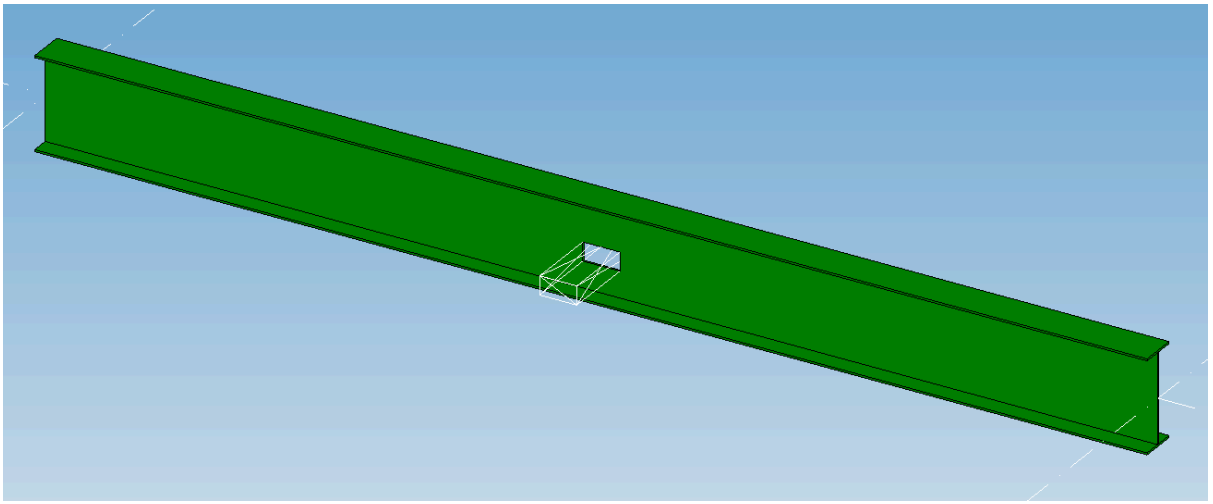


We have the rectangular cutout in the beam.

To view the cutout solids (Boolean parts), press the ALT key and double-click on the screen. Turn on the option to view Boolean parts when the part is selected:

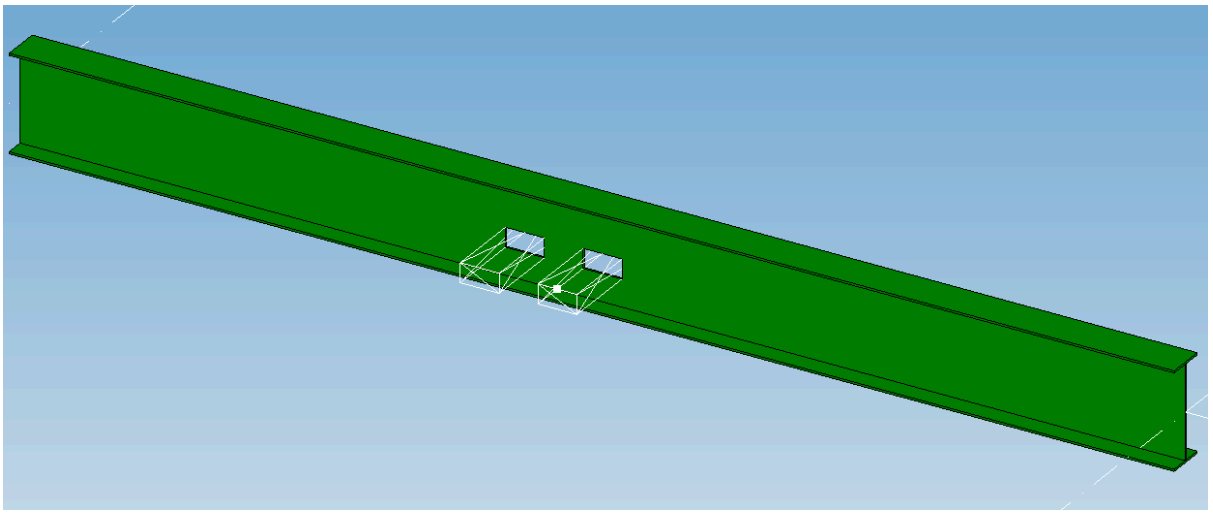


Now click on the beam:



The plate used for cutting is represented in a special way, showing that it is not a part of the model but a Boolean part.

Select the Boolean part and make a copy, you will have created a new cut in the beam:



Try selecting the boolean part and moving it. You will see that you will be moving the cutout. Similarly, select the Boolean part and delete it, and you will be deleting the clipping.

Using the polygonal cropping command

Polygonal cutting (PC command - Polygonal Cut - on the toolbar) makes these Boolean operations easier. From a 2D view of your part, you draw the cutting polygon and TSteel will:

1. Create a contour plate with the given points
2. Calculate the thickness of the sheet in order to cut the entire section of the main piece
3. Perform the Boolean operation
4. Eliminate the auxiliary part

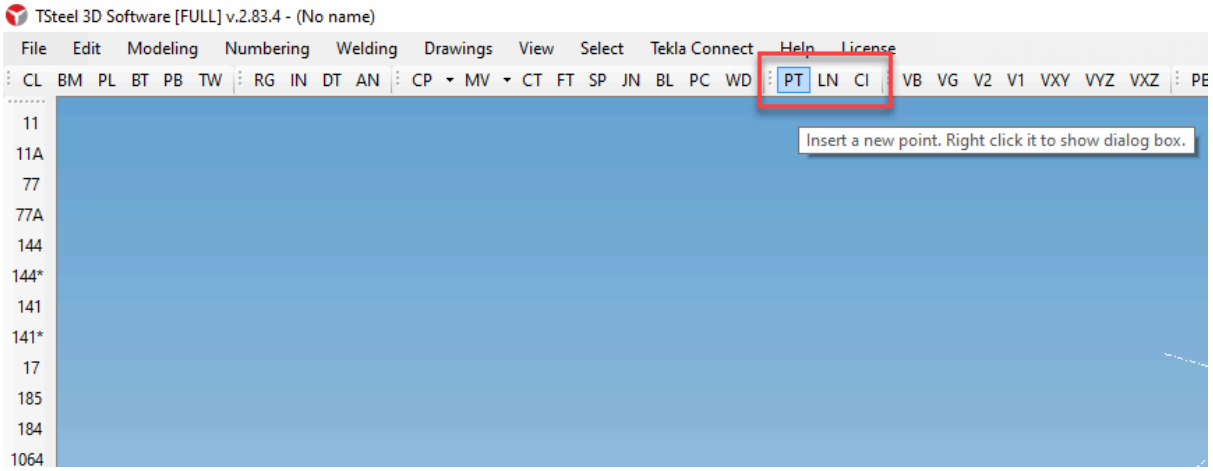
The best way to see this working and understand the command is by watching the video at the beginning of this section.

Find new points from known points

You can define points from a known point plus any offset DX, DY, DZ. Once the first point is selected, simply type in the displacements (an automatic displacement window will open) that you want and press ENTER. Ready!See it working in this video.

Create points, lines and work circles

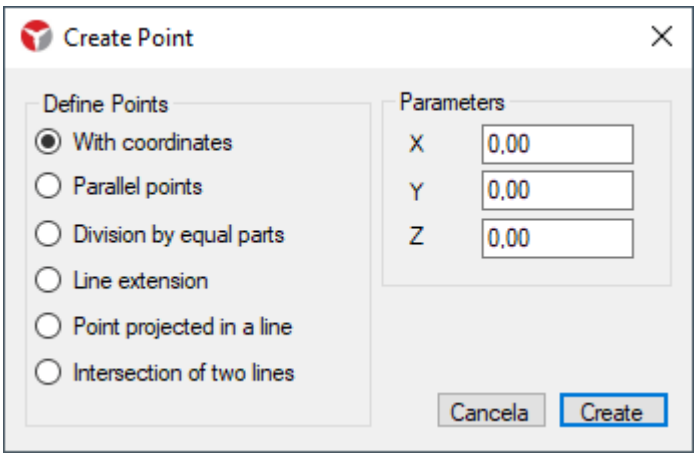
For video on YouTube, click here.
TSteel 3D has three auxiliary elements, all of which are located in the toolbar:



- Points (PT)
- Lines (LN)
- Circles (CI)

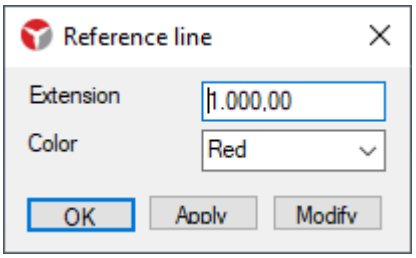
Auxiliary points

Auxiliary points can be created in several different ways, which makes them very useful in building the model. See the possibilities below:



Auxiliary Lines

You can right-click on LN to define parameters for creating auxiliary lines.



Auxiliary circles

Auxiliary circles are constructed on the current work plane, always defining the center point and a second point to define the radius (remember that you can define the second point by entering the relative coordinates with respect to the center).

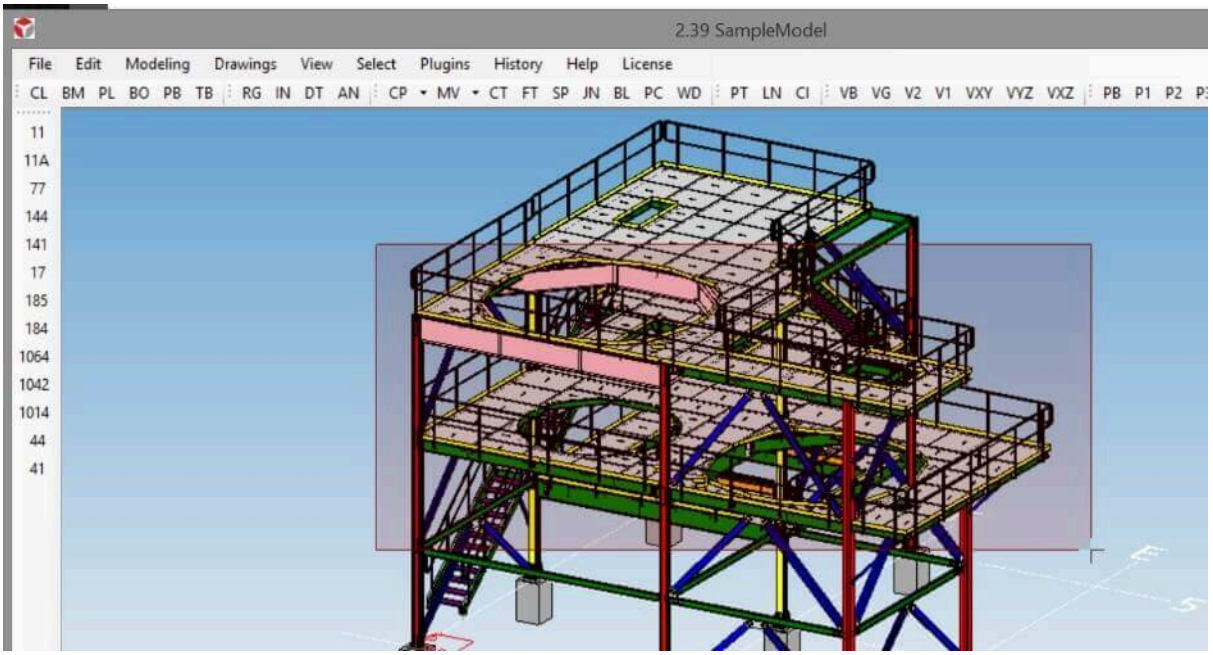
Selection of parts and points

Click selection

You select a part when you click on it. To add pieces to your selection, hold down**SHIFT** pressed while clicking new tiles. To delete a part from the selection, click on it while pressing the**CTRL**.

Selection by window

To create a selection window, click anywhere on the screen and drag the mouse. If you drag the mouse from left to right, a selection window is created where all the pieces that are 100% within the window will be selected.



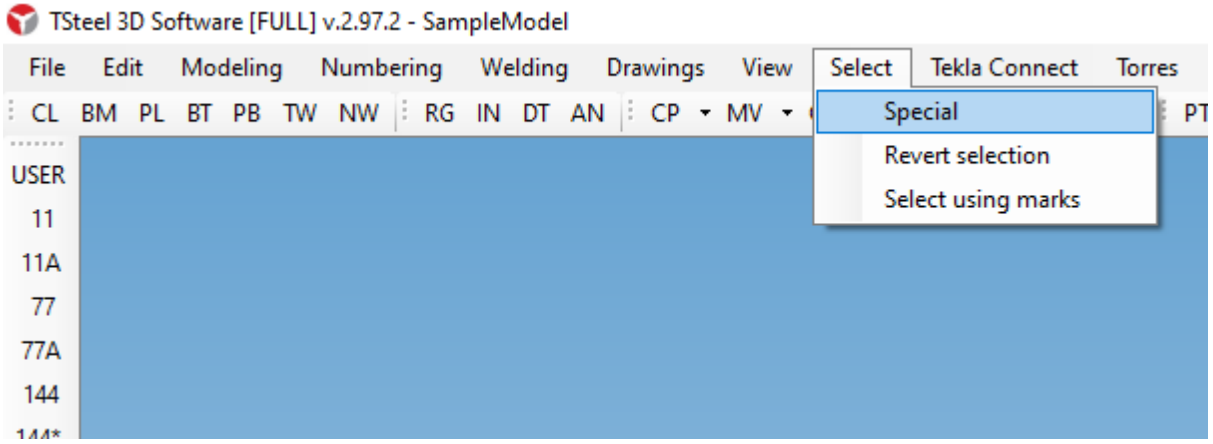
If you drag the mouse from left to right, you select the pieces that cross the window, that is, pieces that have any part of them inside the window.

The SHIFT and CTRL keys also work to include or exclude parts from the selection.

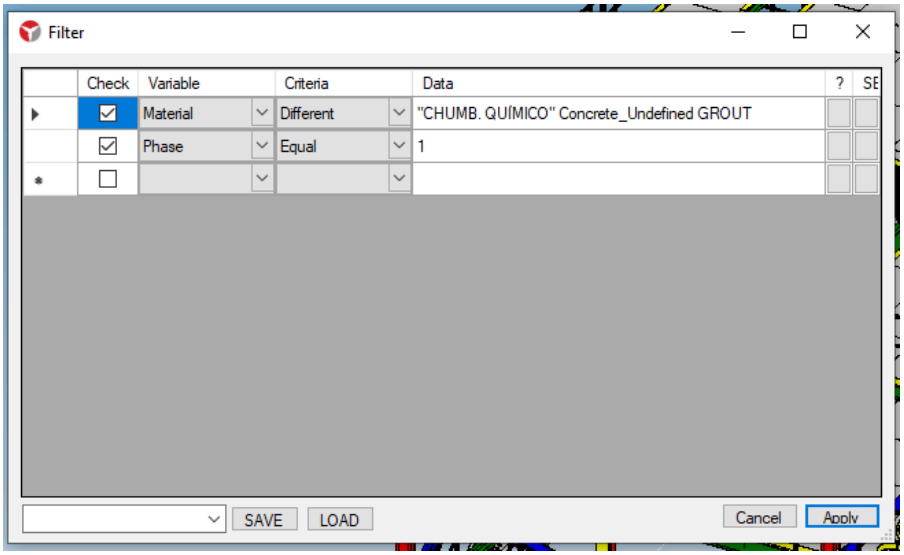
Selecting points

When you select parts, TSteel 3D shows the construction points. The white point is the starting point and the red point is the end point (or end points for contour plate and polybeams).
You can move points and change geometry, or change the configuration of points on plates and polybeams.
To select points, hold down**EVERYTHING** pressed. Points can be selected by clicking or using windows.

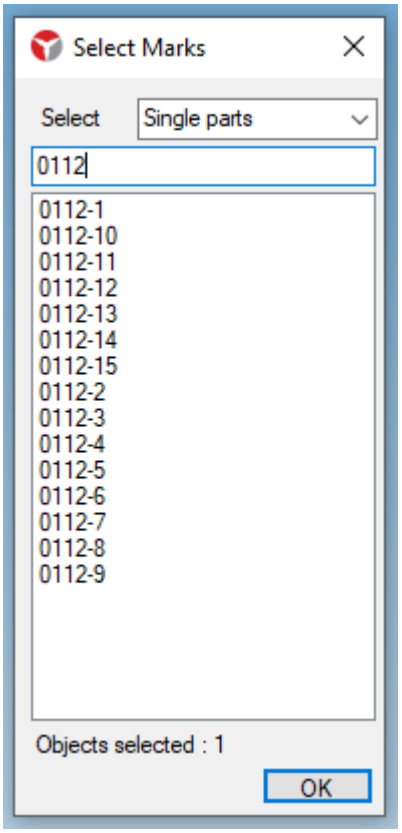
The selection menu



Special selection is through a smart filter. You determine the selection filter rules.



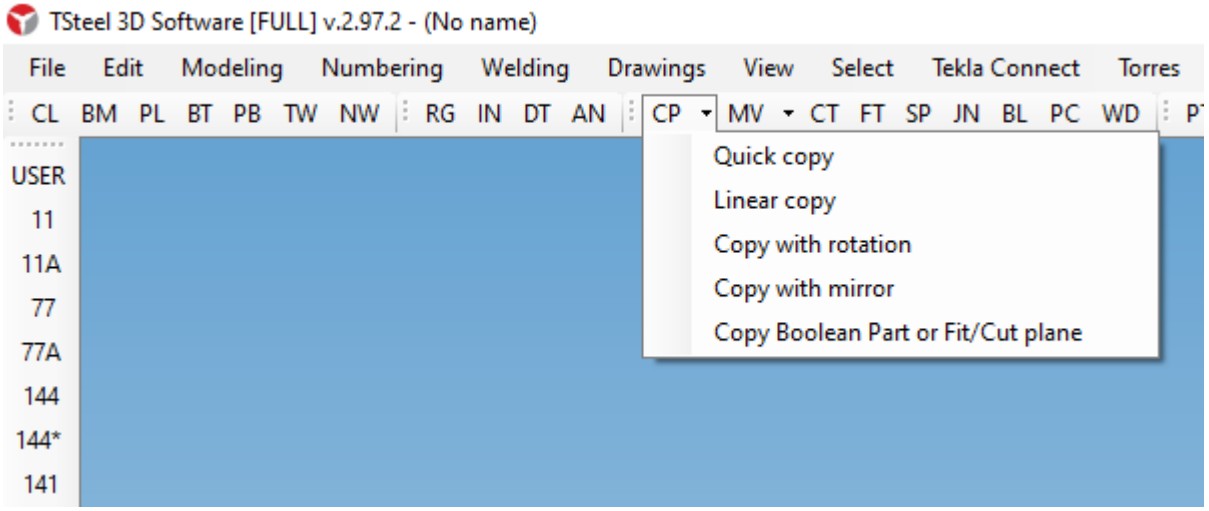
Selection using marks opens a window:



Note that there is a “0112” filter in the first line. It only shows parts starting with 0112. If you leave this line empty, you will have a list of all brands of the model.
Just click on the brand you want to select in the template.
This window can be used to select parts or assemblies and is a quick alternative for identifying marks on the model.

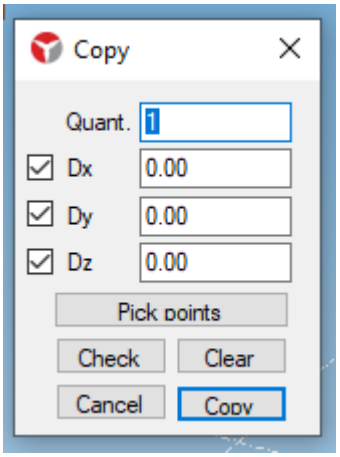
Copy parts

The copy options are in the toolbar, under the CP button:

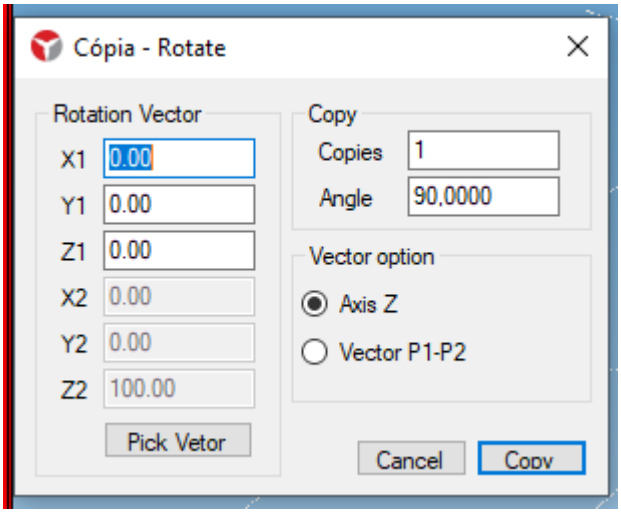


Quick copy(CTRL+C): Copies the already selected pieces, asks for two points to define the displacement. You can click a first point and manually enter the displacements dx,dy,dz. Just enter the coordinates separated by commas.

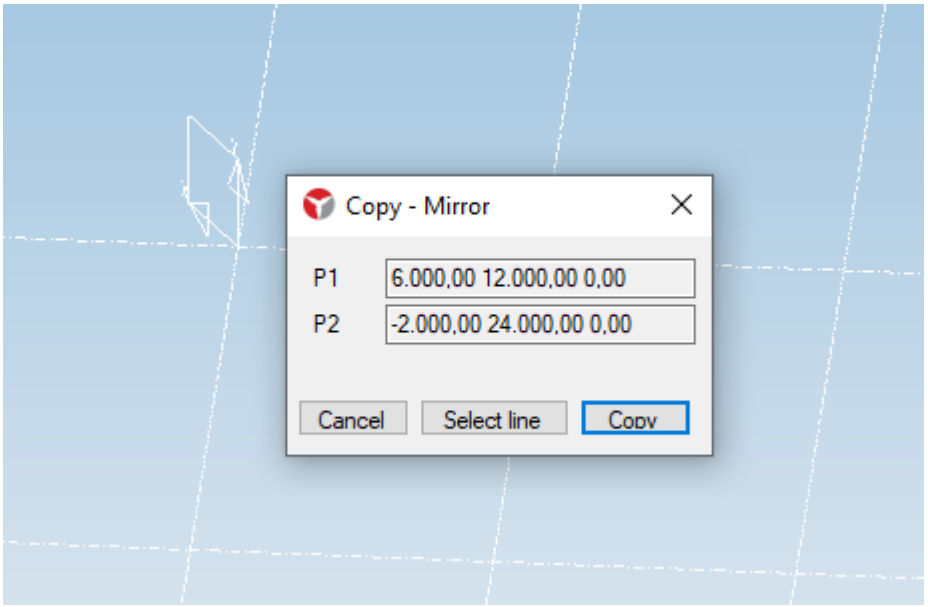
Linear Copy: Opens a window for defining copy parameters. Note that it is possible to create several copies at once. Displacements can be defined by entering values or selecting 2 points (Pick points).



Copy with rotation: Once a rotation axis and angle are defined, the program creates a copy of the selected parts applying the defined rotation.



Copy with mirror: Copies selected parts, mirroring them from a defined plane. The mirror plane is a plane perpendicular to the current working XY plane. As the work plane is already defined, you need to choose 2 points that position the mirror plane.

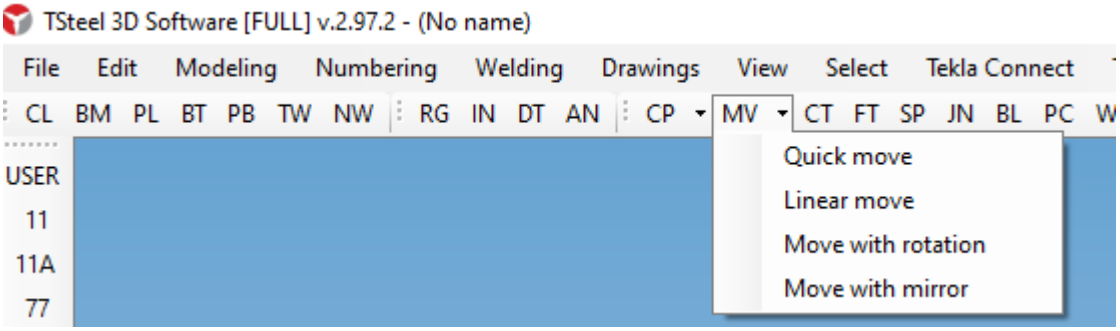


As soon as you make the copy, TSteel will indicate the axes of the mirror plane on the screen.

Copy boolean part or Fit/Cut Plane: Used to copy Boolean parts or part editing plans. You need to select the objects and apply the copy offsets.

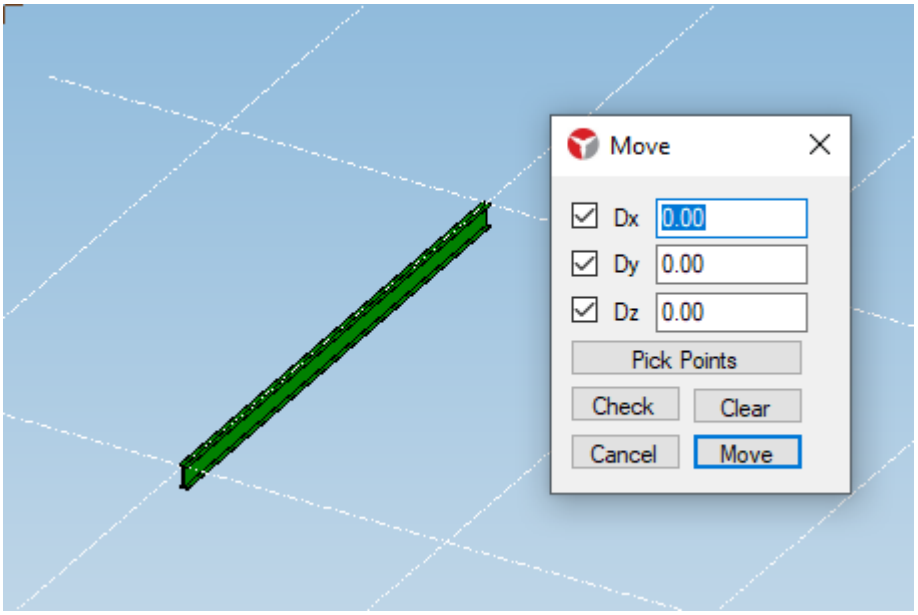
Move pieces

The options for moving pieces are:



Quick Move (CTRL+M): The program asks for 2 points and moves the selected pieces at the given offset.

Linear Move: Opens the parameters window for moving parts.

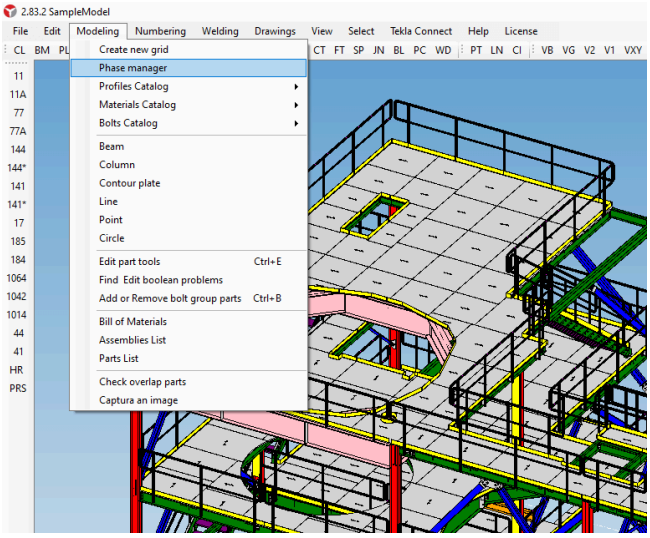


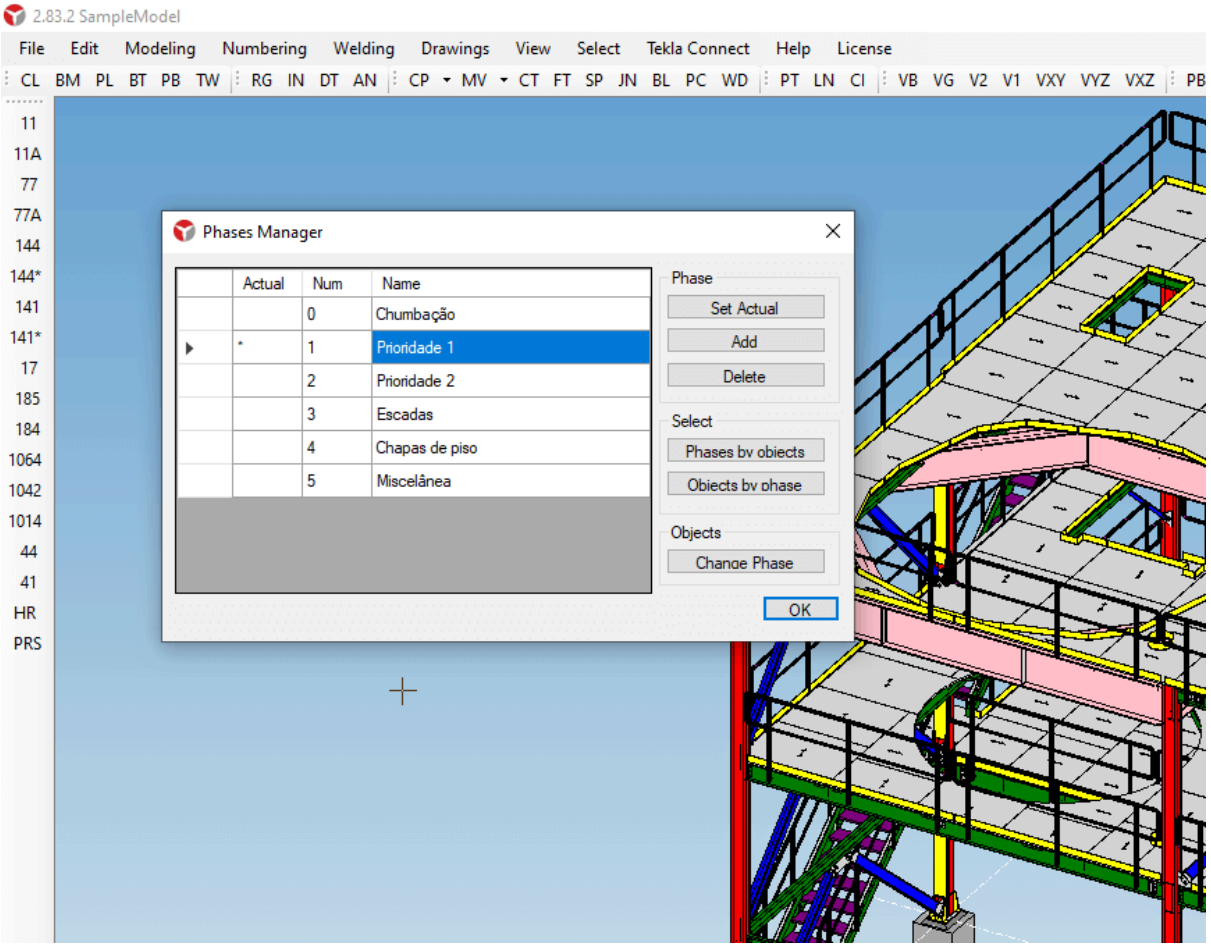
Move with rotation: Same criteria and operation as Copy with rotation, but in this case, it moves the pieces

Move with mirror: Move by mirroring the pieces. See more about the mirroring plan in “Copy with mirror” in the previous item.

Managing phases

It is very common to divide the model into phases to help separate material lists, filter specific parts of the model, indicate priorities, etc…
You can open the level manager via the Menu or the shortcut CTRL+H.





Keyboard shortcuts

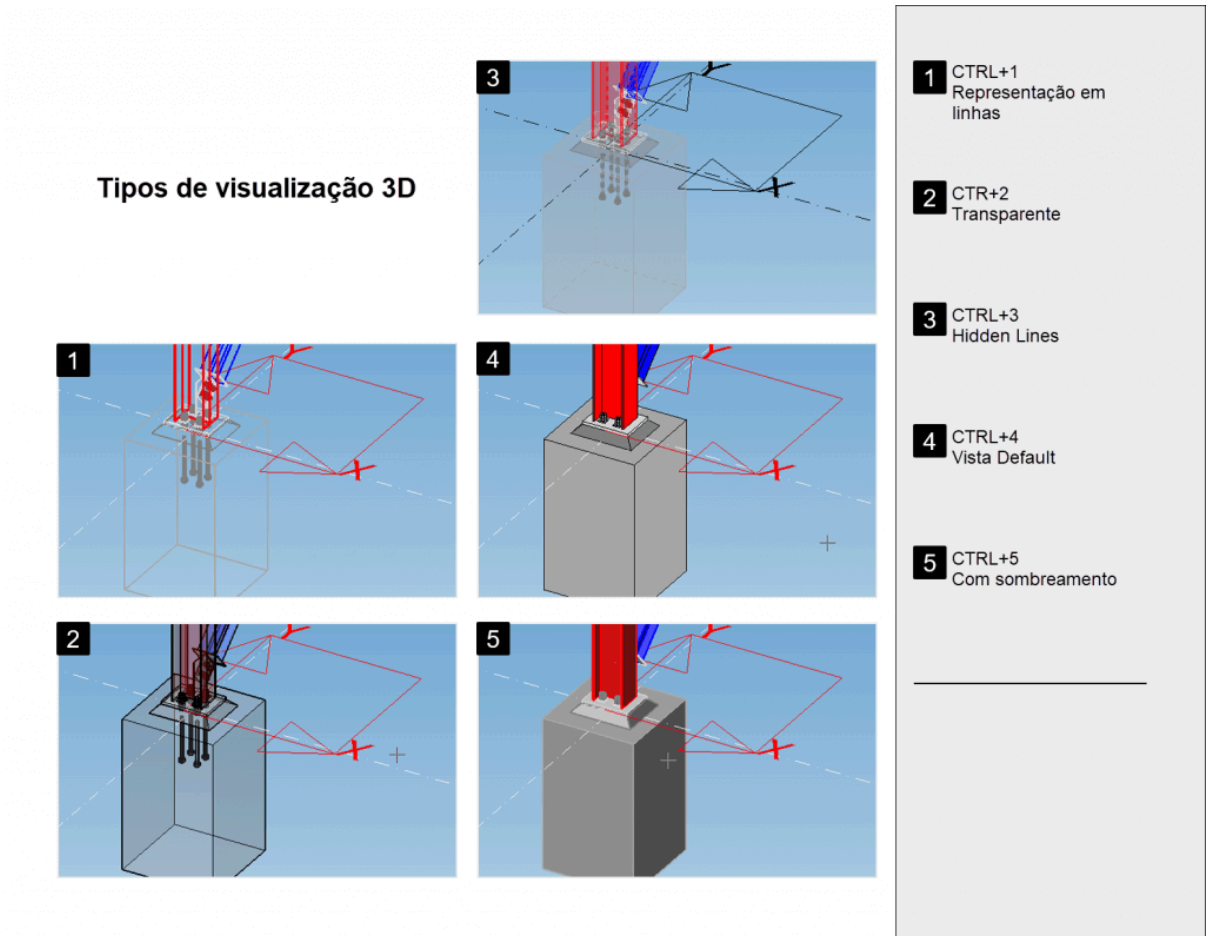
Below are the keyboard shortcuts:

CTRL+H	Hides the selected part (A Regen returns the part view)
SHIFT+Z	Zoom to selected parts
CTRL+A	Select all pieces
SHIFT+H	Phase Manager
SHIFT+I	Shows part information
OF THE	Deletes selected parts
F4	Turns the “snap” (search) for general points (corners, edges, etc.) on and off
F5	Turns the “snap” (search) for profile creation points on and off, that is, points on the definition axis.
F6	Open the OSNAP window
CTRL+C	Quick copy command. Makes a copy of the selected parts.
CTRL+M	Fast move command. Move already selected pieces
CTRL+P	Change view to the plan’
CTRL+S	Save the model
M	Calls the move command
C	Invokes the Copy command
CTRL+1/2/3/4/5	Change the screen viewing style

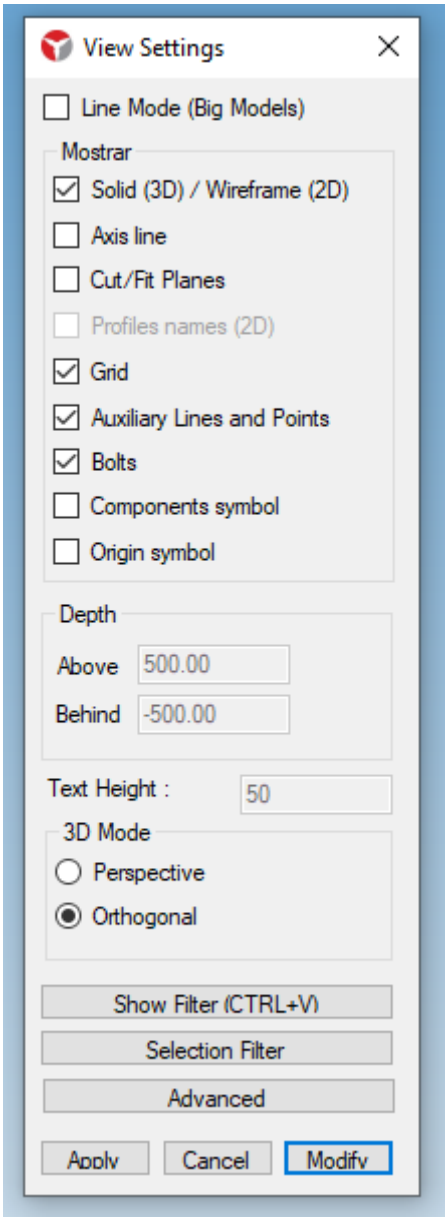
Preview

Main 3D screen and its settings

The main view of the 3D model can be configured in several ways, starting with how the structure is viewed:

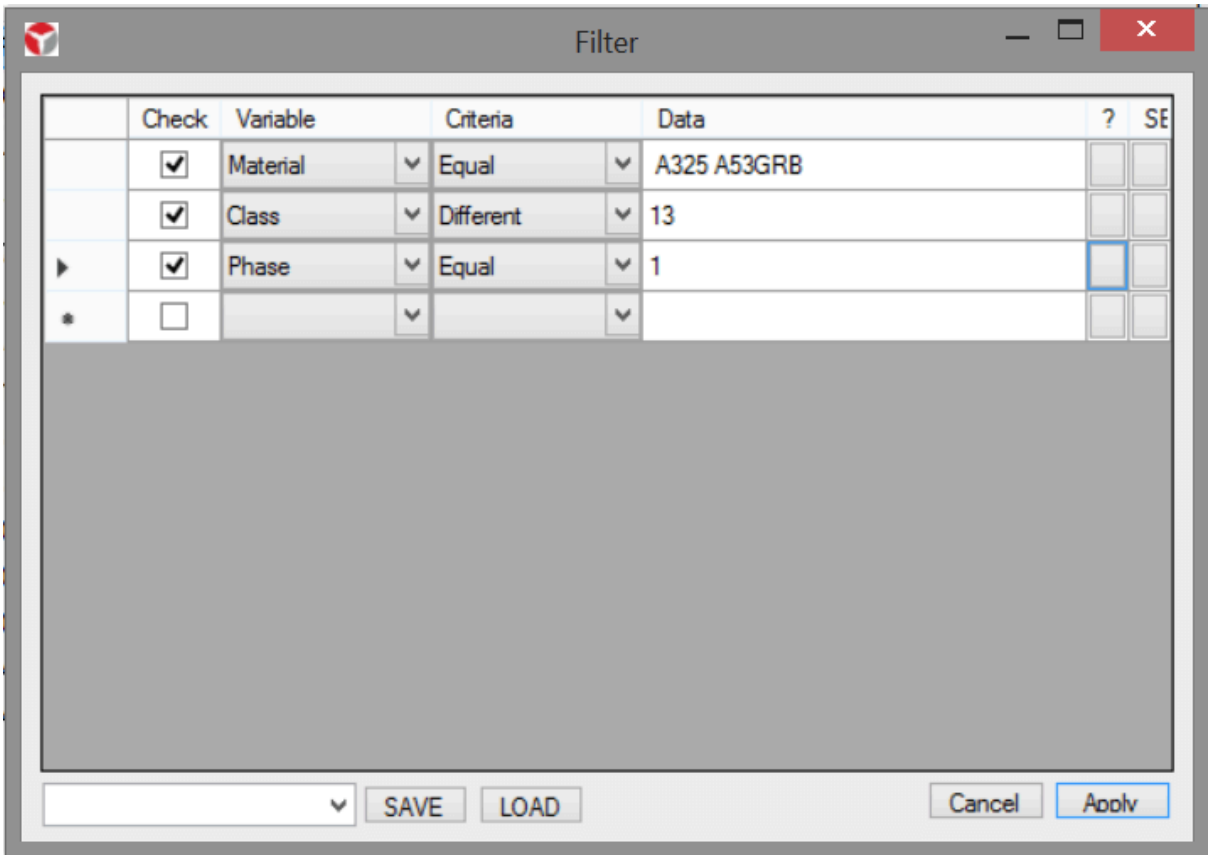


By double clicking on the empty screen, the settings window appears:

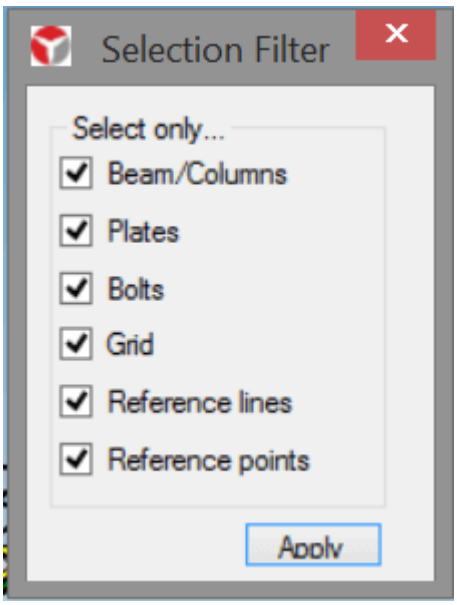


Line Mode: Used to represent the model by the axes of the parts (lines). There is great memory savings and high import and export speed.
Solid(3D)/WireFrame(2D): Turns the representation of model parts on and off.
Axis Line: Turns on/off the representation of the construction axes of the parts (line between the start and end points)
Cut/Fit Planes: Turns on/off the representation of cutting planes and adjustment planes.
Profiles Names (2D): Insert text with part profiles in 2D drawings. You can move the texts freely with the mouse to adjust the view.
Grid: Turns on/off the representation of the structure's Grids.
Auxiliary Lines and Points: Turns on/off the representation of auxiliary points and lines.
Bolts: Turns on/off the representation of model screws.

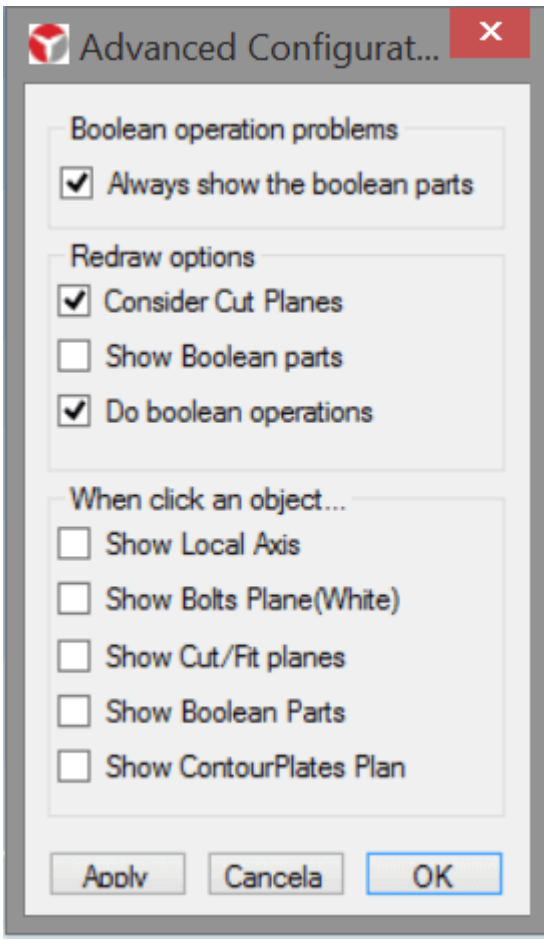
Components symbol: Turns on/off the representation of existing components/macros. The symbol is a icon that, when clicked, offers access to the macro window.
Origin symbol: Turns on/off the representation of the origin, point (0,0,0)
Depth (Above/Behind): For plan views, configures the depths that the view considers, both above the plane and below.
Text Height: Used only in 2D views, it regulates the size of inserted texts automatically.
3D mode(Perspective/Orthogonal): 3D view representation mode.
Show Filter: Opens the window where the user configures a filter of what should be shown in the view. Filters can be configured considering different aspects of the parts, allowing you to display only the part of the model you want.



Selection filter: allows you to turn on/off the selection of element types. For example, if Grid is turned off, you will not be able to select the Grid via windows or click.



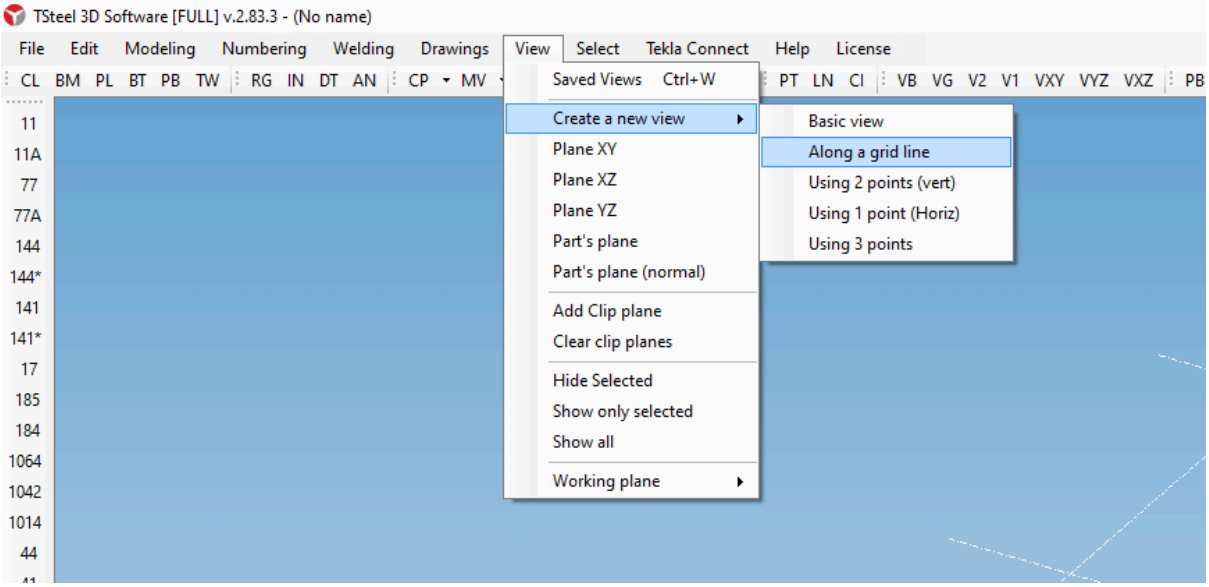
Advanced Configuration: The advanced configuration window can also be accessed by ALT+double clicking on the window.
This window allows additional settings on the program's behavior and how it draws the model.



2D view of the model

In some situations, working in a plan view is easier. TSteel 3D allows you to create any 2D view of the model.

How to create a 2D view



No menu "**VIEW**", you will find the different ways available to create 2D views.

- **Saved Views**: Access views you have previously saved
- **Plane XY, XZ, YZ**: Principal planes created from a part or grid line given as a reference
- **Part's Plane**: Plane of the selected part
- **Basic View**: One of the principal planes (XY, XZ, YZ) from an ordinate. (for example: XY Plan at quota 3400)
- **Along a grid Line**: Vertical plane coincident with the selected grid line
- **Using 2 points**: Click on two points to define the X axis of the view, the Y axis will be the vertical axis
- **Using 1 point**: Set a point for the height and an XY view will be created at this point
- **Using 3 points**: define any plane by 3 points.

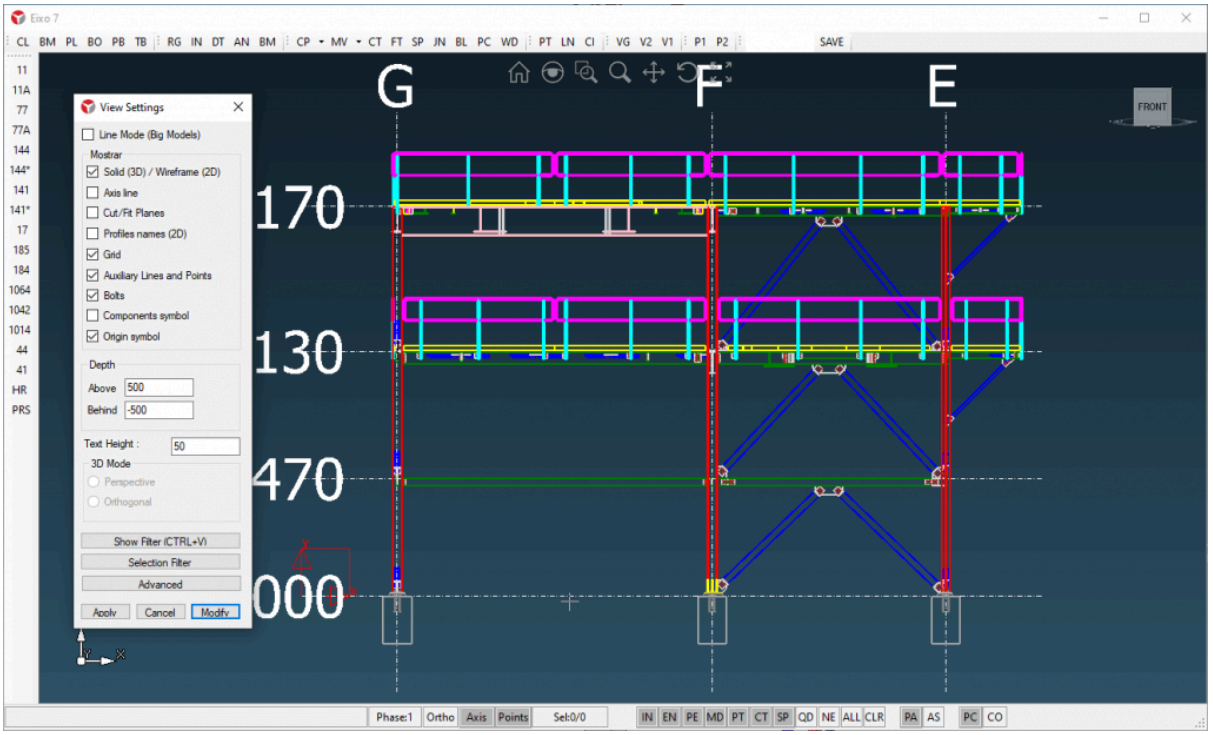
The most used alternatives are available in the toolbar:

- **VB**: Basic view
- **VG**: Grid view (from a grid line)

- **V2**: View from two points
- **V1**: View by 1 point
- **VXY**: XY plane view
- **VXZ**: XZ plan view
- **VYZ**: YZ plane view

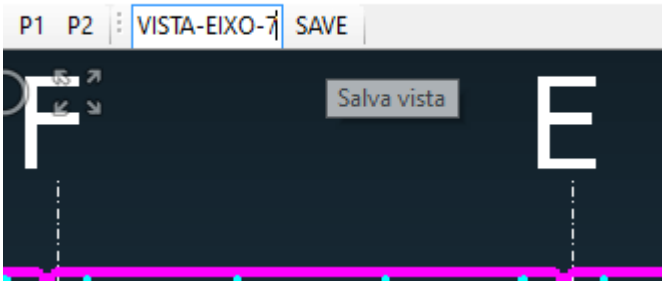
Settings for a view

Similar to the 3D view, simply double-click on an empty point in the view to open the settings window.
All elements in this window aredescribed in post about 3D view. Let's look in more detail at some items that concern the 2D view.

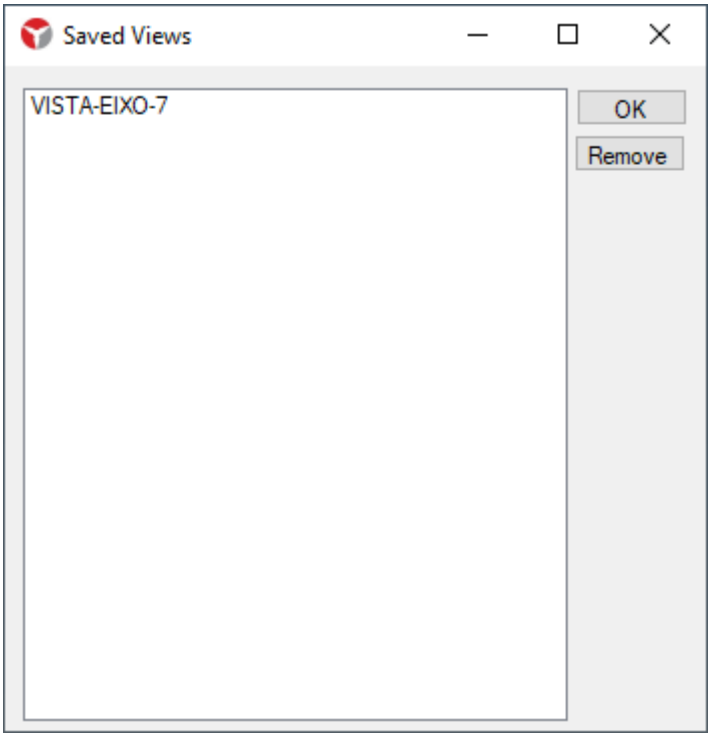


In the 2D view, we have the depth setting (Depth) which is configured above the plane (Above) and below (Behind). These values define which objects will be included in the view.

How to save a View

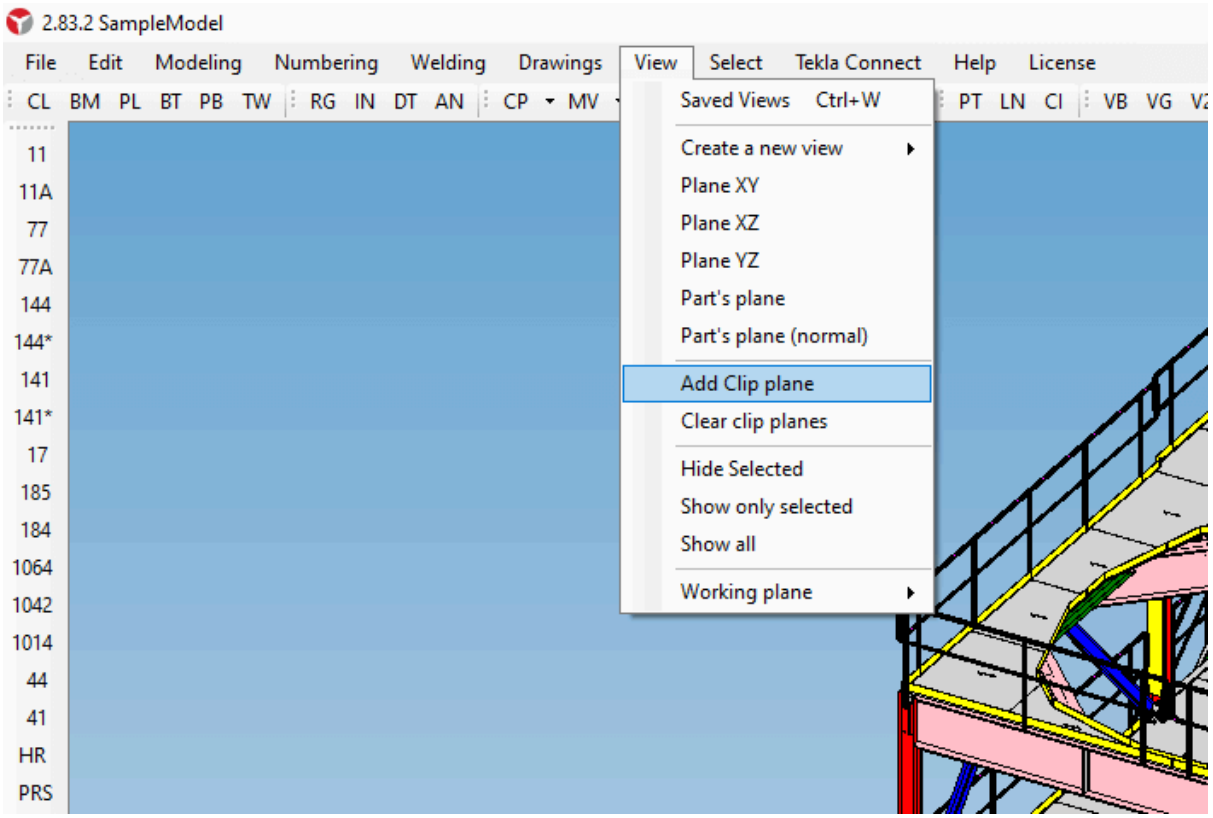


In the text box located on the toolbar, type the name of the view and click SAVE. After saving the view above with the name “VIEW-AXIS-7”, when we open the window with saved views:



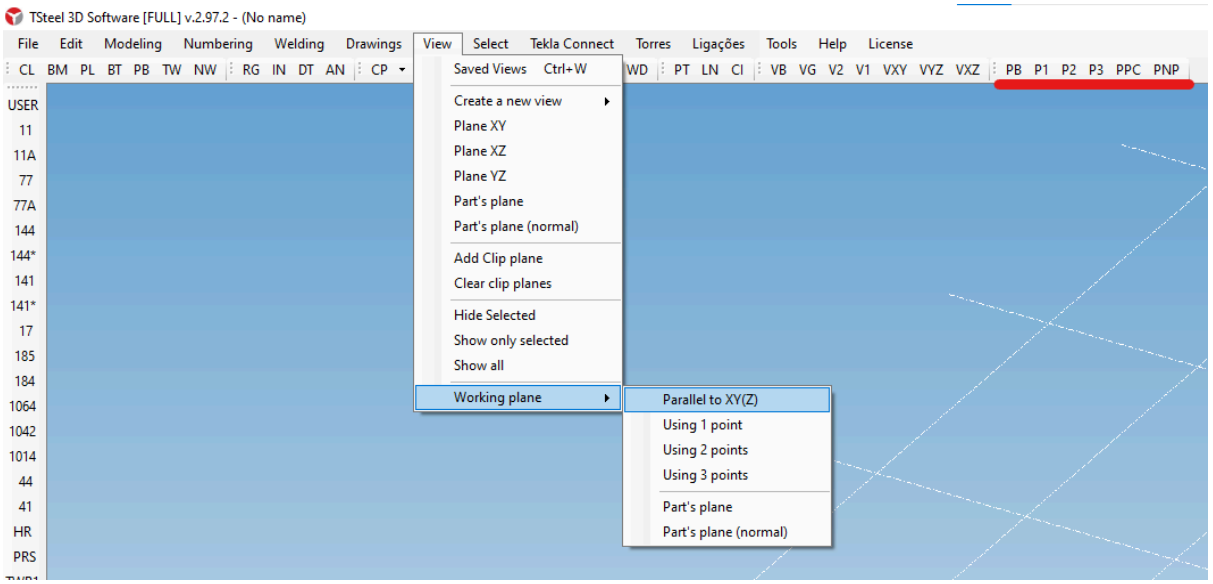
Clip Planes

You can use up to 6 clip planes to limit which parts of the model you want to see.See how to use clip planes in this video.

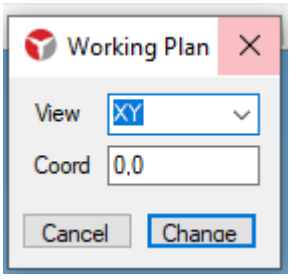


Work plan

The “Working Plan” can be configured via the menu or taskbar:



Parallet to XY(Z) (or PB button): Allows you to position the plane parallel to the global XY, XZ or YZ planes.

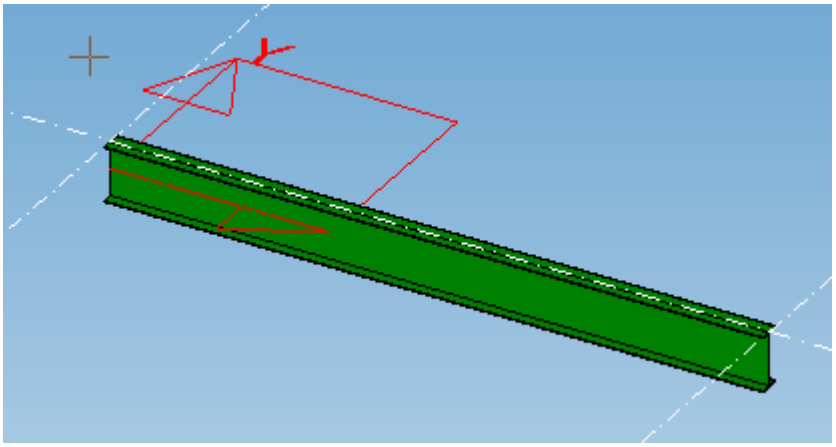


Using 1 Point (P1 button): Moves the origin of the current work plane to the selected point

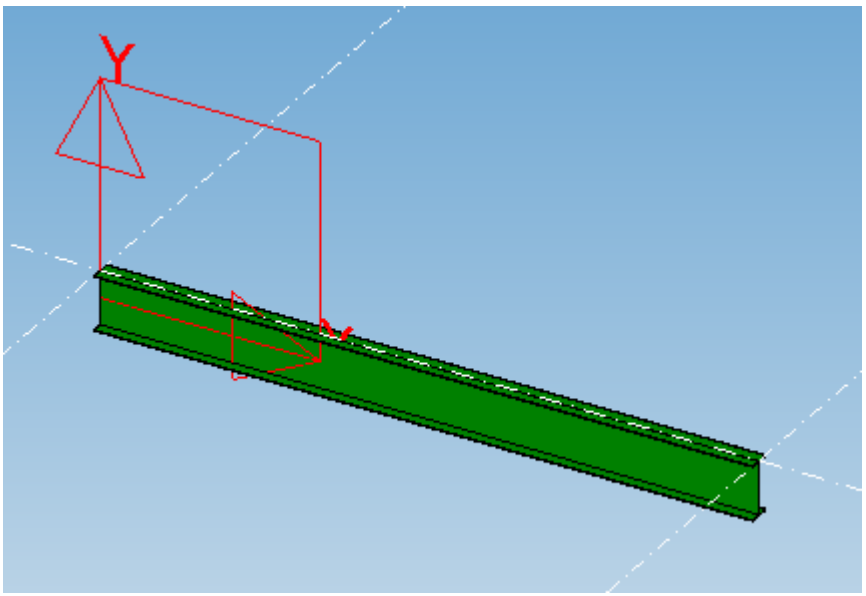
Using 2 Points (P2 button): Shifts the origin of the current plane to the first selected point and aligns the X axis in the direction of the second selected point

Using 3 Points (P3 button): The first point defines the new origin, the second point the direction of the X axis and the third the direction of the Y axis.

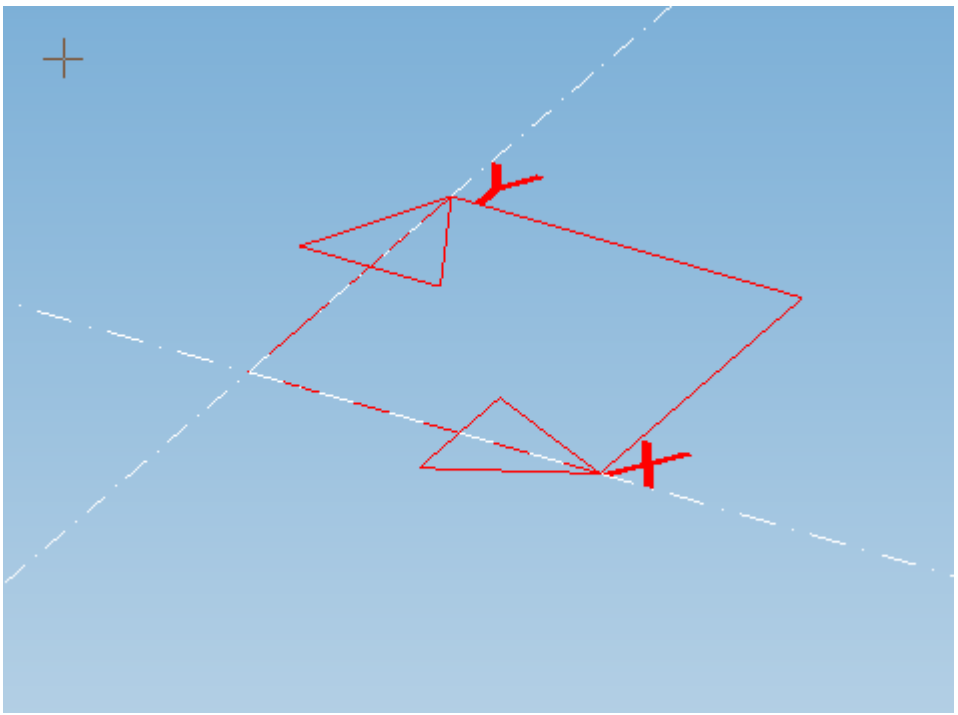
Part's Plane (PPC button): Prompts you to select a part and positions the work plane to match its parent plane.



Part's Normal Plan (PNP): Prompts you to select a part and positions the work plane to coincide with the plane normal to the main plane of the part.



The current location of the work plane is indicated by the red axes. Initially they start at point 0,0,0 and with X,Y,Z axes coinciding with the global axes.



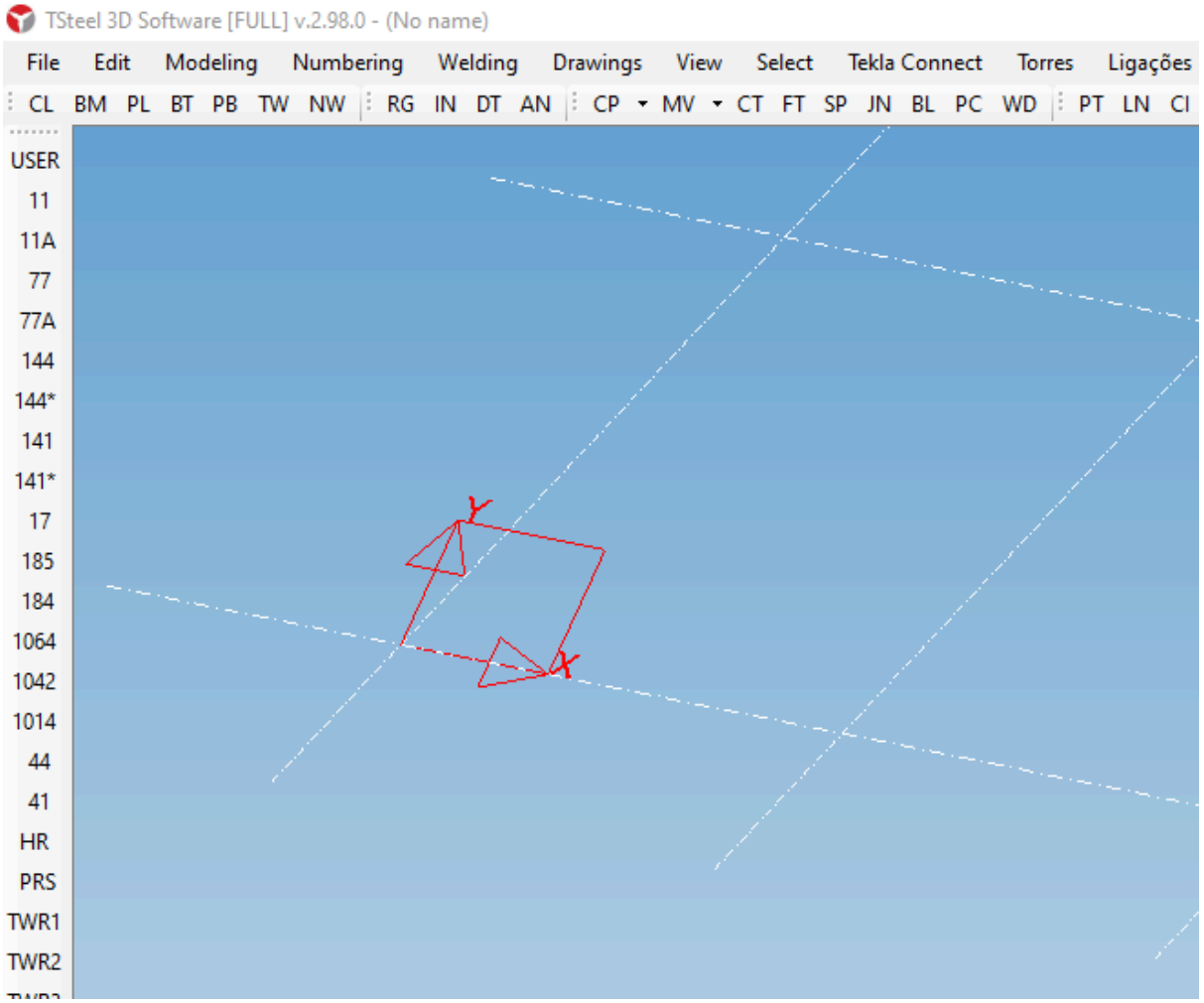
Changing the work plane changes many references in the model, for example:

- Rotation of parts (Top, Front, Back, Below)
- Initial level of column creation
- Points by coordinates

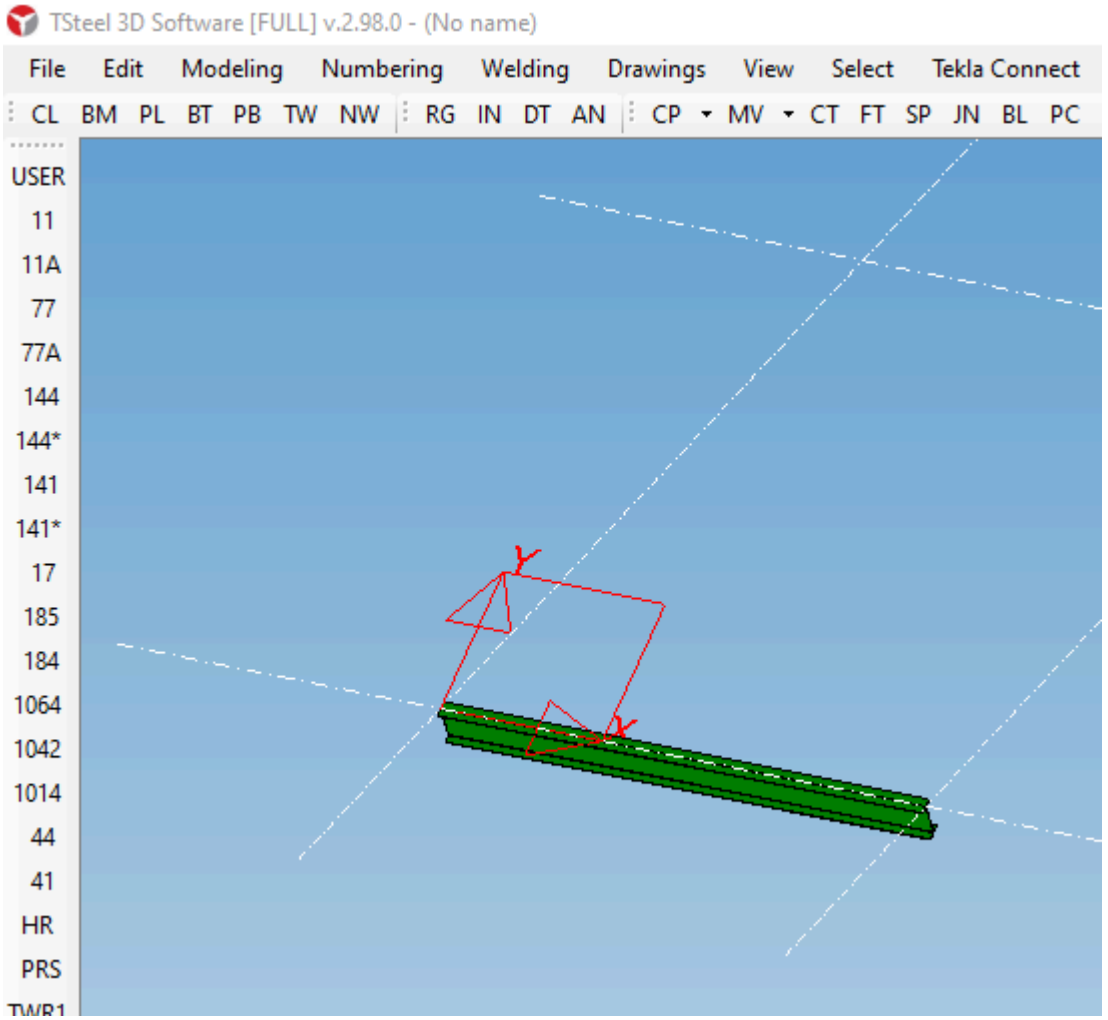
Working on an inclined plane

We will use the work plane to model on an inclined plane, a very common situation in roofing.

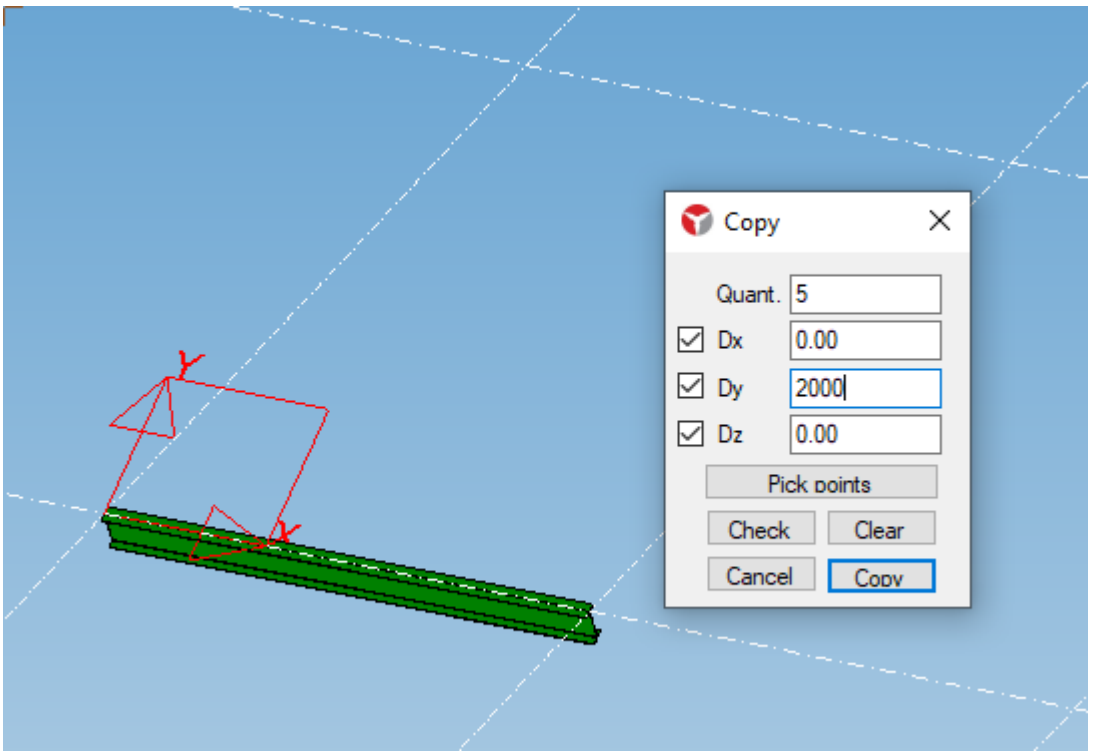
I created an inclined work plane, as shown below:

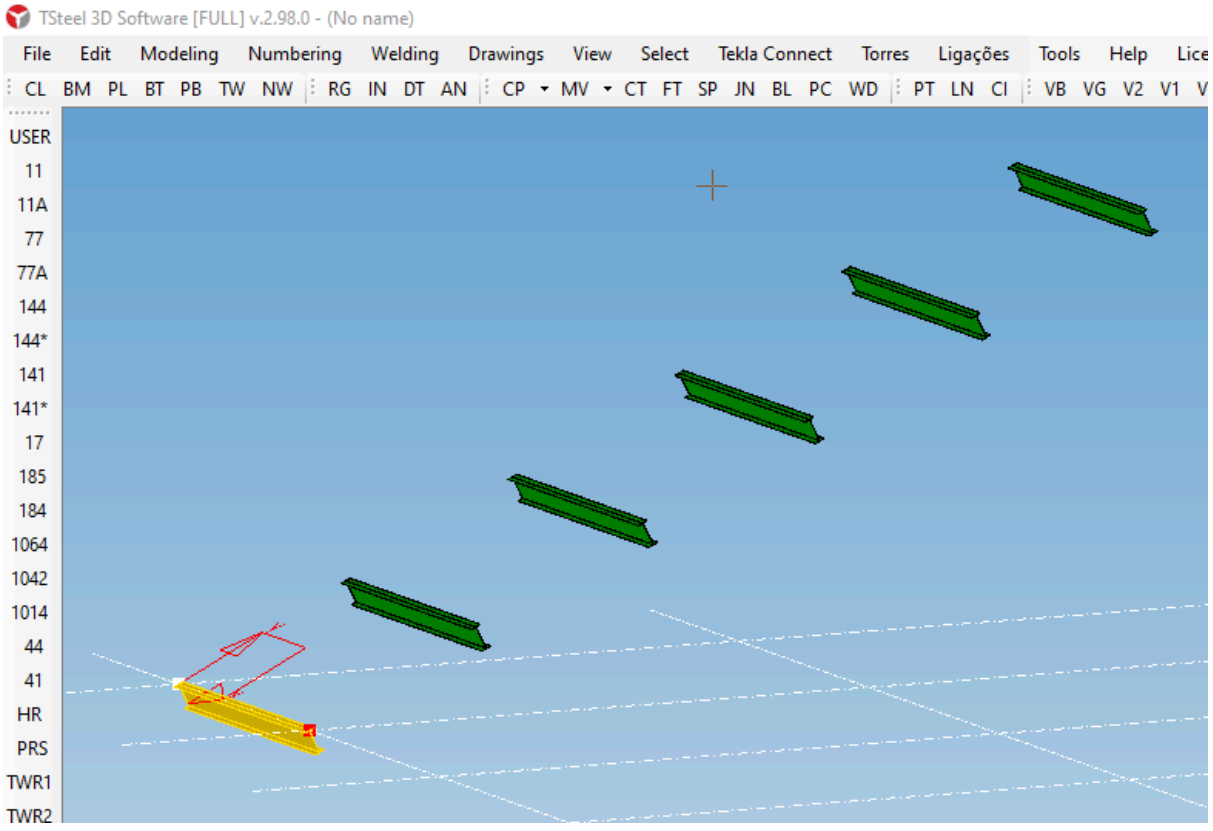


Now I'm going to create a beam along the X axis, see how it comes with the inclination of our work plane:



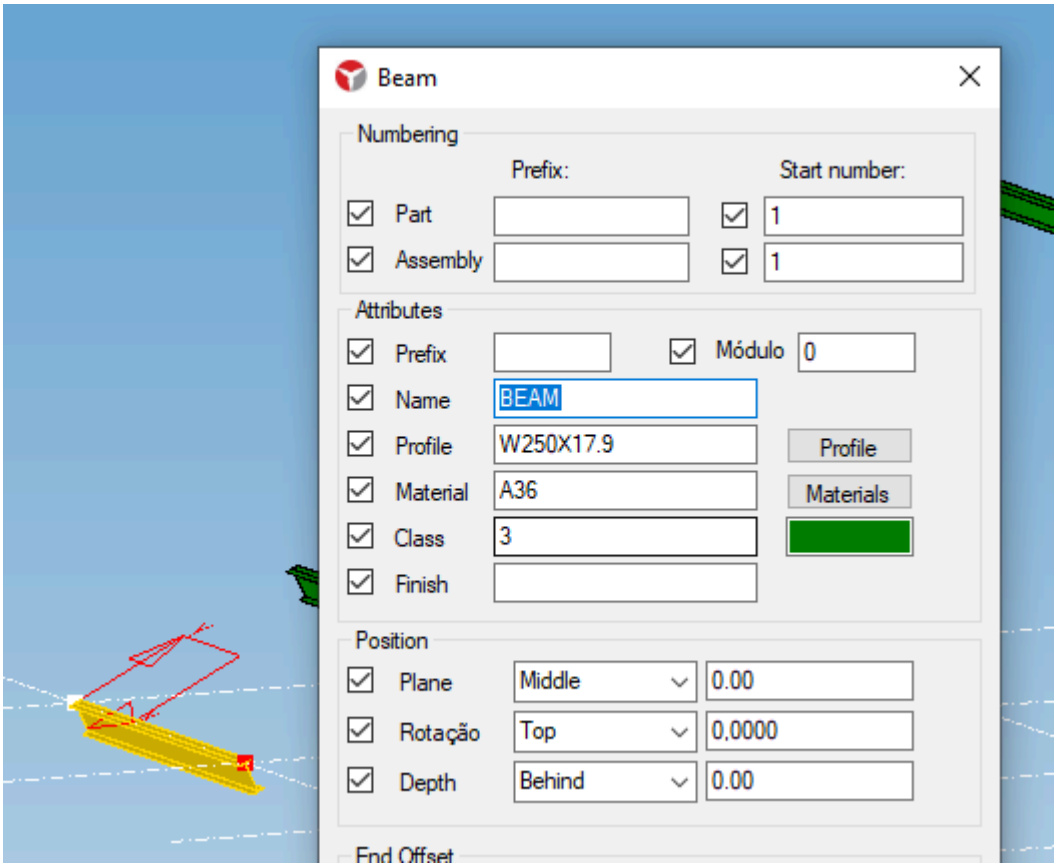
I will make 5 copies of the beam spaced 2000mm apart. Note that I'm giving the distance on the Y axis:



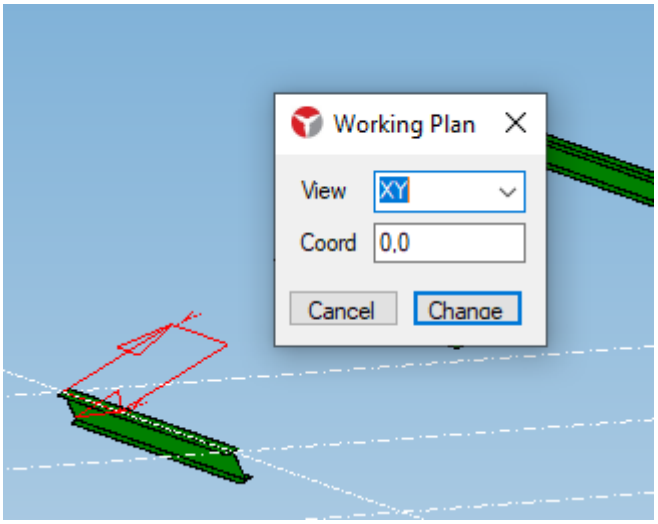


Any construction and editing will consider our work plane as the XY modeling plane

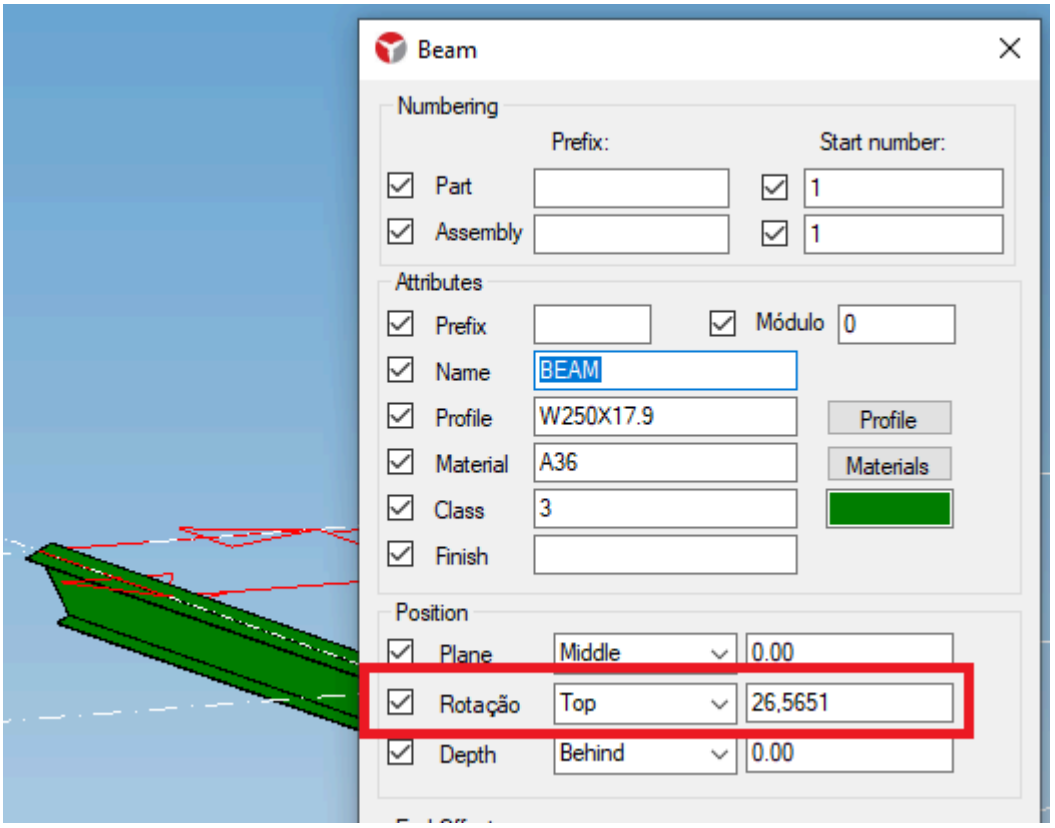
Double click on one of the beams and observe the rotation:



Now, let's change the work plane to the global plane by clicking on the PB button (toolbar)

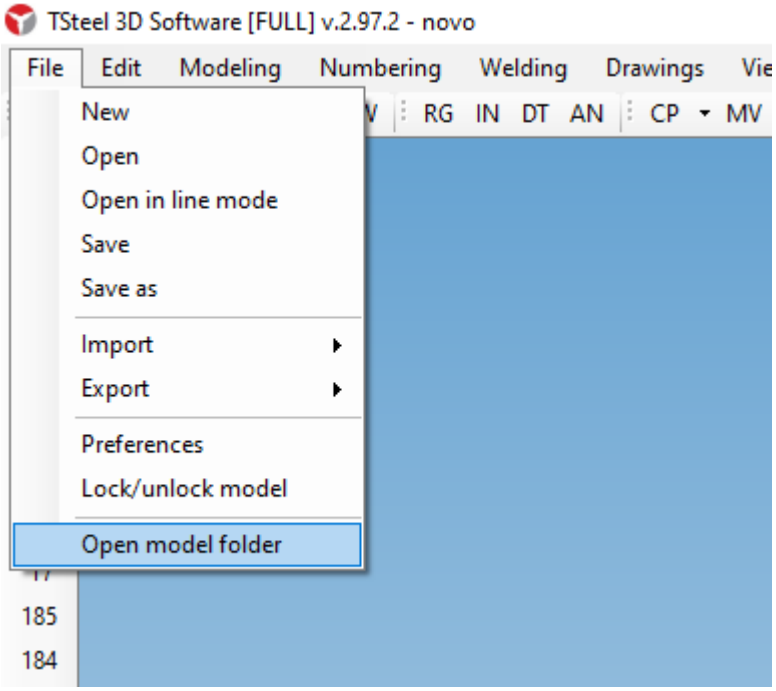


In the image below, notice that the work plane is already in the global plane and see how our beam rotates:

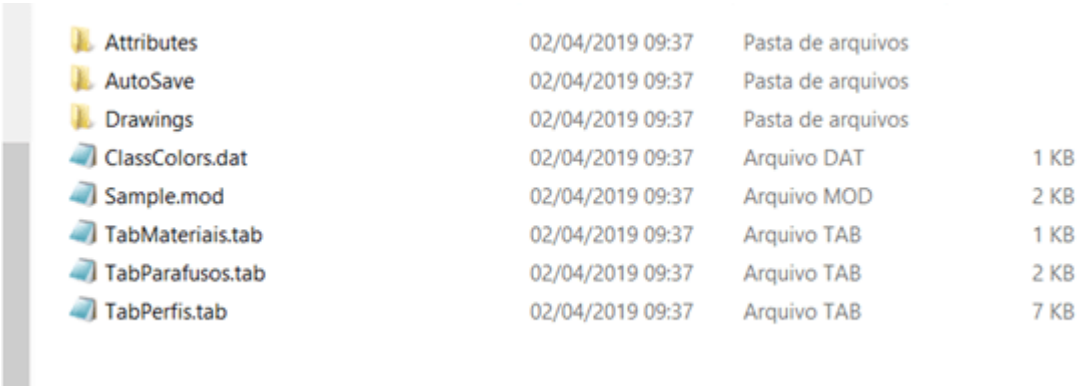

Beam
Numbering
Prefix:
Start number:
Part
1
Assembly
1
Attributes
Prefix
Módulo
0
Name
BEAM
Profile
W250X17.9
Profile
Material
A36
Materials
Class
3
Finish
Position
Plane
Middle
0.00
Rotação
Top
26,5651
Depth
Behind
0.00

Opening and Saving the Model

When you save your model, there is not just one file to save as we will see next. This way, each model has a folder where it organizes the files.
Let's view a model's folder. Use Windows Explorer or the “Open Model Folder” option in the File menu.



A template folder will look like this:



- **ClassColors.dat**: Table with the colors used for each class. You can copy this file to other models if you want to use the same color setting.
- **Sample.mod**: It is the file with the model data.
- **TabMateriais.tab**: Table (catalog) ofmaterials.
- **TabParafusos.tab**: Table (catalog) of screws.
- **TabPerfis.tab**: Table (catalog) ofprofiles.
- **Attributes Directory**: Used to save various configuration files. Do not copy these files to another model.

- **AutoSave Directory**: Directory where the “autosave” files are saved. Auto save is configured through the File>Preferences Menu.

Other files you can find in the directory:

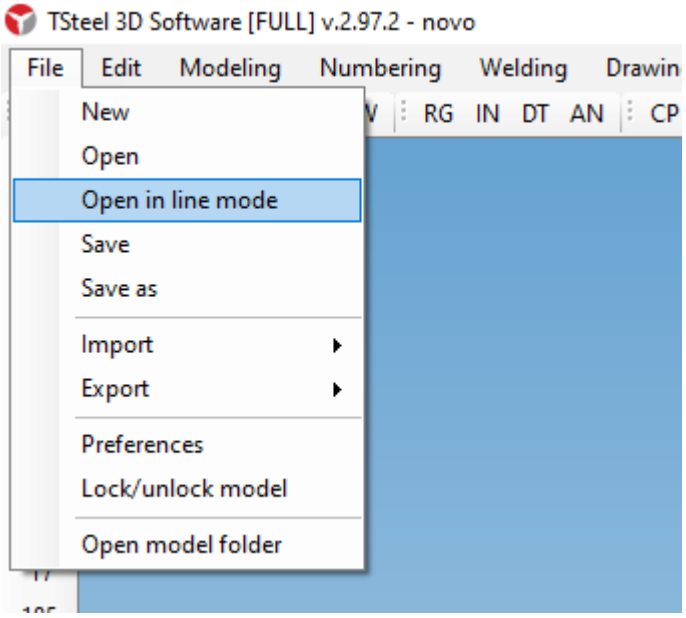
- **Sample.backup:** It is a file with the backup of the last opened model. If something unexpected occurs, you can rename this file and return to the last saved template.
- **Sample.locked:** It is a file created when the model is opened and deleted when the model is closed. TSteel 3D uses this file to know who opened the model and prevent two people from overwriting a model when opened over a network. If you are receiving a message that the template is being used improperly, delete this file and log in again.

TSteel will also create separate folders for:

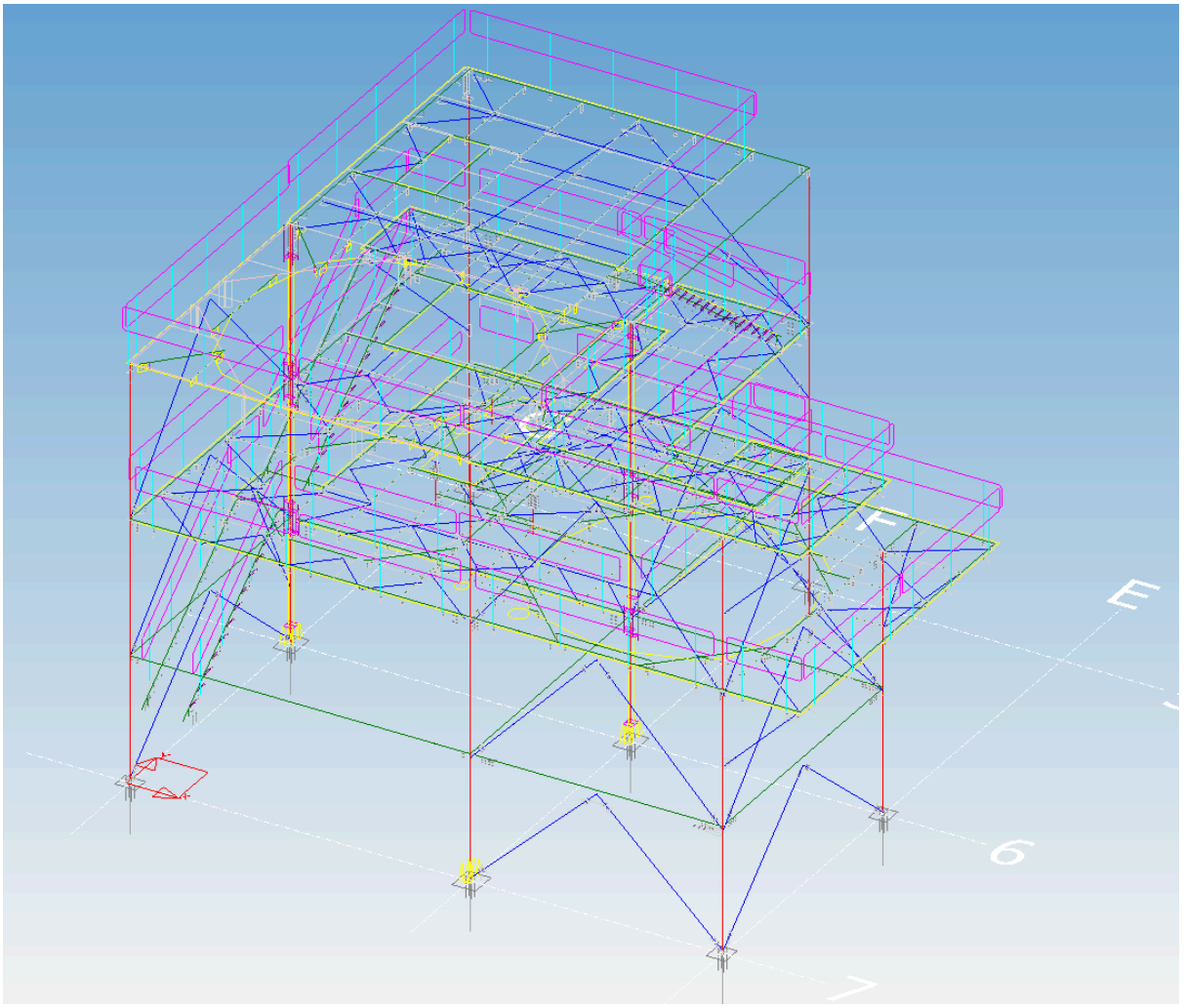
- Automatic sketch drawings
- Automatic assembly drawings
- CNC Files (DSTV)

Open model in line mode

You can view the model as a line, we saw this in the 3D screen settings, but you can also open the file in line mode.



See below the example model opened in line mode:

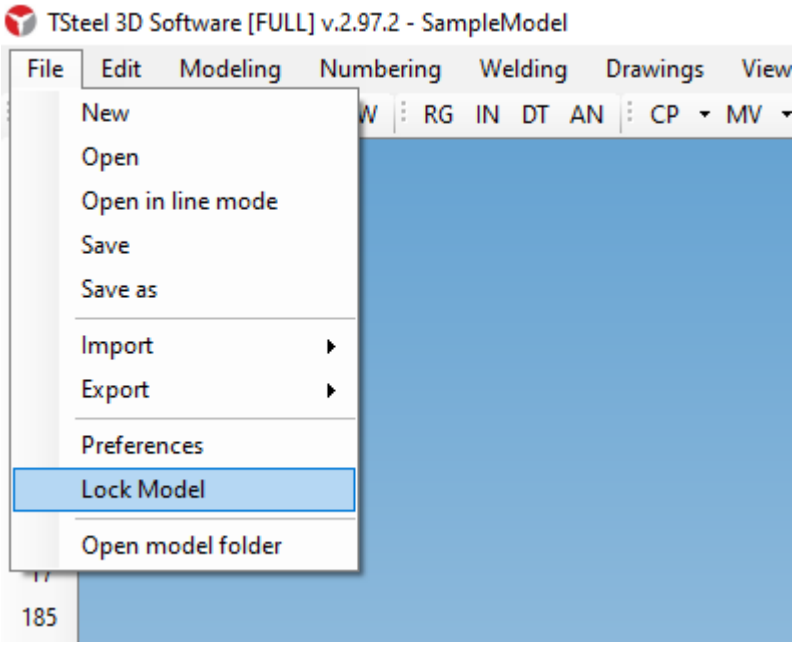


This option is useful for opening very large models. You can select parts of the model to view in 3D and thus save memory and time.

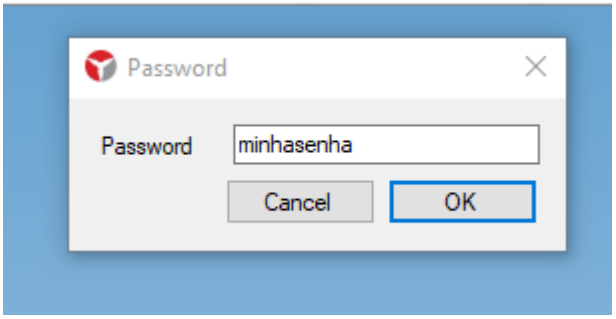
Lock a model (Lock/Unlock)

TSteel allows you to lock a model with a password. A locked model does not allow changes or exports to other formats. You can share without worrying about the template being used in an unauthorized way.

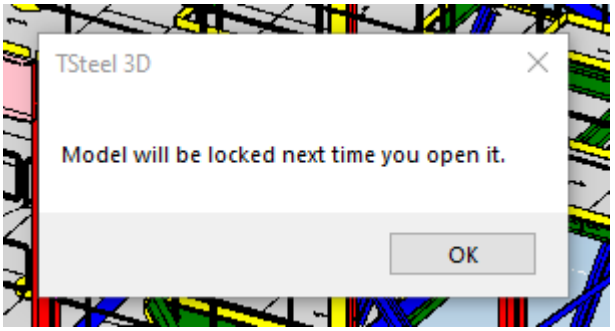
To lock the model:



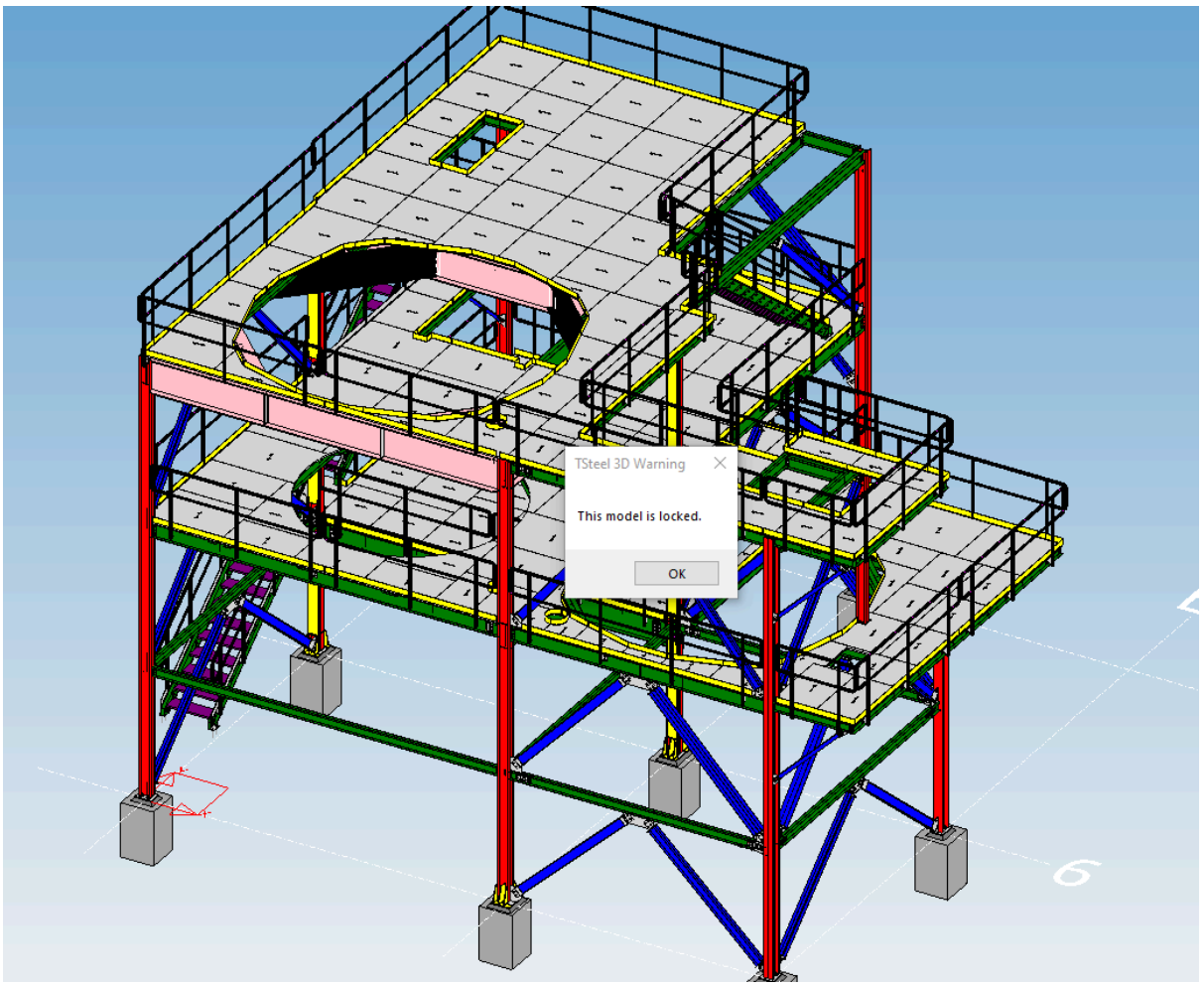
TSteel will ask for a password:



After entering the password, you receive the warning:

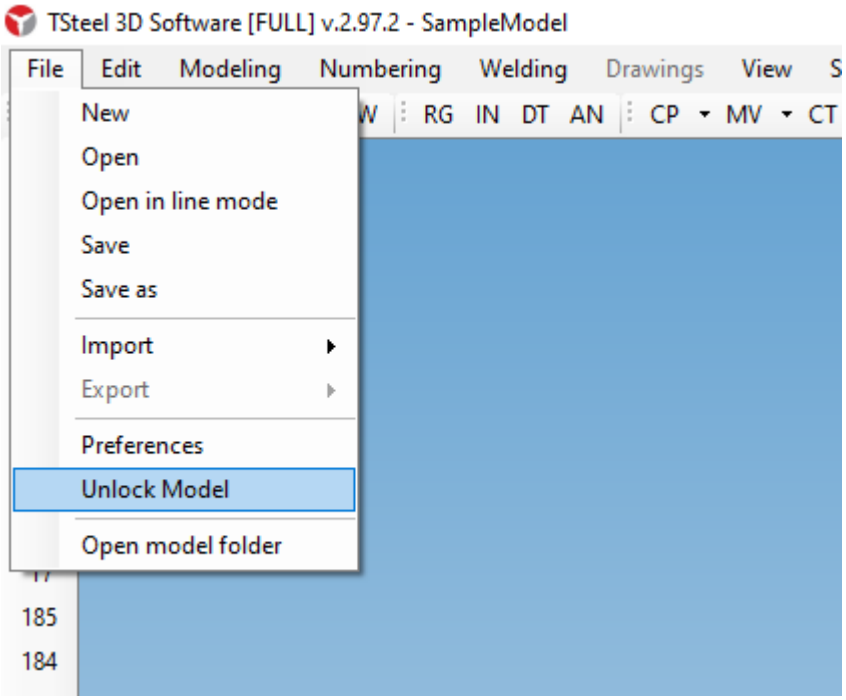


Open the model again, you will receive the message:

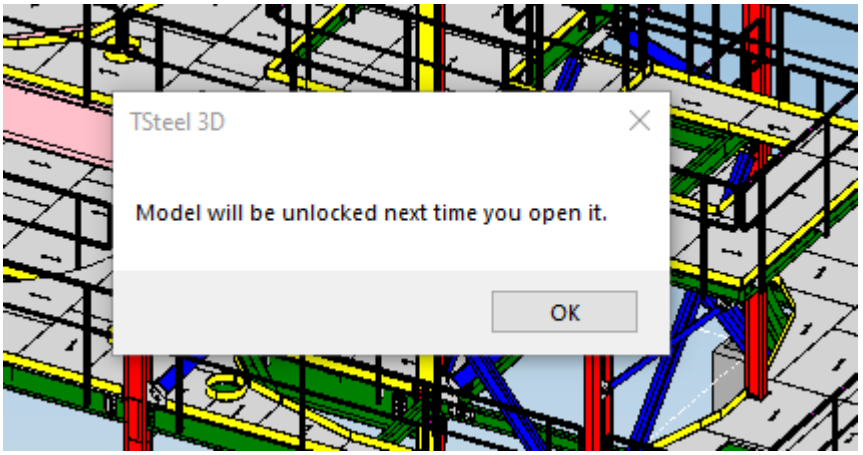


Carry out tests and see that it is not possible to save or export the model. It is for viewing purposes only.

To unblock:



Enter the password and you will receive the message:



There is no way to recover the password, if you lose it you will not be able to unlock the model. Consider having a copy unlocked before locking your template for sharing.

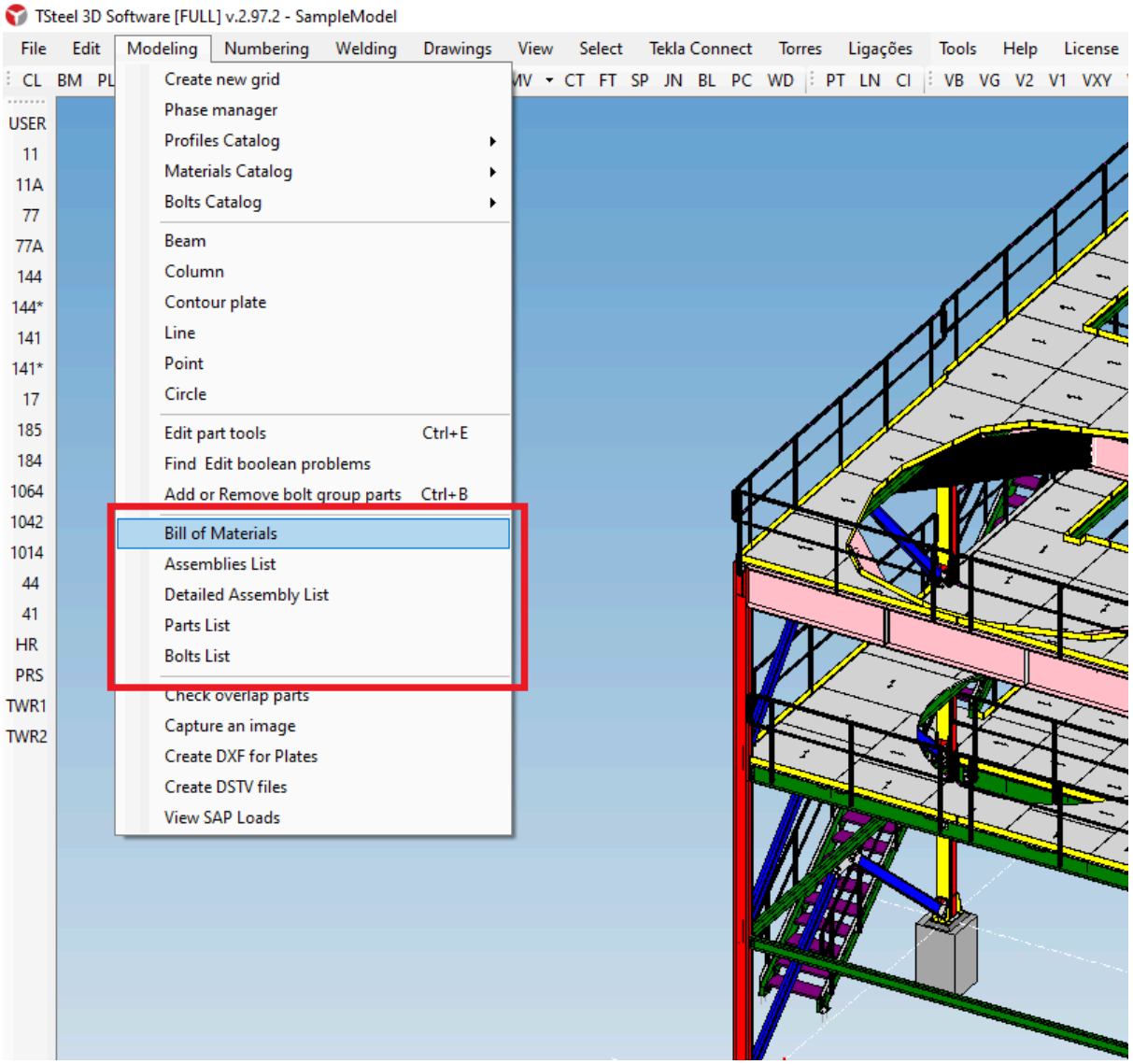
Material lists

Extracting Bills of Materials

There are 5 different types of BOMs available:

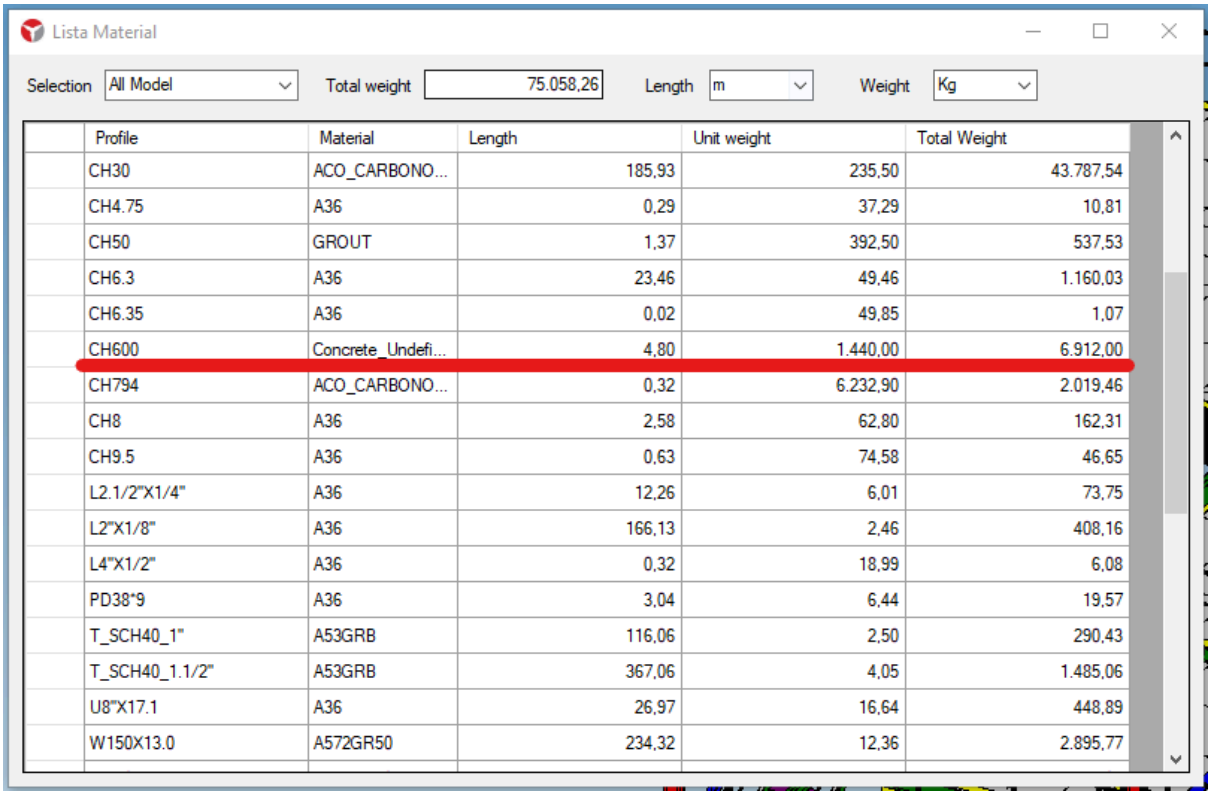
- Bill of Materials
- Assembly List
- Detailed Assembly list
- Parts List
- Bolt Lists

Material lists, shown in the list window, can be selected, copied (CTRL+C) and pasted into a spreadsheet (CTRL+V). From the spreadsheets, you can format and change as you wish.



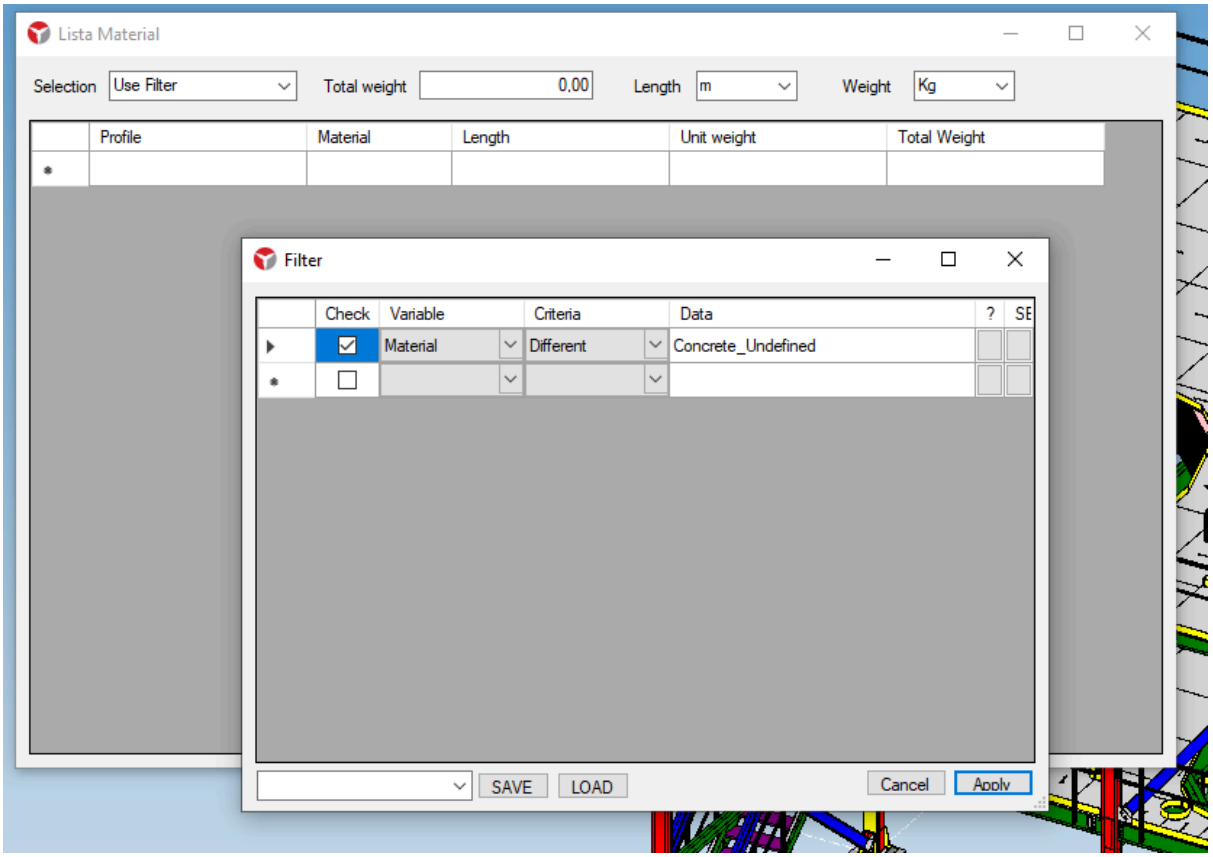
List of materials

The list of materials, as the name suggests, is a summary of each type of material used in your model, an example would be:

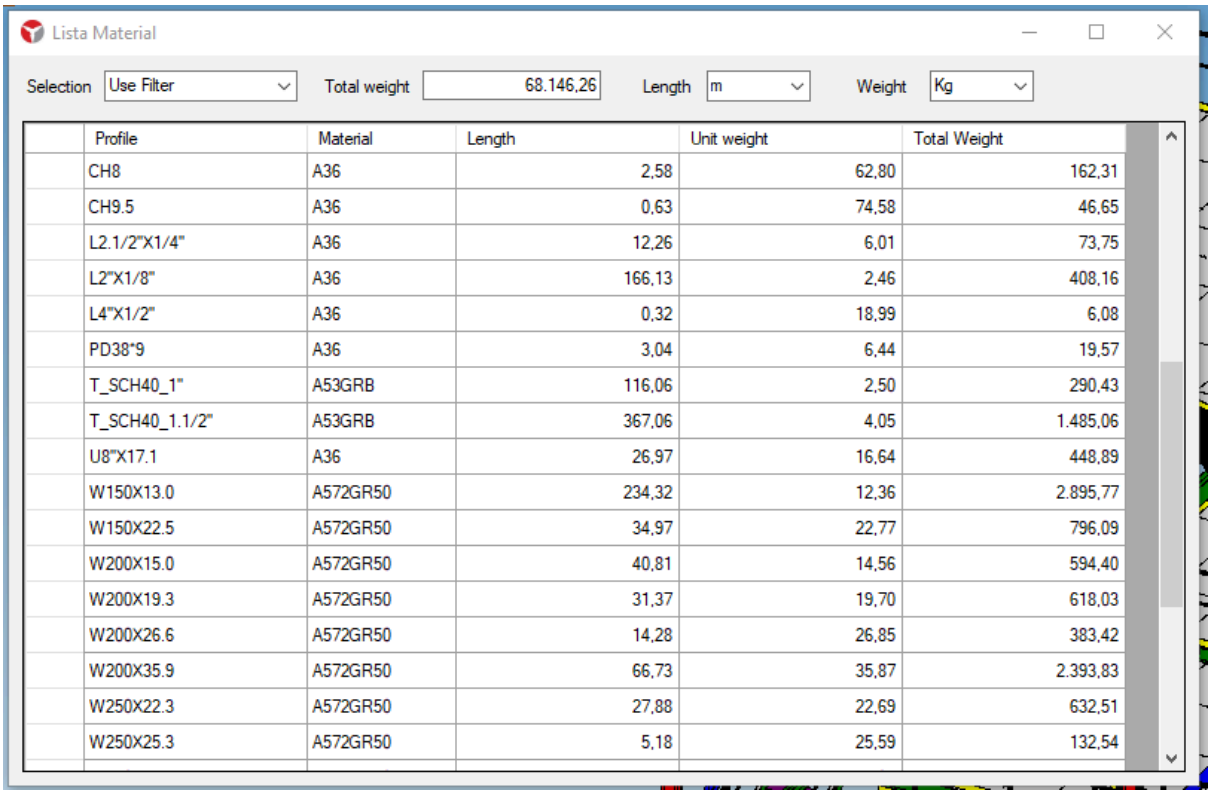


Profile	Material	Length	Unit weight	Total Weight
CH30	ACO_CARBONO...	185,93	235,50	43.787,54
CH4.75	A36	0,29	37,29	10,81
CH50	GROUT	1,37	392,50	537,53
CH6.3	A36	23,46	49,46	1.160,03
CH6.35	A36	0,02	49,85	1,07
CH600	Concrete_Undefi...	4,80	1.440,00	6.912,00
CH794	ACO_CARBONO...	0,32	6.232,90	2.019,46
CH8	A36	2,58	62,80	162,31
CH9.5	A36	0,63	74,58	46,65
L2.1/2"X1/4"	A36	12,26	6,01	73,75
L2"X1/8"	A36	166,13	2,46	408,16
L4"X1/2"	A36	0,32	18,99	6,08
PD38*9	A36	3,04	6,44	19,57
T_SCH40_1"	A53GRB	116,06	2,50	290,43
T_SCH40_1.1/2"	A53GRB	367,06	4,05	1.485,06
U8"X17.1	A36	26,97	16,64	448,89
W150X13.0	A572GR50	234,32	12,36	2.895,77

Note that in the example above, the concrete material used to model the shoes was included. We don't want this material on our list. At the top left, change from "All Model" to "Use Filter" and delete the concrete material (see below).



Now our list will be as we want:



Profile	Material	Length	Unit weight	Total Weight
CH8	A36	2,58	62,80	162,31
CH9.5	A36	0,63	74,58	46,65
L2.1/2"X1/4"	A36	12,26	6,01	73,75
L2"X1/8"	A36	166,13	2,46	408,16
L4"X1/2"	A36	0,32	18,99	6,08
PD38*9	A36	3,04	6,44	19,57
T_SCH40_1"	A53GRB	116,06	2,50	290,43
T_SCH40_1.1/2"	A53GRB	367,06	4,05	1.485,06
U8"X17.1	A36	26,97	16,64	448,89
W150X13.0	A572GR50	234,32	12,36	2.895,77
W150X22.5	A572GR50	34,97	22,77	796,09
W200X15.0	A572GR50	40,81	14,56	594,40
W200X19.3	A572GR50	31,37	19,70	618,03
W200X26.6	A572GR50	14,28	26,85	383,42
W200X35.9	A572GR50	66,73	35,87	2.393,83
W250X22.3	A572GR50	27,88	22,69	632,51
W250X25.3	A572GR50	5,18	25,59	132,54

Set List

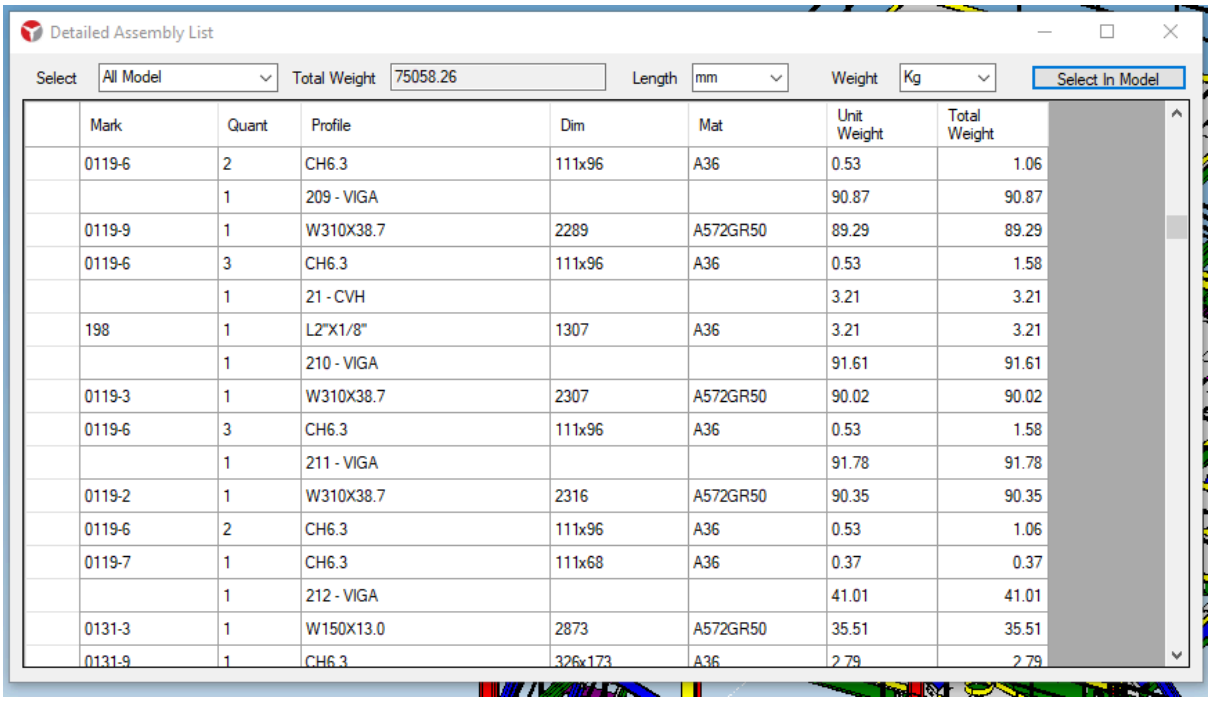
Mark	Quant	Main Profile	Name	Weight
1	19	CH794	DEGRAU	959.24
10	1	ACORED_3/4"	CHUMBADOR	1.34
100	1	W200X35.9	COLUNA	352.12
101	1	W150X13.0	VIGA	10.66
102	1	L2"X1/8"	CVH	4.57
103	4	W150X13.0	VIGA	72.55
104	1	L2"X1/8"	CVH	4.26
105	1	L2"X1/8"	CVH	4.14
106	1	W150X13.0	VIGA	7.38
107	1	L2"X1/8"	CVH	4.24
108	1	L2"X1/8"	CVH	4.21
109	1	W150X22.5	COLUNA	72.61
11	1	ACORED_3/4"	CHUMBADOR	1.34
110	2	W150X22.5	COLUNA	29.03
111	1	W200X15.0	VIGA	31.08
112	1	W310X52.0	VIGA	315.08

This list, shows one set per line, requires the model to be numbered (marked), you will receive a warning if unnumbered sets are found.

Note that in the upper right corner, there is a "Select in Model" button. This button will select the set selected in the material list in the model. This is a tool that allows the user to easily find any set by its number (mark).

Detailed set list

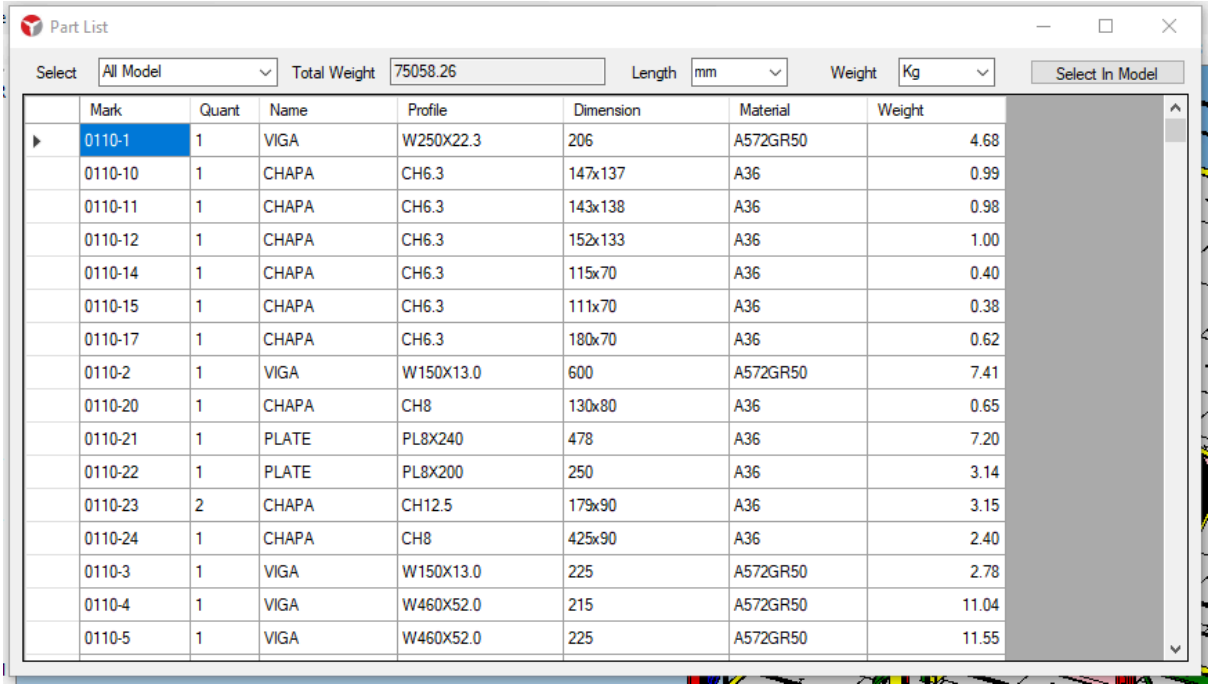
In addition to the existing sets, the components of each set are shown.



Mark	Quant	Profile	Dim	Mat	Unit Weight	Total Weight
0119-6	2	CH6.3	111x96	A36	0.53	1.06
	1	209 - VIGA			90.87	90.87
0119-9	1	W310X38.7	2289	A572GR50	89.29	89.29
0119-6	3	CH6.3	111x96	A36	0.53	1.58
	1	21 - CVH			3.21	3.21
198	1	L2"X1/8"	1307	A36	3.21	3.21
	1	210 - VIGA			91.61	91.61
0119-3	1	W310X38.7	2307	A572GR50	90.02	90.02
0119-6	3	CH6.3	111x96	A36	0.53	1.58
	1	211 - VIGA			91.78	91.78
0119-2	1	W310X38.7	2316	A572GR50	90.35	90.35
0119-6	2	CH6.3	111x96	A36	0.53	1.06
0119-7	1	CH6.3	111x68	A36	0.37	0.37
	1	212 - VIGA			41.01	41.01
0131-3	1	W150X13.0	2873	A572GR50	35.51	35.51
0131-9	1	CH6.3	326x173	A36	2.79	2.79

Parts lists

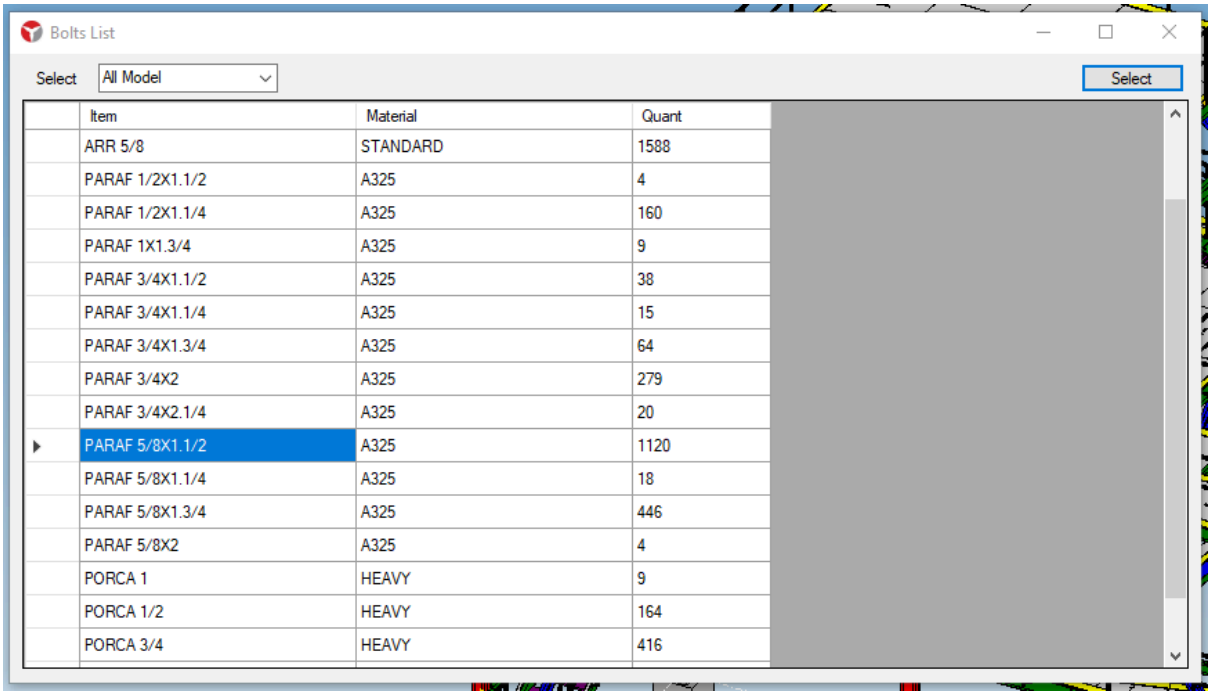
Just like with sets, you need to have the numbered model. Unmarked parts do not appear in the report. Here there is also the "Select in Model" button that allows you to select the part selected from the list in the model.



Mark	Quant	Name	Profile	Dimension	Material	Weight
0110-1	1	VIGA	W250X22.3	206	A572GR50	4.68
0110-10	1	CHAPA	CH6.3	147x137	A36	0.99
0110-11	1	CHAPA	CH6.3	143x138	A36	0.98
0110-12	1	CHAPA	CH6.3	152x133	A36	1.00
0110-14	1	CHAPA	CH6.3	115x70	A36	0.40
0110-15	1	CHAPA	CH6.3	111x70	A36	0.38
0110-17	1	CHAPA	CH6.3	180x70	A36	0.62
0110-2	1	VIGA	W150X13.0	600	A572GR50	7.41
0110-20	1	CHAPA	CH8	130x80	A36	0.65
0110-21	1	PLATE	PL8X240	478	A36	7.20
0110-22	1	PLATE	PL8X200	250	A36	3.14
0110-23	2	CHAPA	CH12.5	179x90	A36	3.15
0110-24	1	CHAPA	CH8	425x90	A36	2.40
0110-3	1	VIGA	W150X13.0	225	A572GR50	2.78
0110-4	1	VIGA	W460X52.0	215	A572GR50	11.04
0110-5	1	VIGA	W460X52.0	225	A572GR50	11.55

Screw list

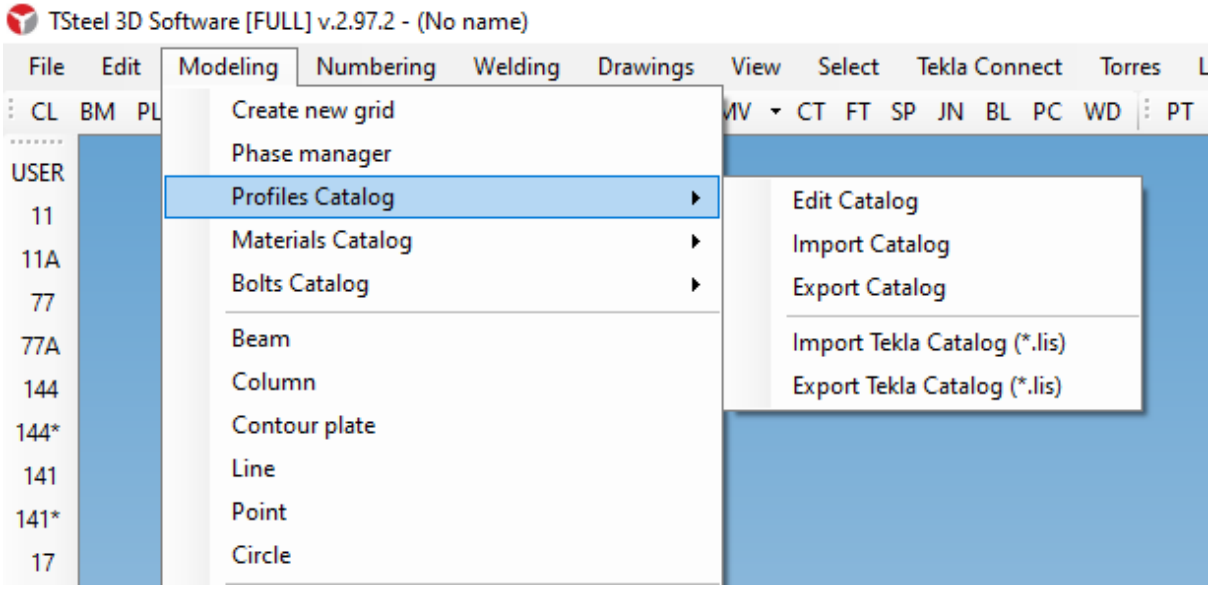
Note that there is a "Select" button, which selects all screws with the gauge selected in the report in the model.



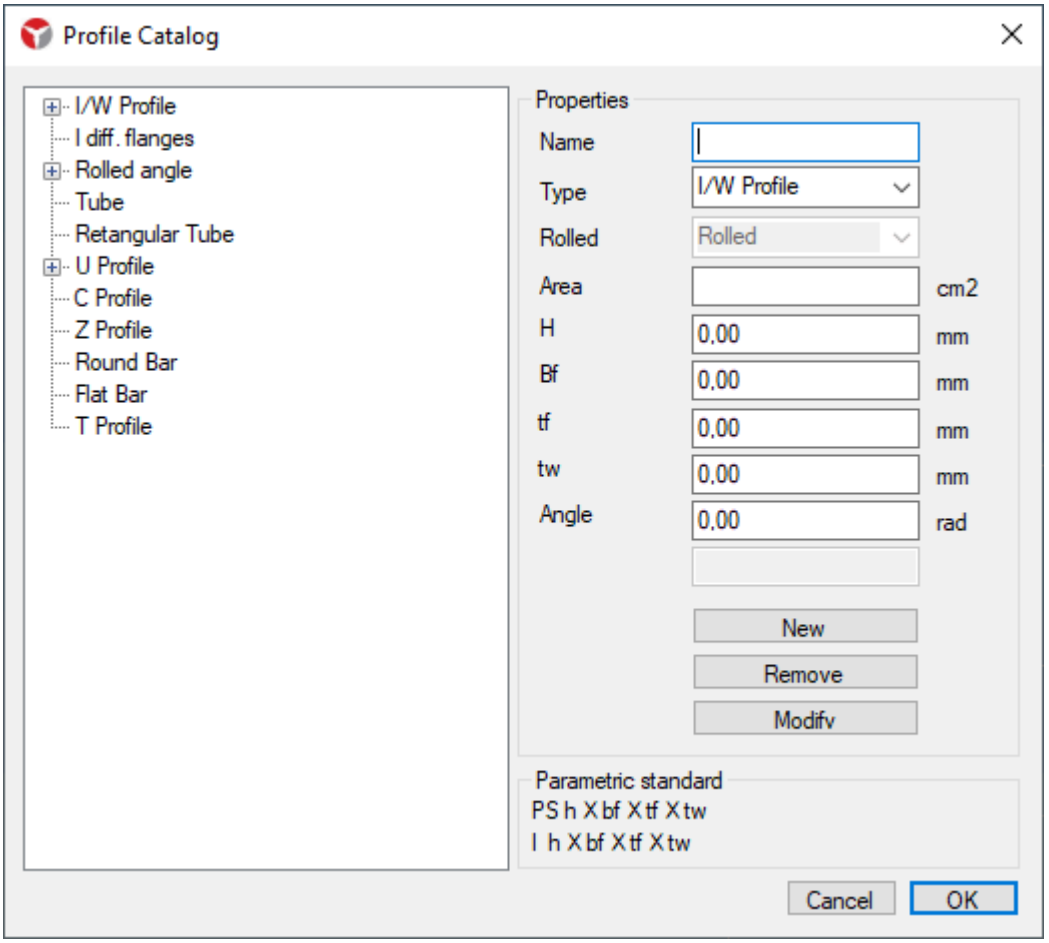
Catalogs

Profile catalogs

Each model has a catalog of profiles where the known gauges and their properties are defined.
In the catalog menu you will find editing options (add, change and delete items), import and export catalogs to be used in other models andmatter and export catalogs in LIS format to access with Tekla.



Select “Edit Catalog” and the program will open a window for maintaining catalog profiles:



Parametric profiles

You can also define a profile parametrically. Note that in the catalog window, at the bottom, there is the parametric rule for creating type I or W profiles. When you change the profile typology (Type), the new parametric definition rule will be displayed.
Some examples of parametric profiling:

- PL thickness x width (plates)
- CH thickness x width (plates)
- PLT thickness x width (plates)
- EU Height x Table x Flap x Thickness (Stiffened U Profile)
- U Height x Table x Thickness (U Profile)
- TQ Side x Thickness (Square tube)
- TR Height x Width x Thickness (Rectangular tube)
- TB Diameter x Thickness (Round tube)
- BR Diameter (Round bar)
- I Height x Table x Length. Table x Esp. Soul (Profile I)
- PS Height x Table x Length. Table x Esp. Alma (Profile I Soldier)

- Z Height x Table x Flap x Thickness (Z Profile)
- CONC Height x Width (Concrete Beam or Column)

How to transform parametric profiles into catalog profiles

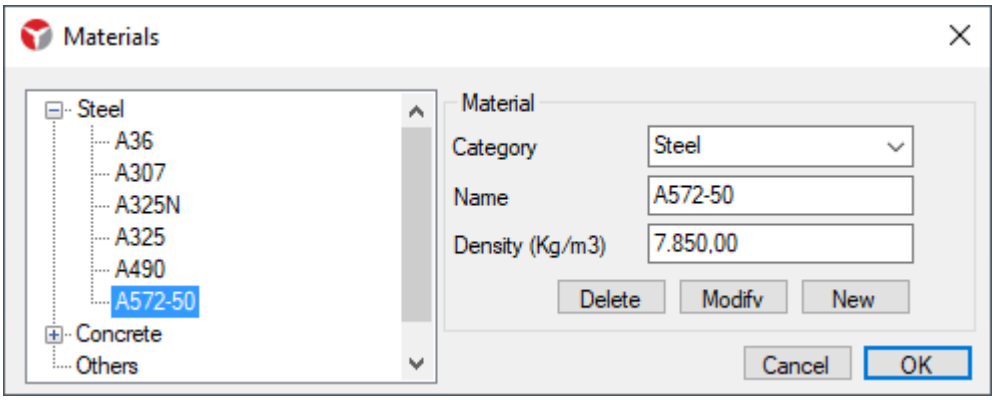
You will notice that in the catalog, the parametric profiles are blue. If you need the program to treat this profile as non-parametric and just save its properties instead of calculating: Select the profile in the catalog and click the New button.
A catalog profile allows you to edit measurements and the cross-sectional area, while parametric profiles use the automatic calculation values.

How to change the weight of a profile

The weight kg/m of a profile is calculated as a function of its cross-sectional area and the density of the material used. In other words, if any adjustment to the weight of a profile is necessary, change the cross-sectional area in the catalogue.
If you want a zero-weight profile, for reference parts, for example, assign them a special material with zero density.
When you register a new profile, you can enter the cross-sectional area, or leave it blank so that the program calculates the area for you.

Material catalogs

See below the window for creating and editing model materials:



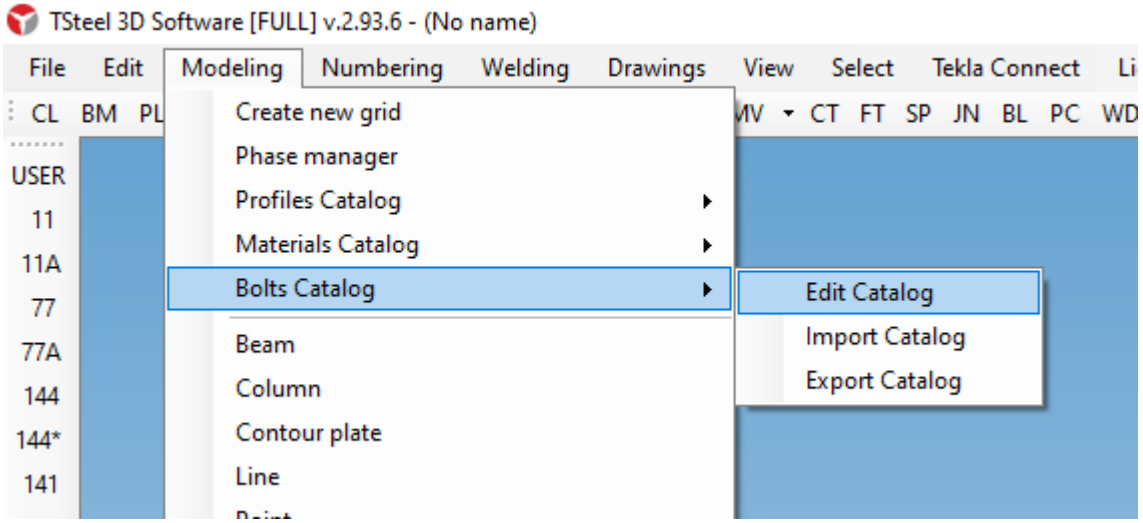
Screw catalogs

For TSteel to model a screw correctly, you need to know:

- Screw quality
- Screw dimensions: head, body, thread, lengths available for each gauge
- Nut quality and dimensions
- Washer quality and dimensions

So, when you decide to use a 1/2" A325N screw, TSteel can define the length of the screw, nuts and washers.

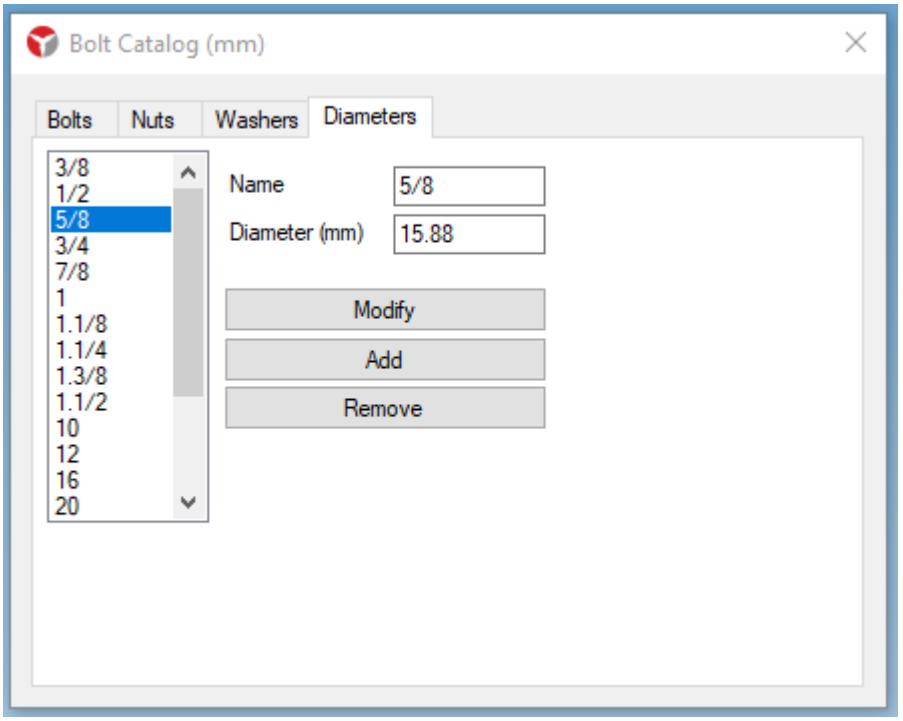
To edit the screw catalogue, access the form via:



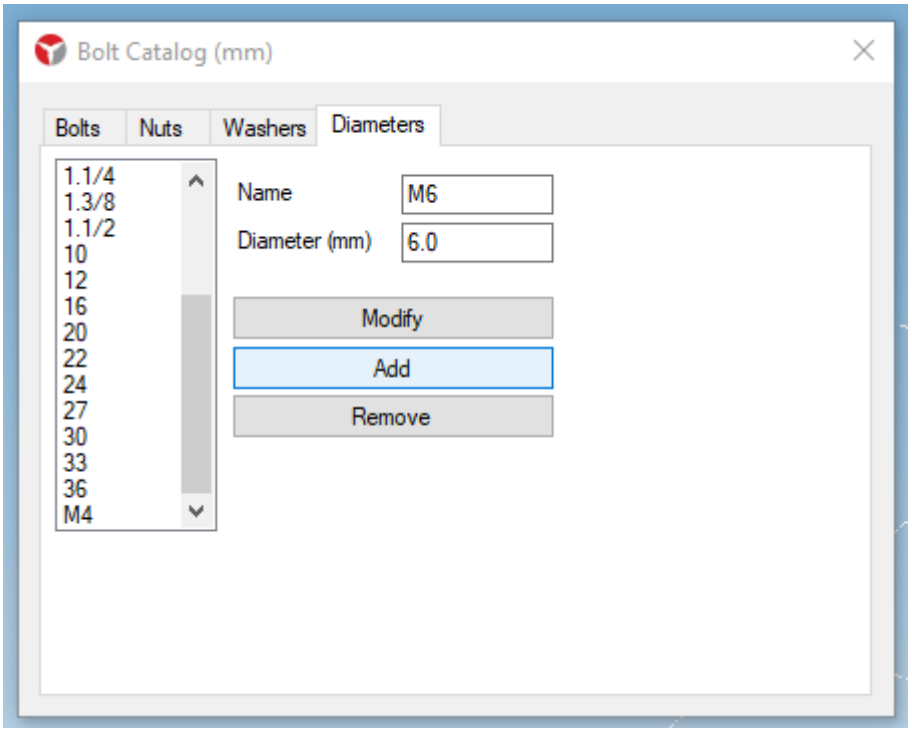
To be able to add new screws to the catalogue, we must follow the following sequence:

Registration of diameters

Each screw diameter has a name and its measurement, see the example below where the diameter named 5/8 has the value of 15.88 mm.

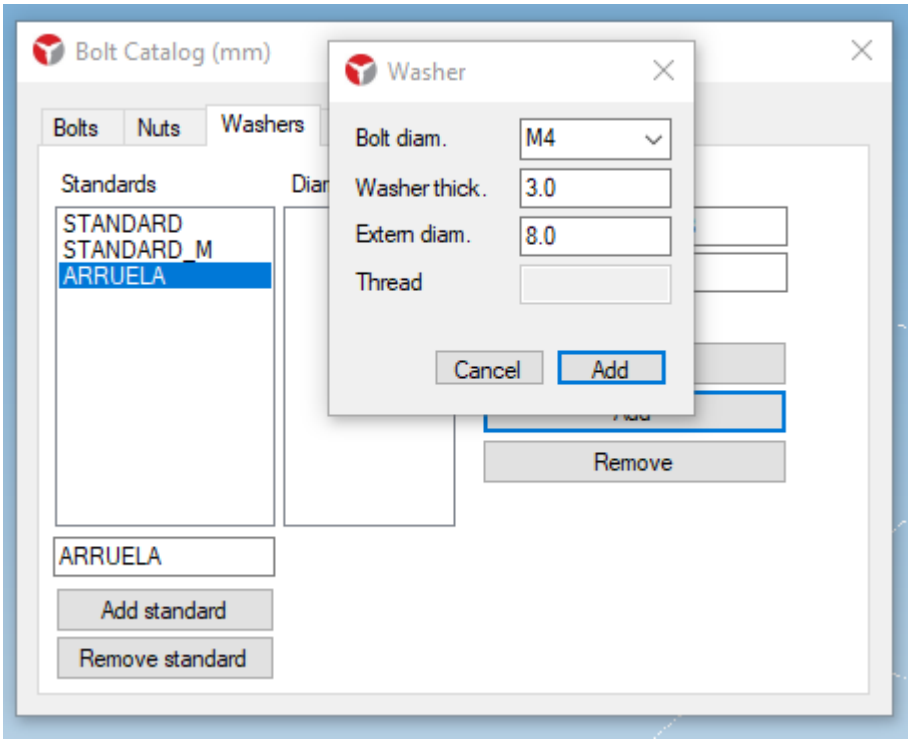


Let's add 2 new diameters, which I'll call M4 and M6:

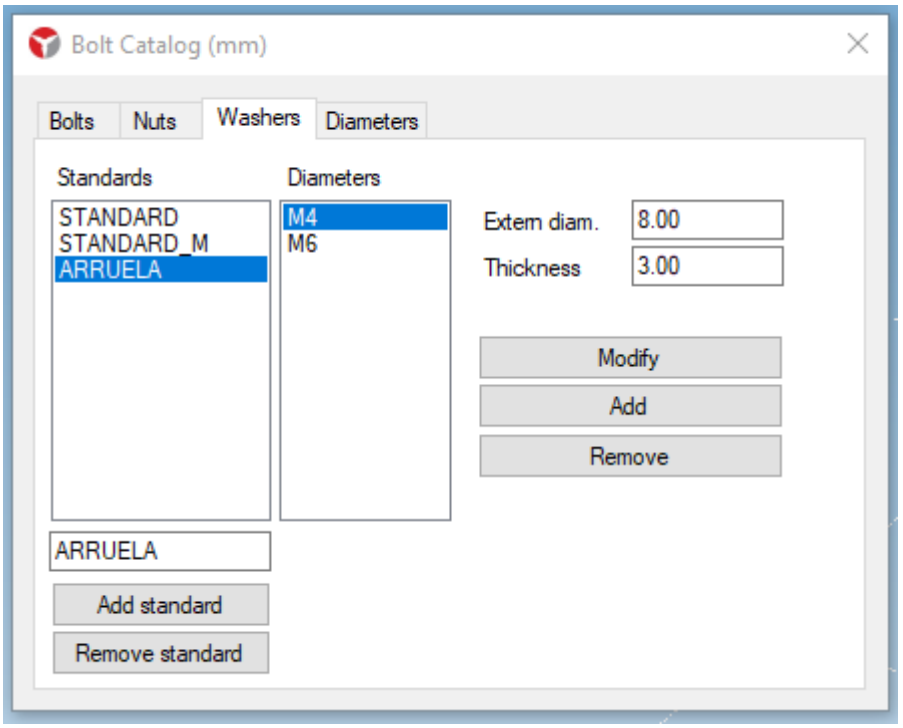


Adding Washers

After we add the new diameters, we need to inform what the washers will be like for these diameters.
For our example, we will create a new washer pattern, which we will call Washer and we will include the two new diameters (M4 and M6).

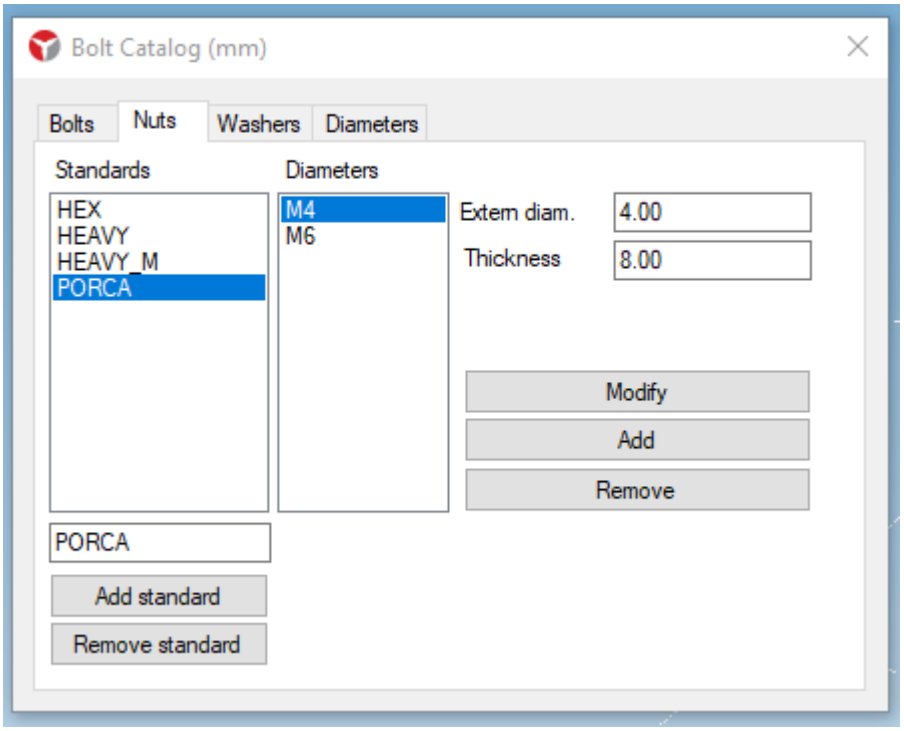


We will have this configuration:



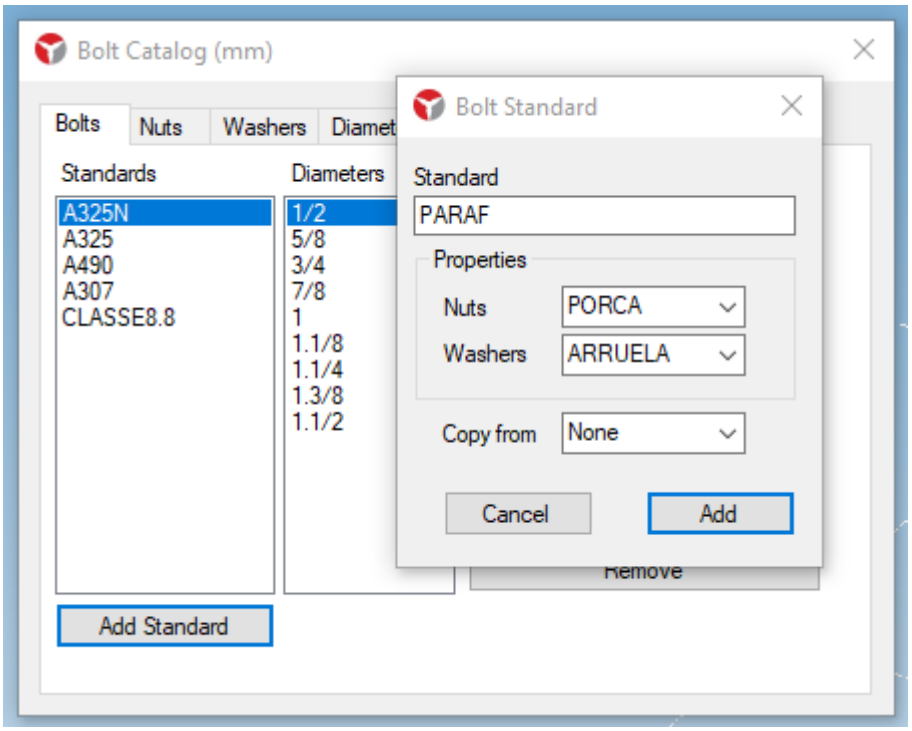
Sow registration (NUTS)

In the same way as we did with the washers, we need to inform the nut sizes for the new diameters.
I will create a new NUT pattern and add the 2 diameters:



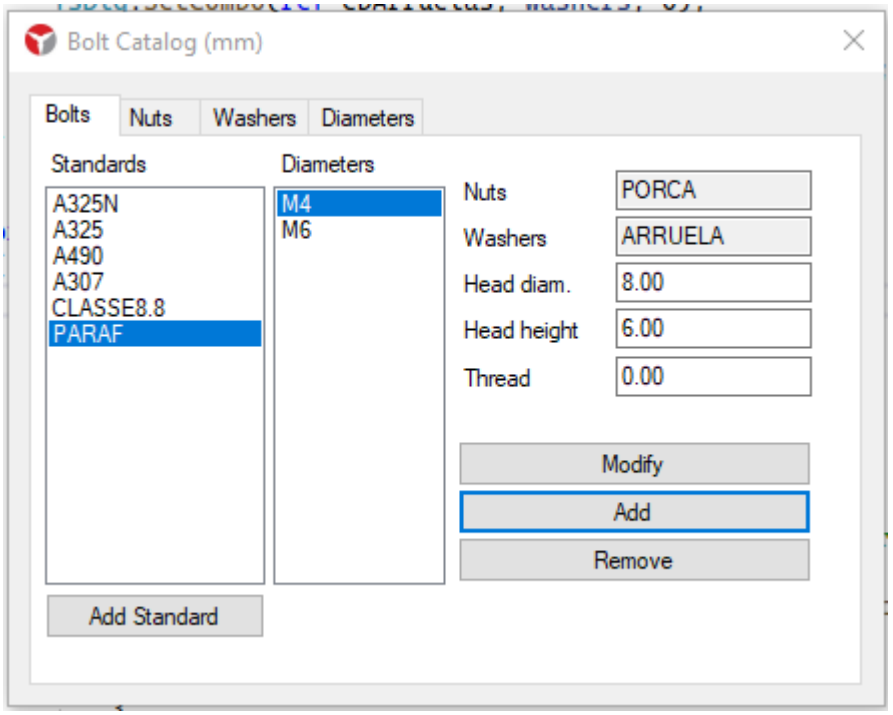
Screw registration

Let's create a new bolt pattern, which I'll call PARAF:



Note that we named the new standard PARAF, and informed the type of nut and washer we want to use. In the “copy from” field, we will leave it null and we will manually create the screws.

The last step is to add the diameters you need. Note that only the diameters that we registered in WASHER and NUT will be available.
Final result:



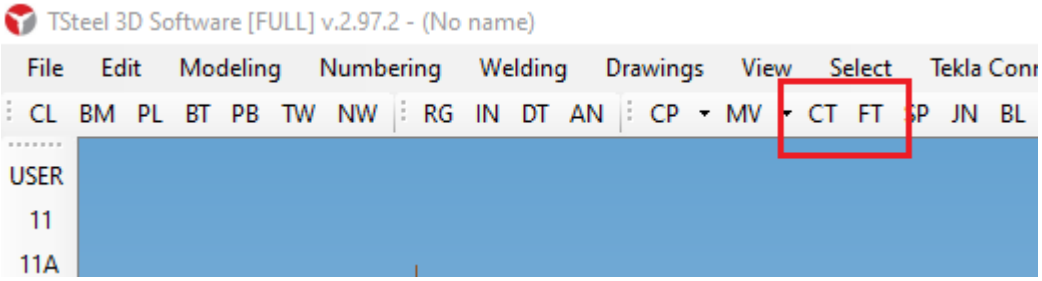
Cutouts and adjustments

Cut planes and fit planes

Cut planes can be used on plates, beams and columns and must cross the part. A piece cut by a plane is divided into two parts, one of which will be eliminated.

The commands are found in the toolbar:

- CT: Cut Plane
- FT: Fit Plane
-



Fit planes can only be used on beams and columns and operate in two different ways:

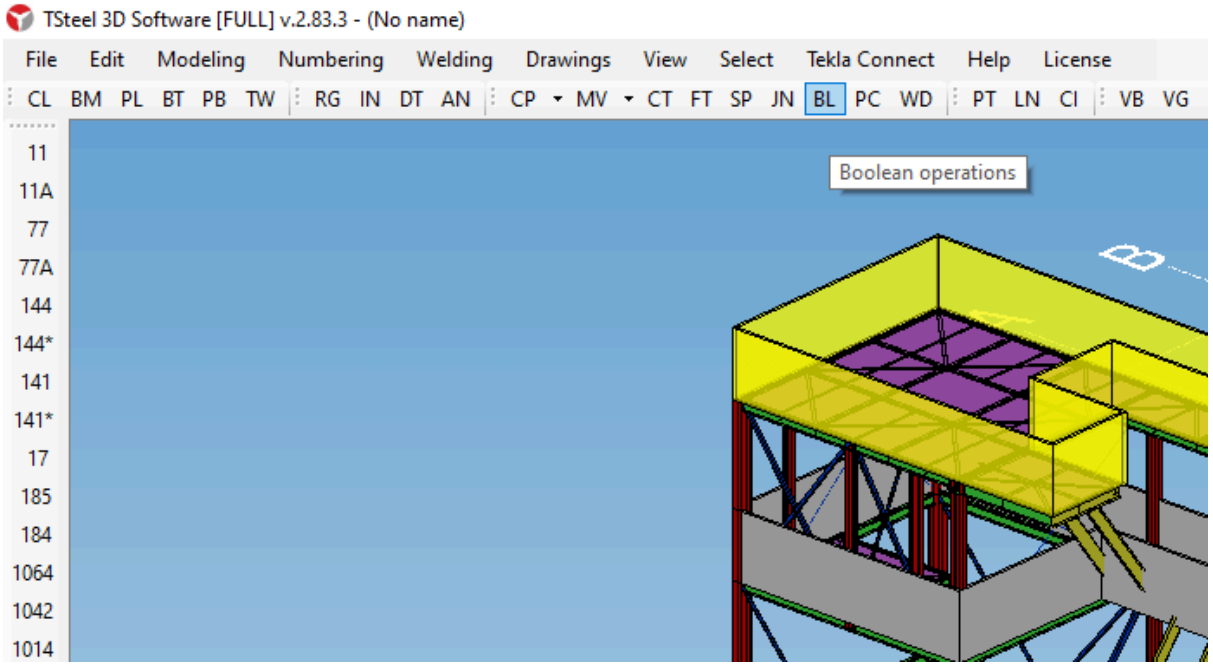
- When they cross the piece, they work like a cut plane, but automatically eliminate the smaller part of the piece;
- When they do not pass through the piece, it will be extended until it adjusts and touches the plane.

Both are included in the pieces in a similar way, being necessary to choose the piece to be edited and then the plan. In the 3D view, it is possible to select an existing plane. If you don't have any plane defined, you will need to use a 2D view and define the plane by two points. To view cutting or fitting planes, you can turn on the display of all model planes in theview configuration window, or you can turn on the option to show the cutting planes of the selected part in advanced visualization settings (ALT + double click on the screen).

See usage video

Holes and Cutouts (Boolean Operations)

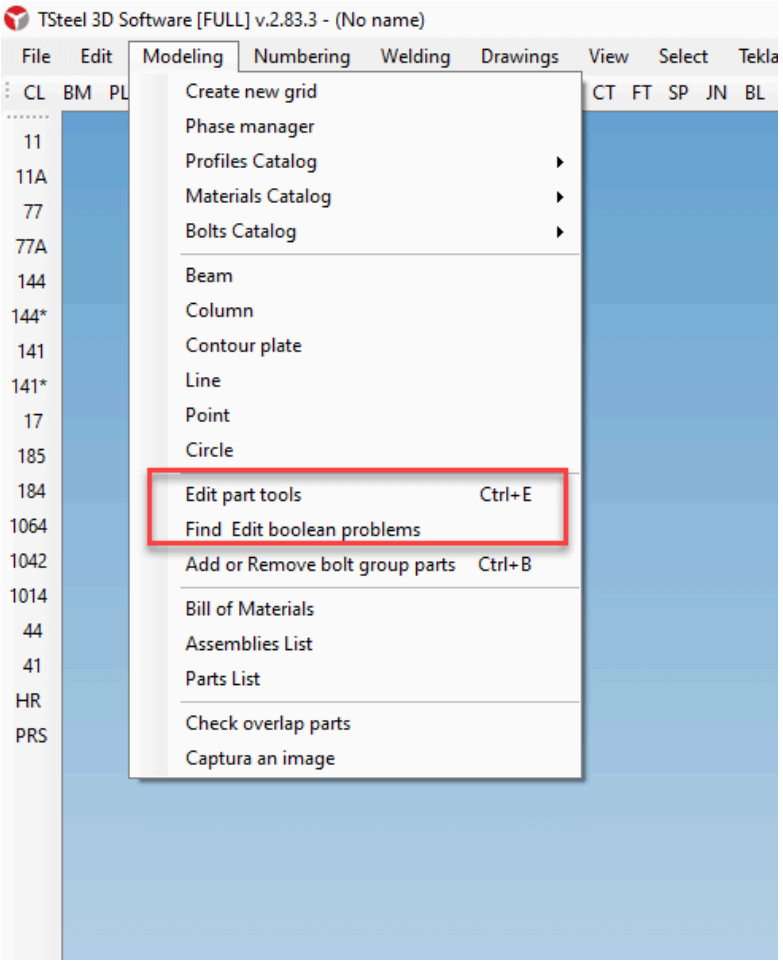
The holes and cuts in the parts are made using Boolean operations, which is nothing more than the subtraction of solids.
The solid used to make the cut or hole is called a Boolean part. The commands are in BL (Boolean operations) and PC (Polygonal cut), which is a faster and easier way to perform a Boolean operation. Let's see both ways working in the video.



[Video of holes and cutouts in parts]

Advanced part editing

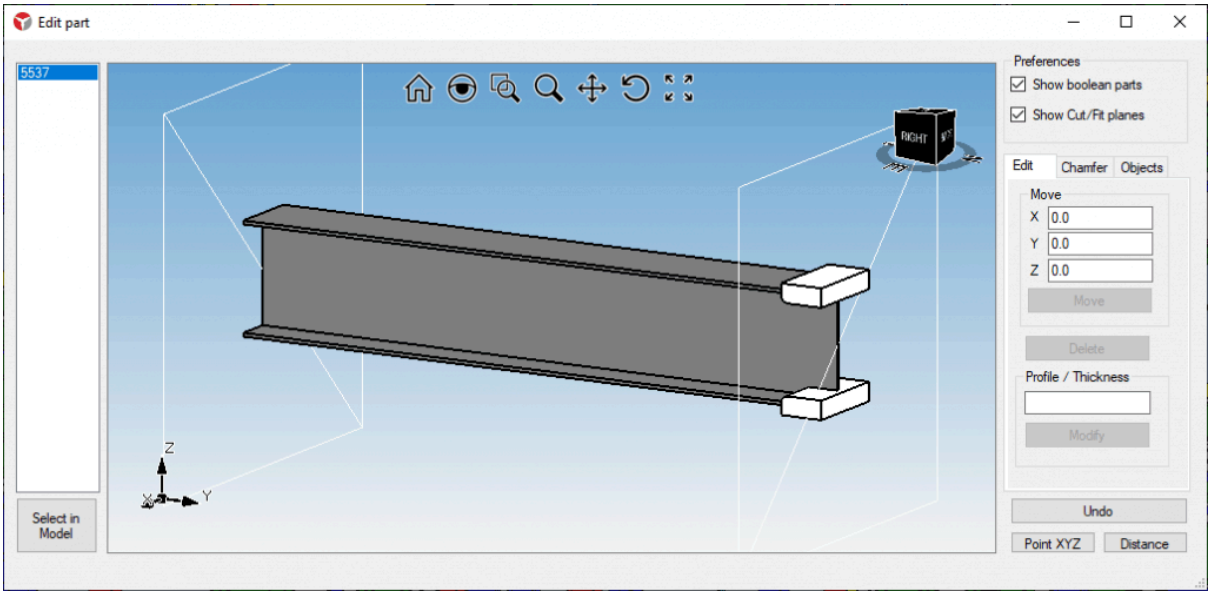
TSteel 3D offers a special window for editing parts, the “edit part tools“, activated by the menu or the CTRL+E key.



The window allows:

- Edit plans (Cut planes, Fit planes)
- To checkboolean parts with error (shown in black)
- To editboolean parts (cuts)
- Add chamfers to profile edges

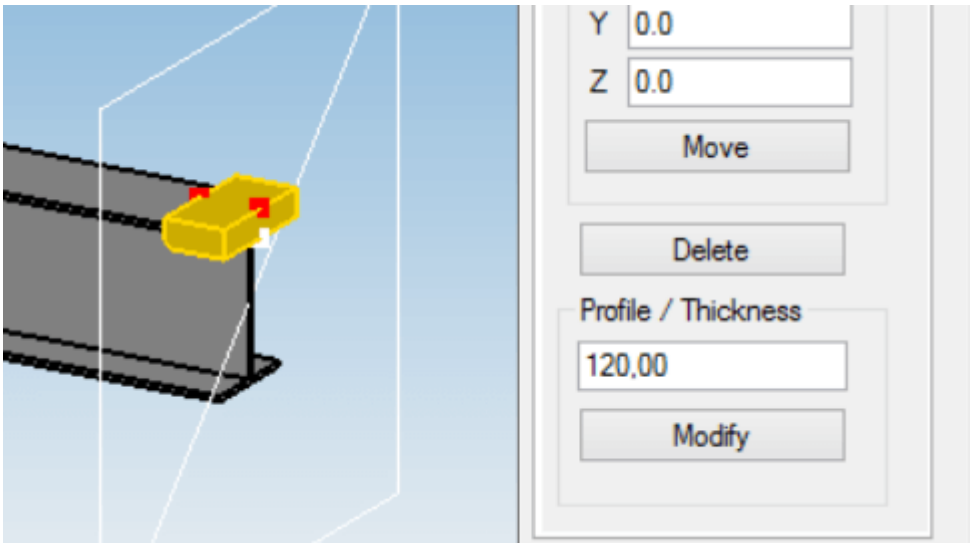
The “Find & Edit boolean problems” menu option opens the editing window with all parts that have a cutting problem. This is useful after importing larger Tekla models.



Note above the window with a beam (ID = 5537) that has two cutting planes at the ends and two cutouts (Boolean pieces).
The window allows you to edit any plane or cut that exists in the part, in addition to helping you create new cuts to make chamfers on the edges of the profile.
Edits made to the profile are immediately updated in the template.

Changing the profile or thickness of a Boolean part

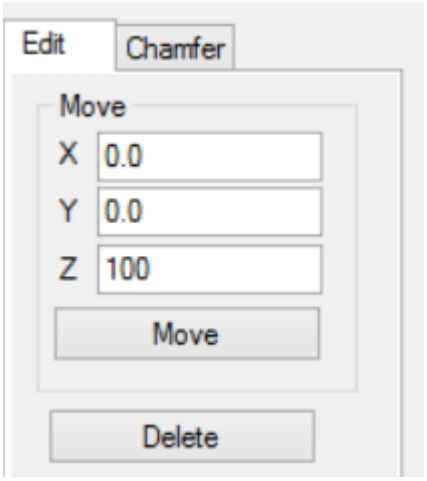
If you click on any Boolean part, the window allows you to change the profile/thickness:



Just change the profile (or thickness value) and click “Modify”

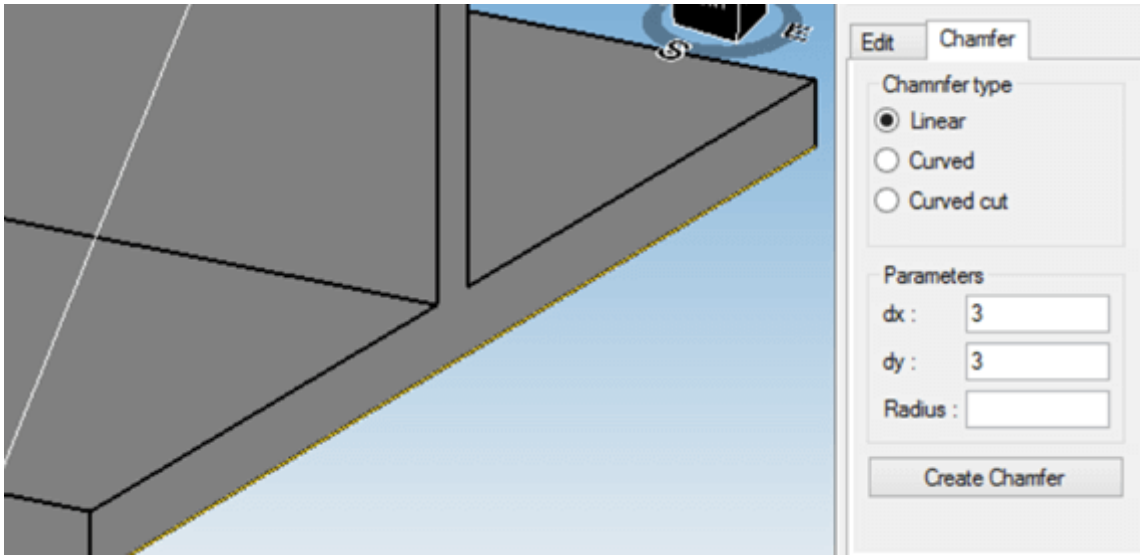
Move or delete a Boleyn tile

After selecting a Boolean part, just press DEL to delete it. If you want to move the piece, enter the X,Y,Z coordinates and click “MOVE”.

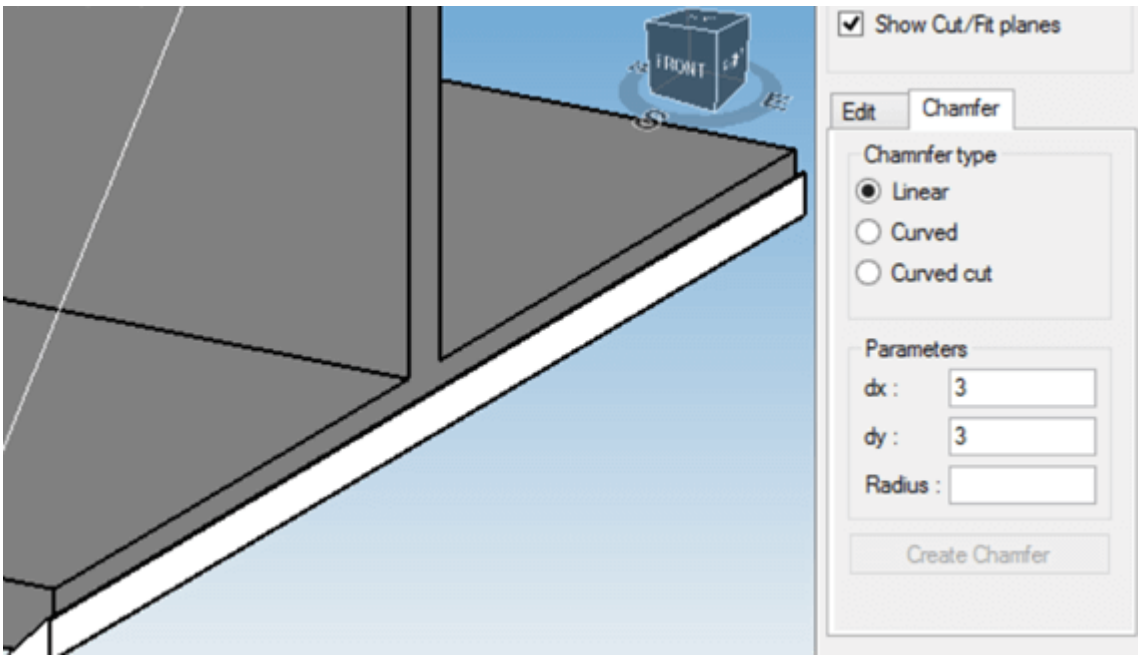


Add a chamfer to an edge

To add the chamfer, click on the edge to select it and enter the chamfer parameters on the “Chamfer” page.



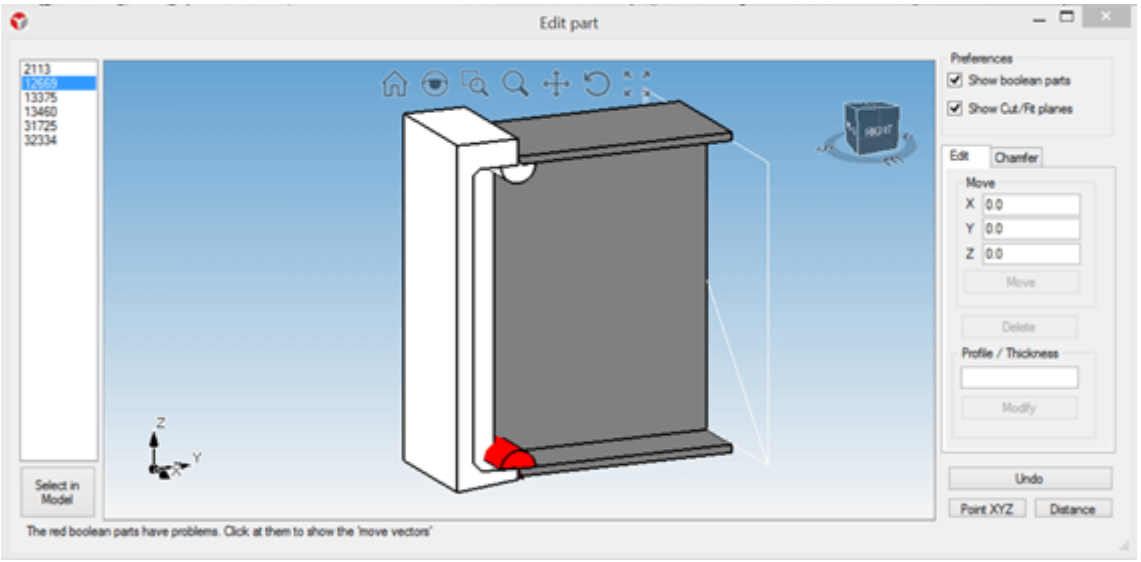
After entering the data (type of chamfer and dimensions), click on “Create Chamfer”. An additional Boolean part will be created that will create the chamfer.



Boolean parts with problems

see the[post about Boolean parts with problems] to understand what they are and what causes these errors.
Here we will show how to get these pieces right to allow the program to perform Boolean operations.

The Boolean part where the error occurred is shown in red.



If you click on the Boolean part in red, the program may suggest displacement vectors for the part. These are very small displacements (in the order of 0.1mm) that seek to avoid coincidences of edges and faces between the two solids. These displacements solve the vast majority of problems.

Black pieces (solid subtraction errors)

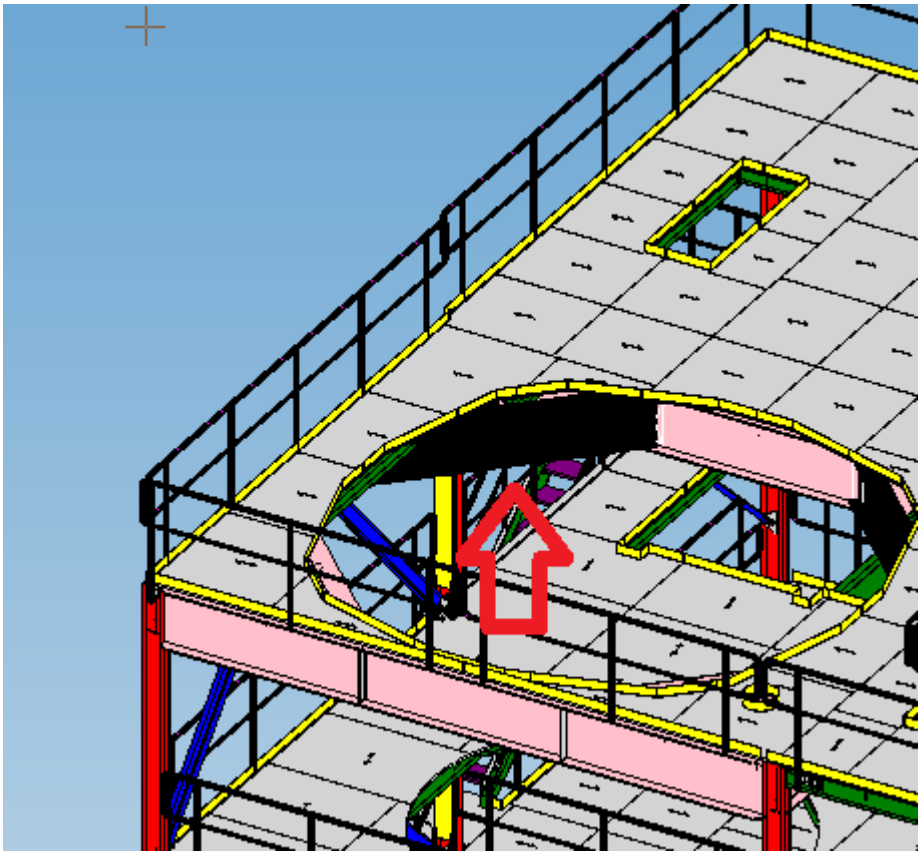
As we saw, the holes and cutouts in the pieces are madethrough “Boolean” operations, which are a subtraction of solids. Sometimes, subtraction is not possible and the final shape of the piece cannot be determined. To signal this, it shows the part in black (regardless of its class), so you can identify that there is a problem to be corrected.

Most of the time, a simple adjustment to the “Boolean” part solves the problem. There are cases where the combination of overlapping cuts and cutting planes makes the situation more complex.

Overlapping cutouts is poor modeling practice and should be avoided. Try to create a clean and simple model, especially because we have to think about interoperability. Complex pieces may have problems when interpreted by other programs.

The “Edit part tool” (seen previously) It is a window that shows all the cutouts of the parts and identifies the cutouts with problems in red.

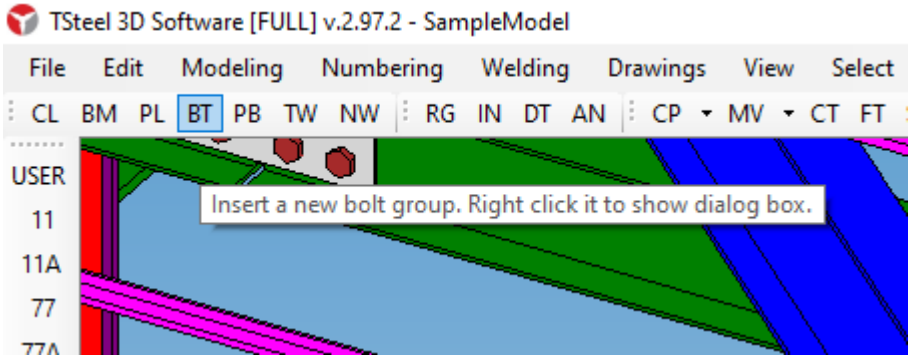
Example of a black part in the model.



Screws

Inserting screws

The command to insert screws:

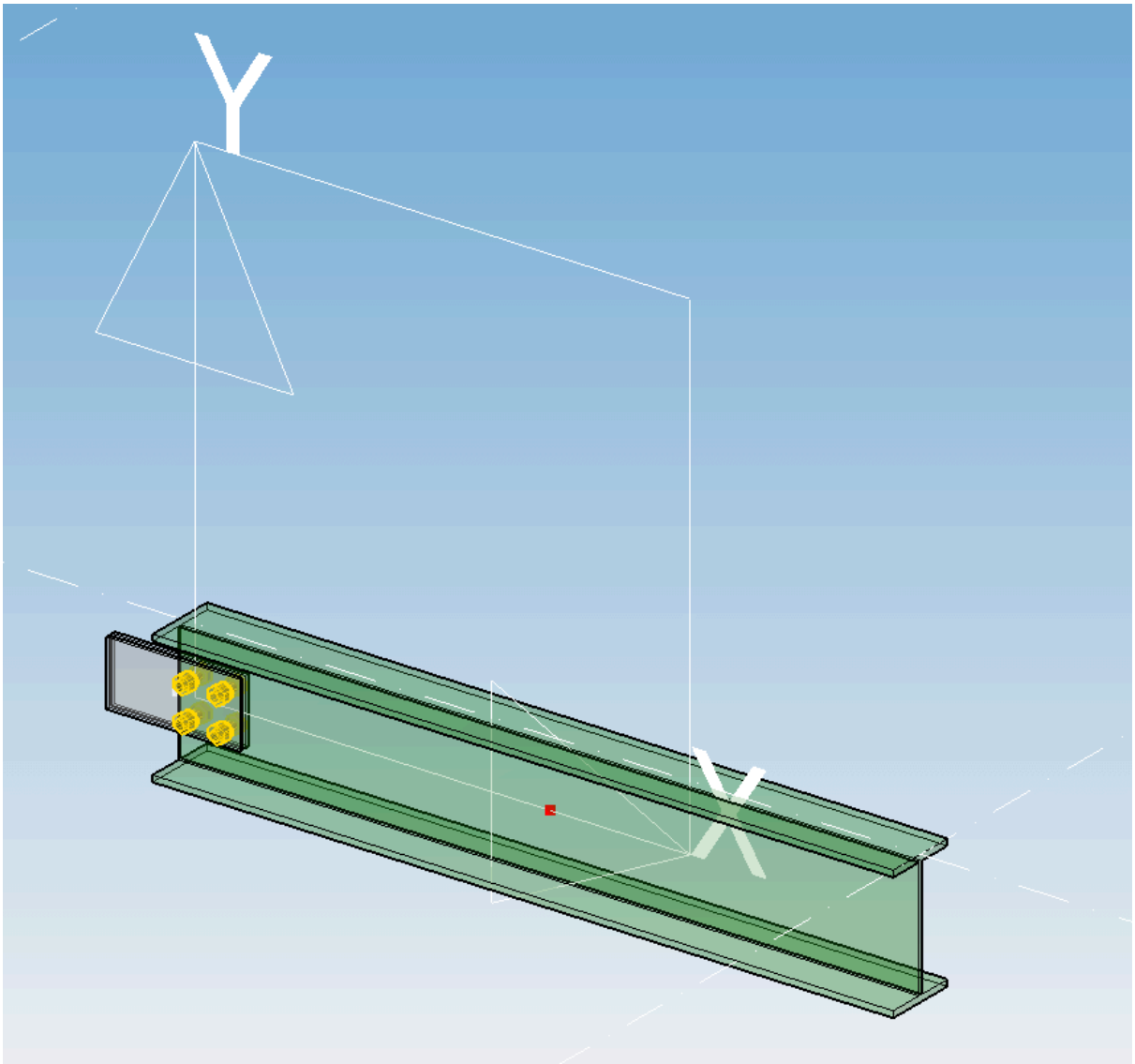


When you call the command, the parts that will be screwed must already be selected.

To insert screws, we need to define:

- Parts to be screwed (at least 1)
- X axis of the screw plane defining two points in the model
- Y-axis of the plane of the screws through rotation
- Spacing between screws

To better understand the definition of the axes, and therefore the screw plane, see the figure below with the plane indication:

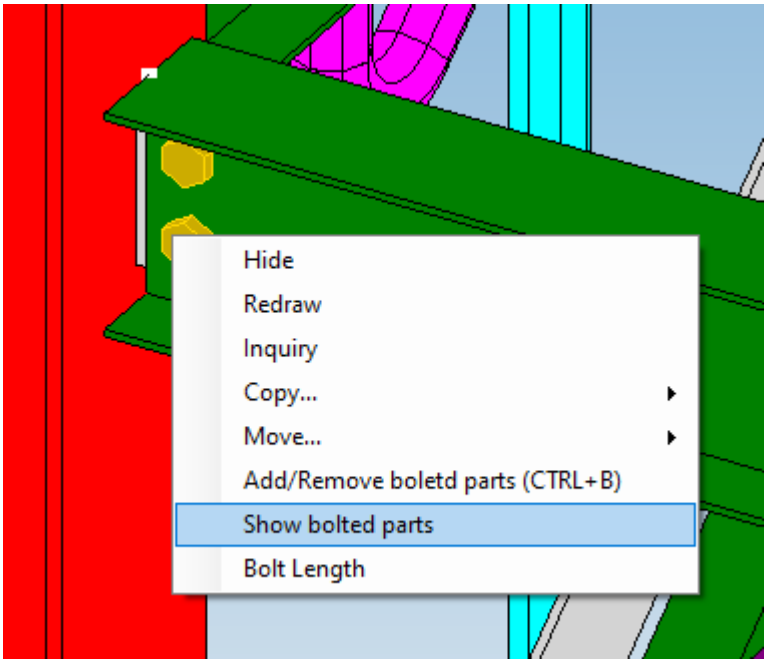


Note the screw creation points (initial is white and the end is red) that define the X axis. The Y axis will be defined by the rotation (TOP, FRONT, BACK, BELOW) which is similar to the rotation of beams and columns.

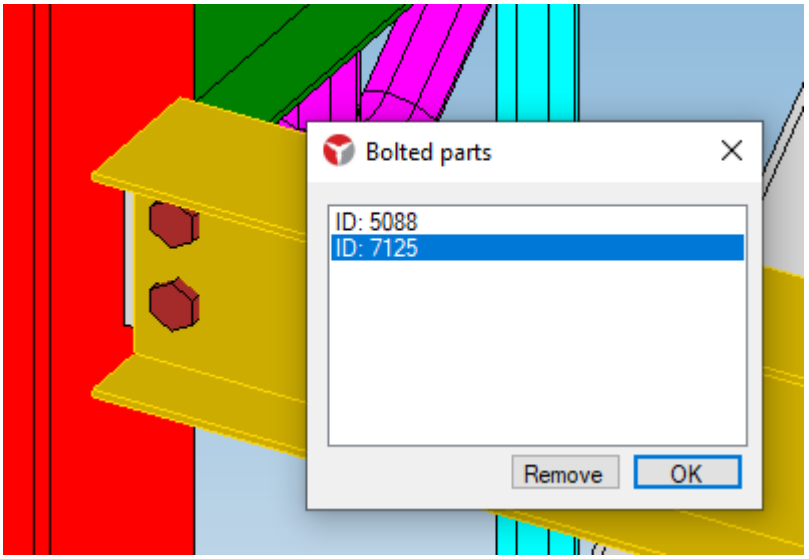
Video on how to create screws

Bolted parts

To check which parts are screwed together, right-click the screw:



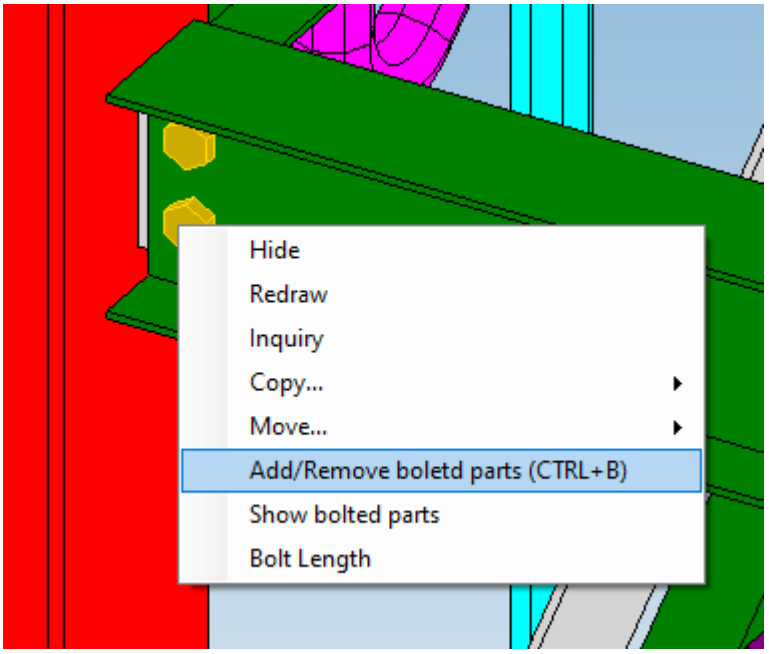
The screen with the screwed parts appears:



Note that you click on the form part and it is selected in the model.

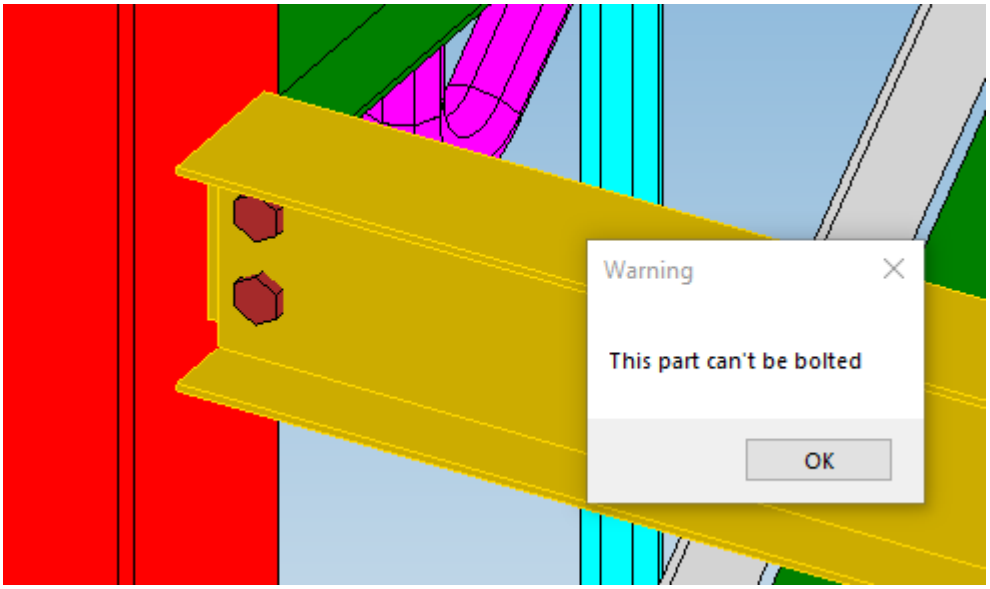
Include and exclude bolted parts

Right click on the screw and select:



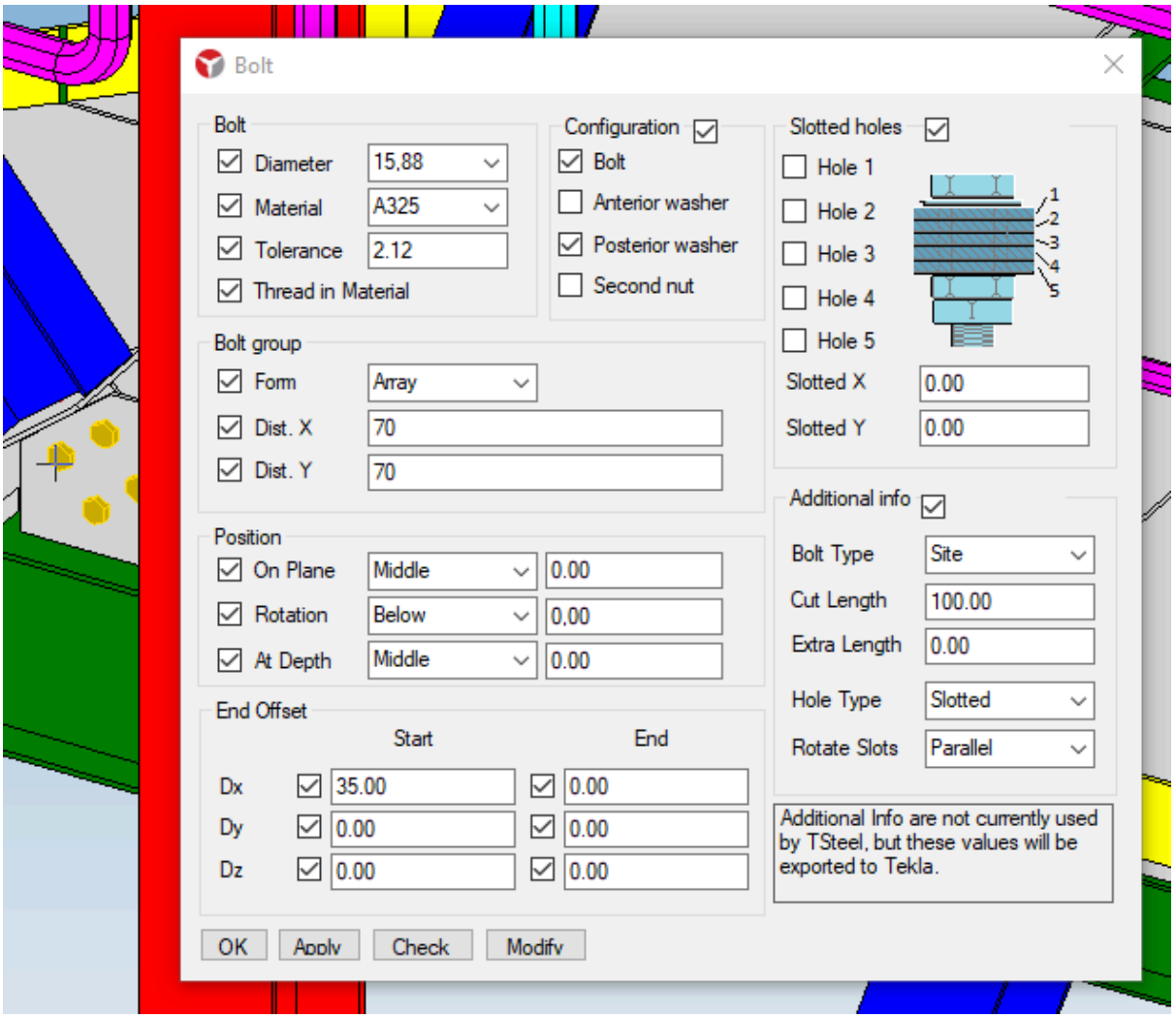
All parts included in the screw are selected. From now on you can click and add new parts, or click on an already screwed part and exclude it from the screw.

And if you select a part that TSteel cannot include on the bolt:



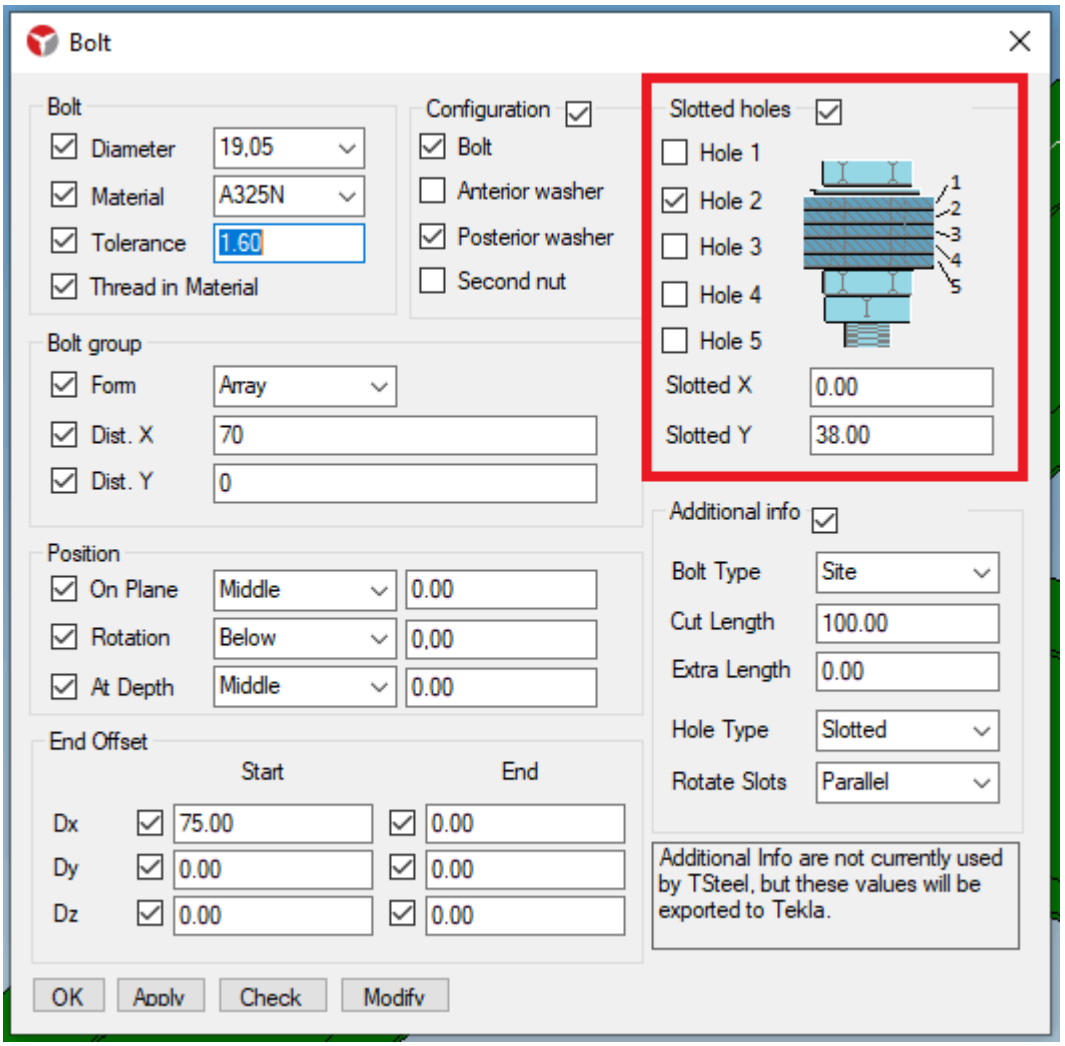
Possible configurations

Double clicking on a set of screws opens the window with its settings:



Oval (oblong) holes

To add an oval hole to a part, use the screw configuration.

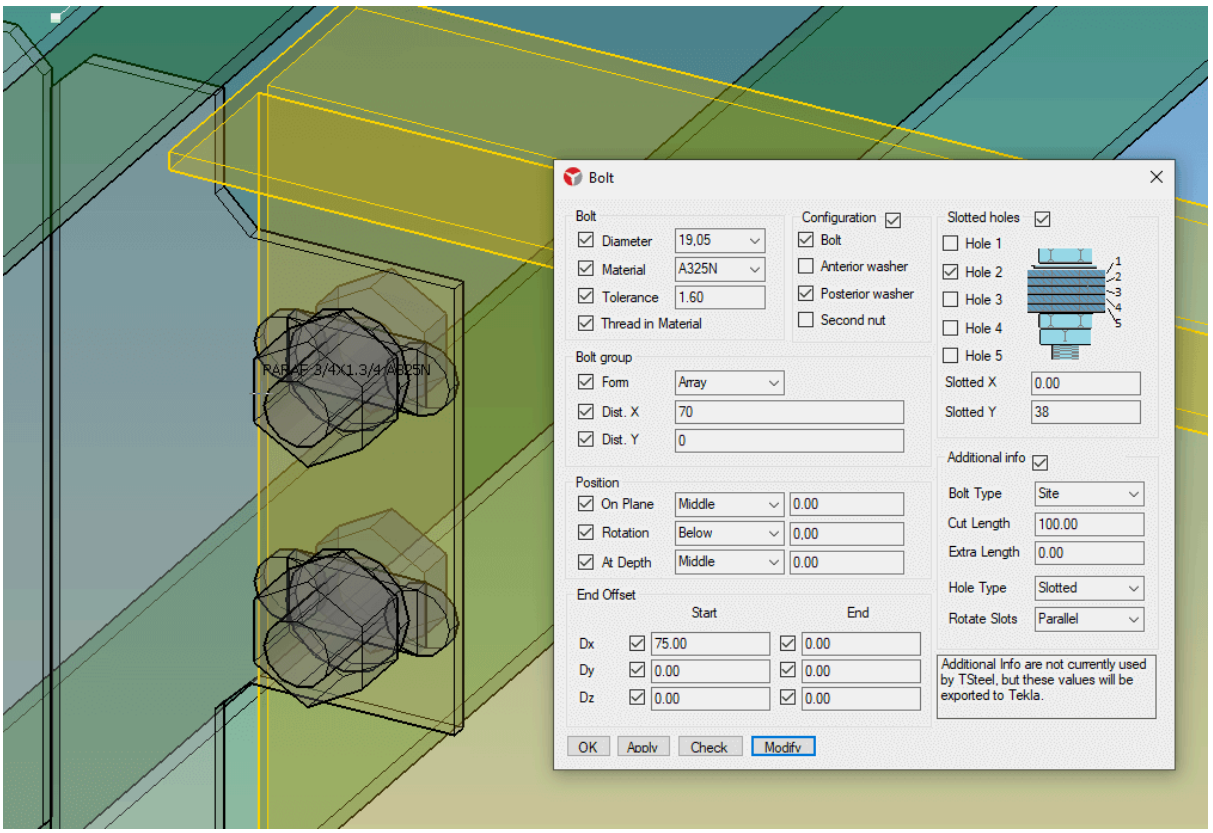


You configure which part will have the oval hole, note that the parts are numbered in the screw head-nut direction.

If none of the 5 possible parts are marked to receive the oval hole, the hole dimensions (Slotted X, Slotted Y) will be ignored.

You can make the oval hole in the X or Y direction, considering the hole creation axes. If you mark both dimensions, it will appear in the model, but not in the DXF or machine programming files (CAM, NC, etc…)

You will be able to see in the model where the oval hole is being considered.



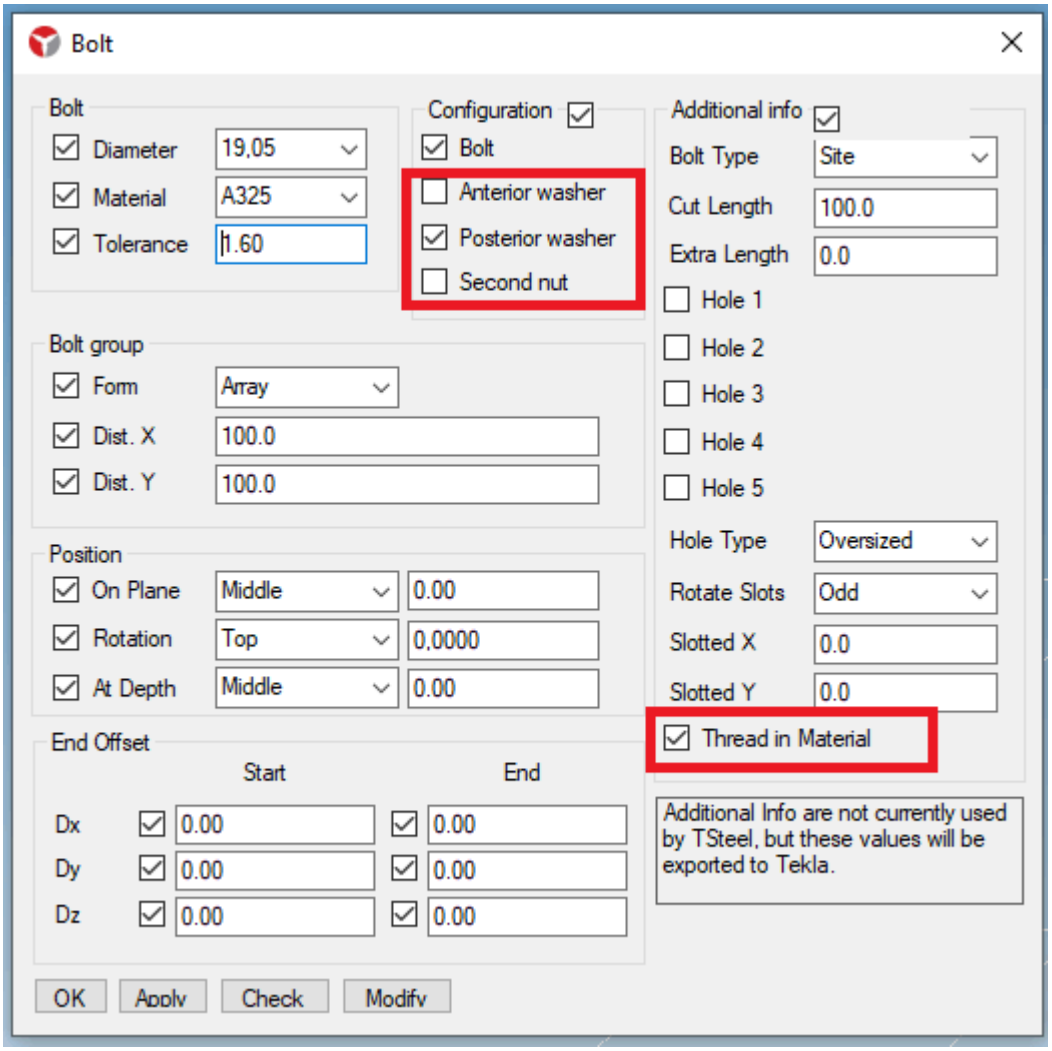
How screw lengths are calculated

See how TSteel 3D calculates screw lengths, following user settings and practical requirements.

What are the Screw configurations

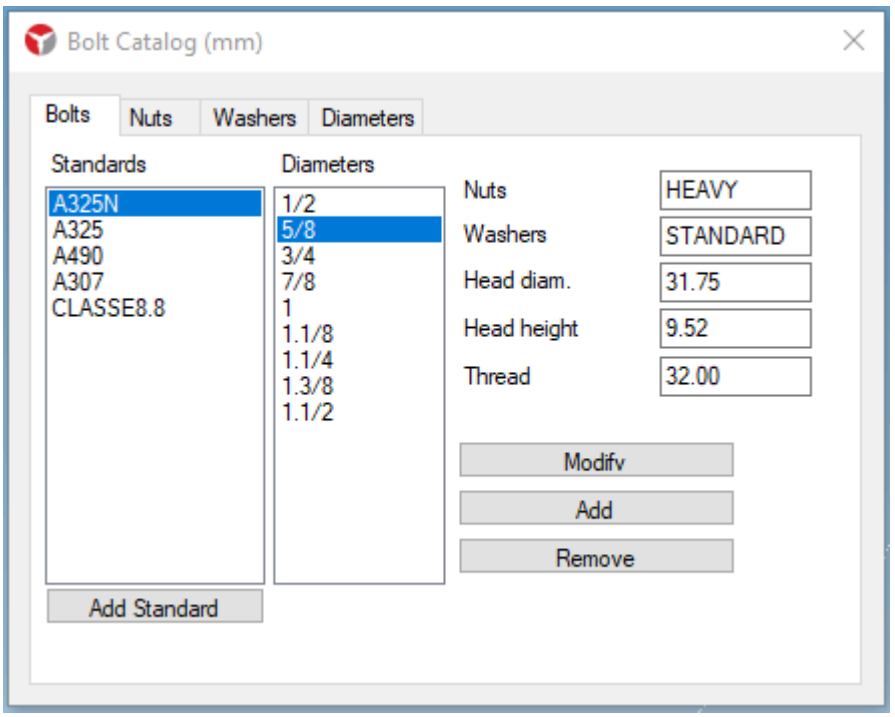
The data needed to calculate the length of a screw are:

1. Material and diameter (define the thread length)
2. If there is a washer on the screw head (previous washer)
3. If there is a nut washer (back washer). Here, although it is a user option, TSteel 3D will add the necessary number of washers to tighten it.
4. Whether to use two nuts
5. Whether there may be threads in the cutting plane



Where are thread measurements, washer thicknesses and nut sizes defined?

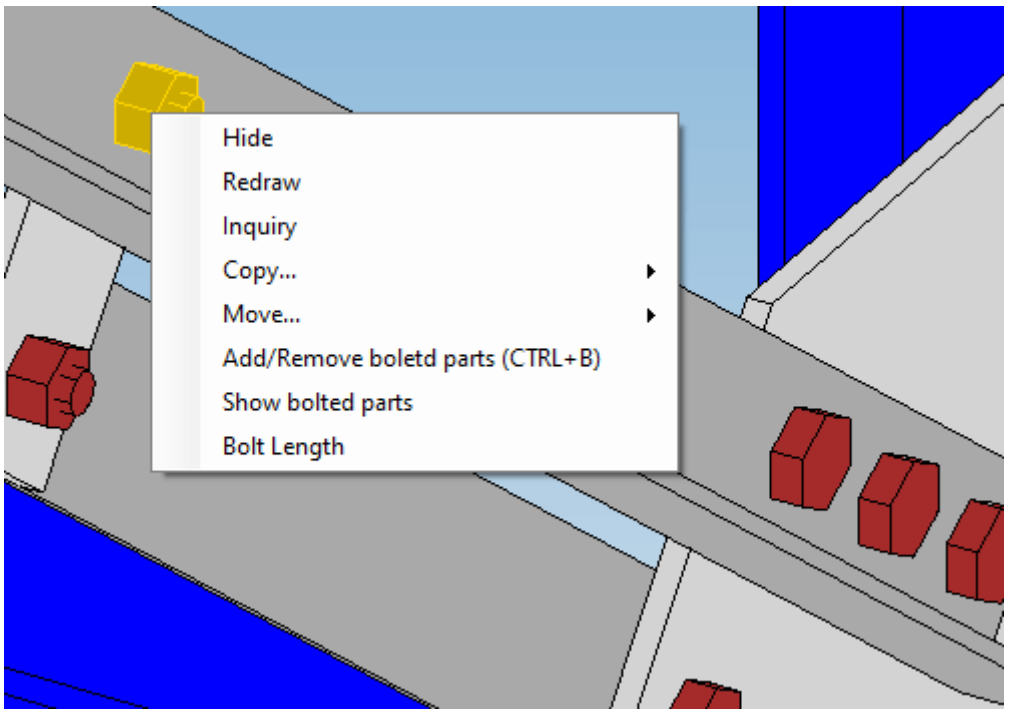
All this data, which is essential for correctly calculating the screw length, is defined in the screw catalogue.
The thread length must be defined for each material and each screw diameter. See below for the definition that a 5/8″ A325N screw has a thread length of 32mm.



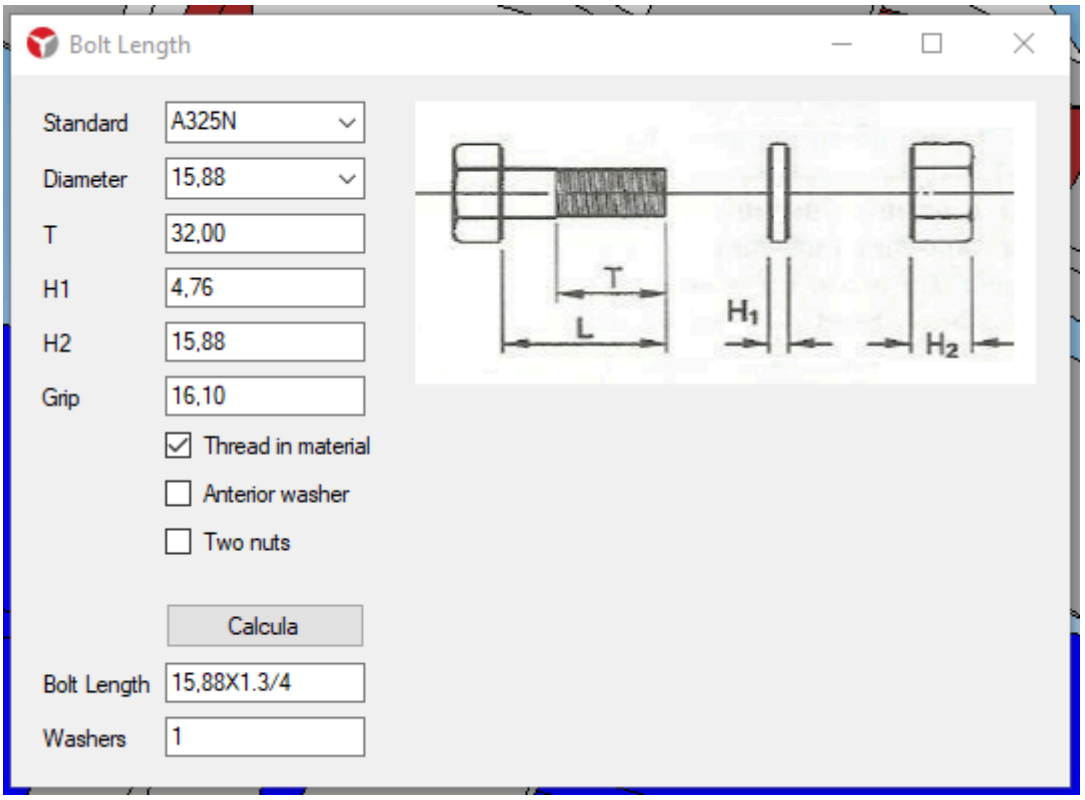
The sizes of the nuts and thickness of the washers are also defined in the screw catalogue.

How do I know the data used to calculate a screw?

Right click on the bolt and choose the “Bolt Length” option:

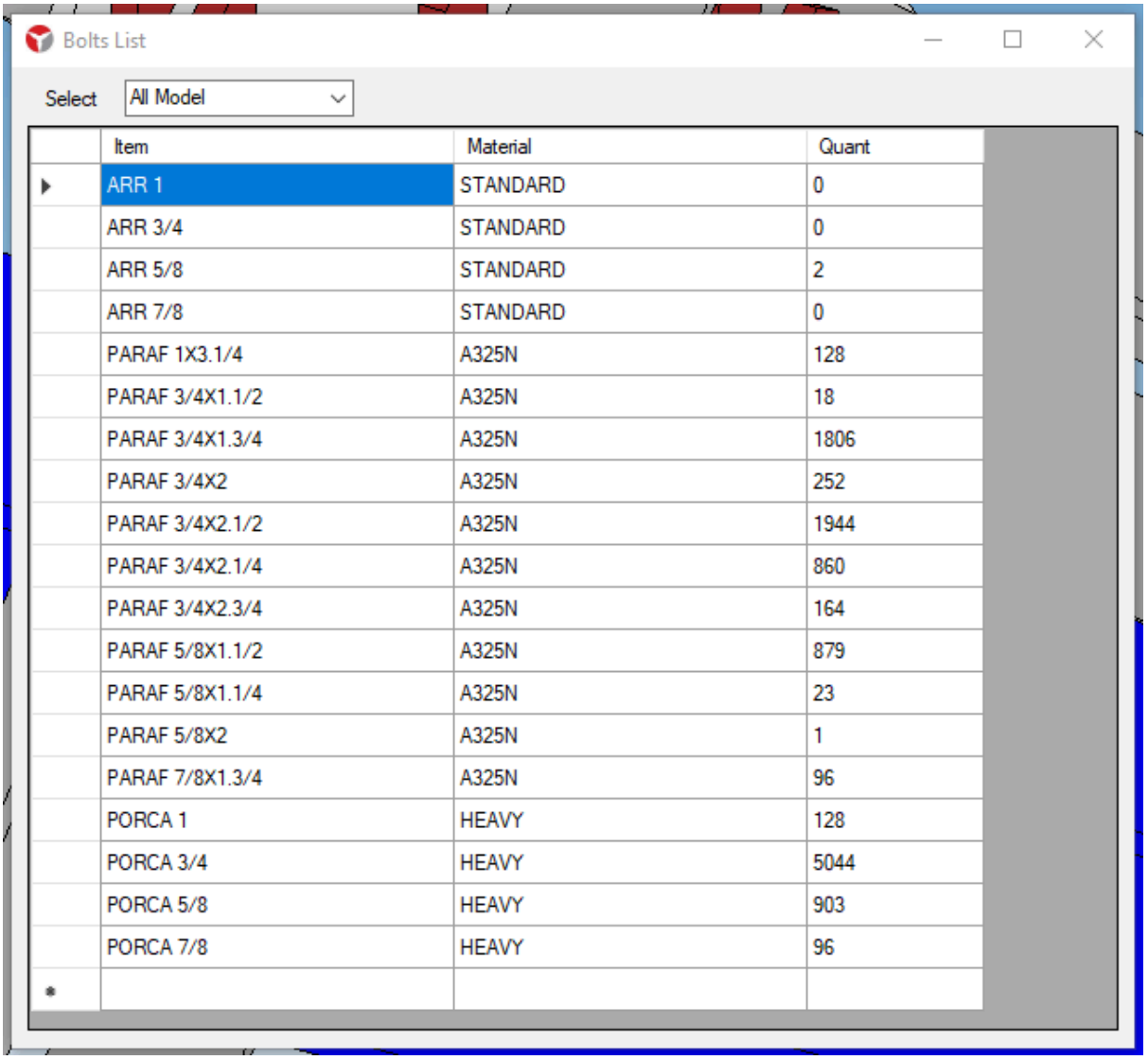


TSteel 3D will open the window below, where you will see your screw data. In this same window, it is possible to modify the data and recalculate the length. Modifying parameters in this window does not modify the screw, it is only used to check the calculation process.



How to get the screw list.

Vá no menu Modeling>Bolts List.
You will see the list as in the example below:



	Item	Material	Quant
▶	ARR 1	STANDARD	0
	ARR 3/4	STANDARD	0
	ARR 5/8	STANDARD	2
	ARR 7/8	STANDARD	0
	PARAF 1X3.1/4	A325N	128
	PARAF 3/4X1.1/2	A325N	18
	PARAF 3/4X1.3/4	A325N	1806
	PARAF 3/4X2	A325N	252
	PARAF 3/4X2.1/2	A325N	1944
	PARAF 3/4X2.1/4	A325N	860
	PARAF 3/4X2.3/4	A325N	164
	PARAF 5/8X1.1/2	A325N	879
	PARAF 5/8X1.1/4	A325N	23
	PARAF 5/8X2	A325N	1
	PARAF 7/8X1.3/4	A325N	96
	PORCA 1	HEAVY	128
	PORCA 3/4	HEAVY	5044
	PORCA 5/8	HEAVY	903
	PORCA 7/8	HEAVY	96
*			

How screw length is calculated

The calculation sequence is as follows:

1. Calculates the theoretical minimum screw length
2. Search for the available length immediately greater than the desired length
3. Check if you need to add washers to ensure tightness
4. Check that, by adding the washers, you have the extra length needed. If there is none left, adopt a longer length and go back to step 3

Minimum required length

The minimum length is the sum of:

- Previous washer (if present)
- Grip (thickness to be gripped)
- Rear washers (quantity calculated in step 3)
- Nut
- Second nut (if any)
- Extra length at screw end

In the case of screws without threads in the cutting plane, we have to check a second length. The minimum length will be the greater of the two calculated lengths.

- Previous washer (if present)
- Grip
- Thread length – 3.0mm. This ensures that the cutting line does not contain threads.

The extra length is calculated depending on the screw diameter. The criteria is as follows:

Diâmetro	Comprimento extra (mm)
>33.0	10.0
>27.0	8.0
>22.0	7.0
>16.0	6.0
>12.0	5.0
>7.0	4.0
>4.0	2.5
<4.0	1.5

When the program cannot calculate the screw length

When this is not possible or when your model was created in a version of TSteel 3D prior to 2.87. This is because, before we included the calculation of screw lengths, there was no thread length information in the catalogue. You will notice, in the catalogue, that the thread lengths are equal to zero. Therefore it is not possible to calculate the screw.
You can do 2 things:

- Edit the screw catalog, including the thread size for each gauge. It's something simple and quick.

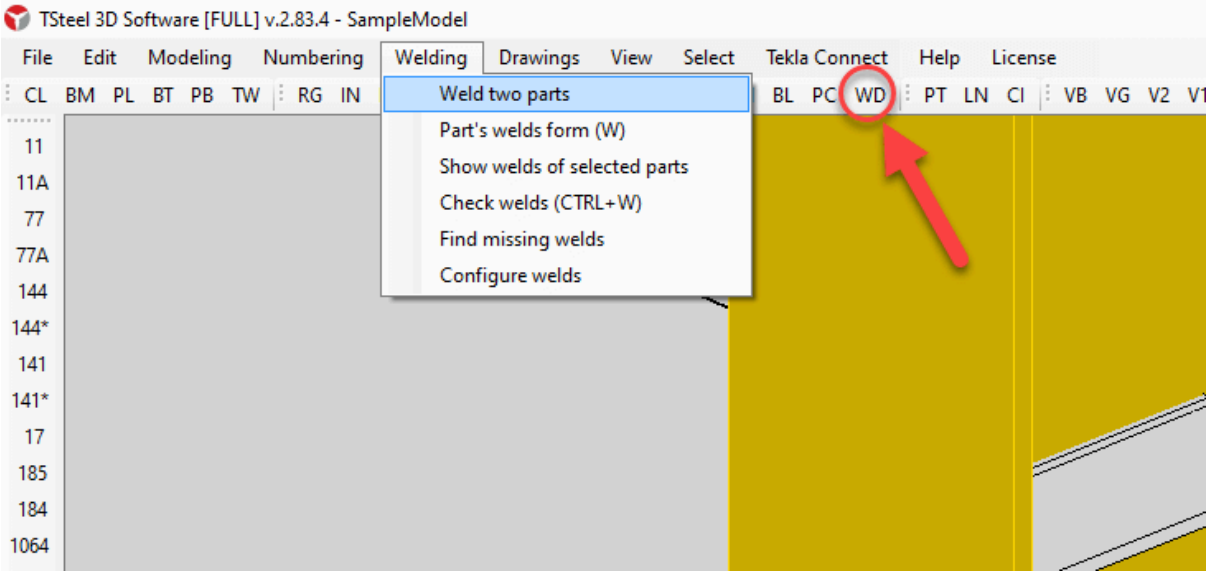
- You can delete the screw catalog “TabParafusos.tab” in the model directory. When you enter your model, TSteel 3D will issue a warning that it did not find the table and will create the default table (which already contains the thread lengths).

Welds

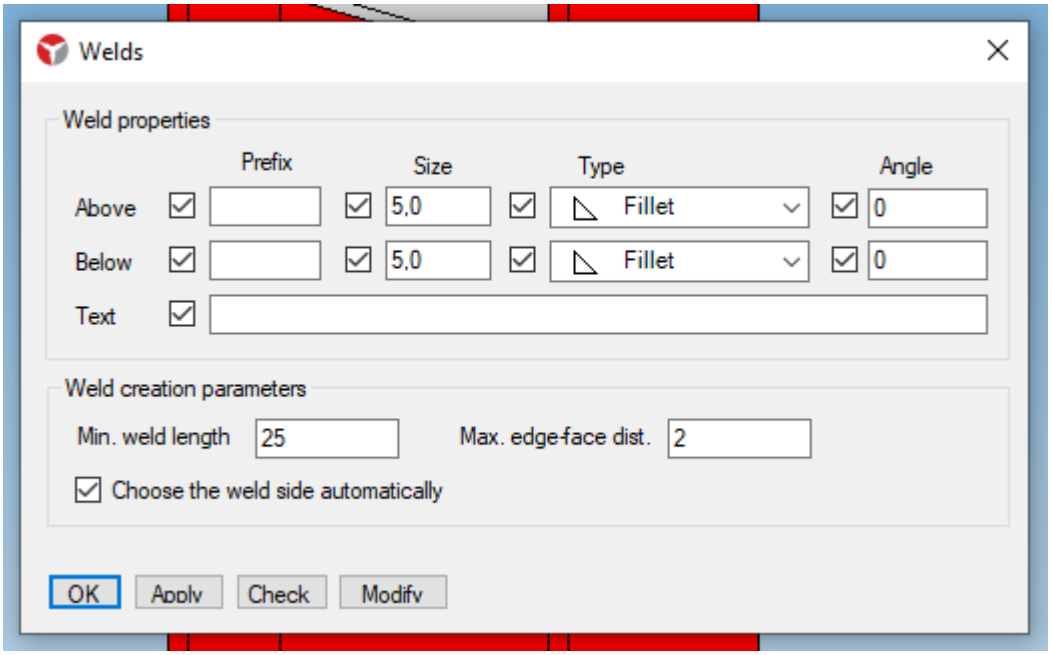
TSteel 3D allows you to create different types of welds between parts and manage existing welds easily.
Weld lines can be visualized in the model, in order to identify and check the data.
TSteel 3D only allows you to create a weld if it finds possible weld lines. In other words, it is not possible to weld parts that do not connect.

What is the welding process like?

To weld two parts, use the WD (WELD) command on the toolbar or the weld menu:



Right-click the WD command in the toolbar. You will see the weld configuration window:



The above parameters will be used for each new weld you apply. Change the parameters you need and click APPLY for the new weld configuration.

Select the main part first and then the secondary one. How important is this? TSteel needs to know which should be the main part of the assembly formed by the welded parts.

Watch a video on how to weld

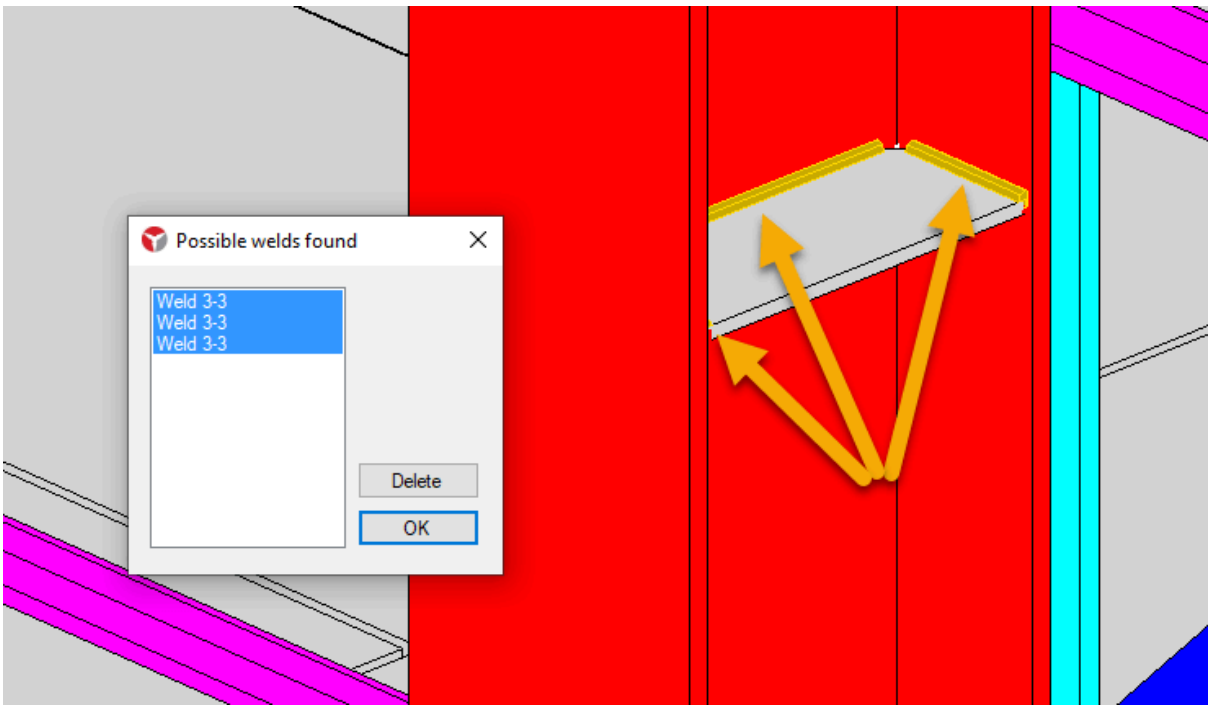
If TSteel 3D does not find weld lines between parts?

TSteel 3D only allows you to create welds when you find contact lines between the parts. When it receives a welding command, the program searches for possible weld lines. If none are found, you will receive a warning and the weld will not be created.

If there are already welds, TSteel will warn you that there is no way to create welds. Try creating the same weld 2 times and see what happens.

How to check the weld lines found by TSteel 3D?

When the program finds one or more possible weld lines, it displays a dialog box with the found lines. In this dialog box you can accept or reject weld lines.

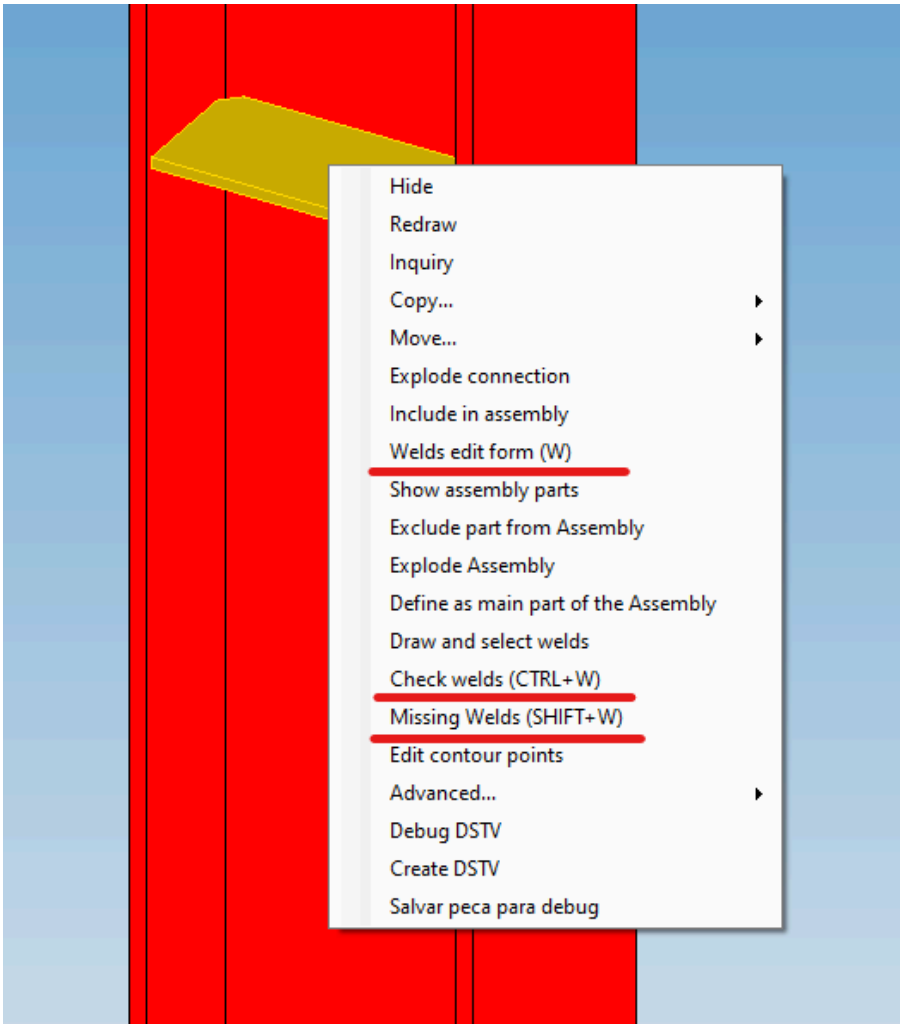


Graphical representation of the Weld

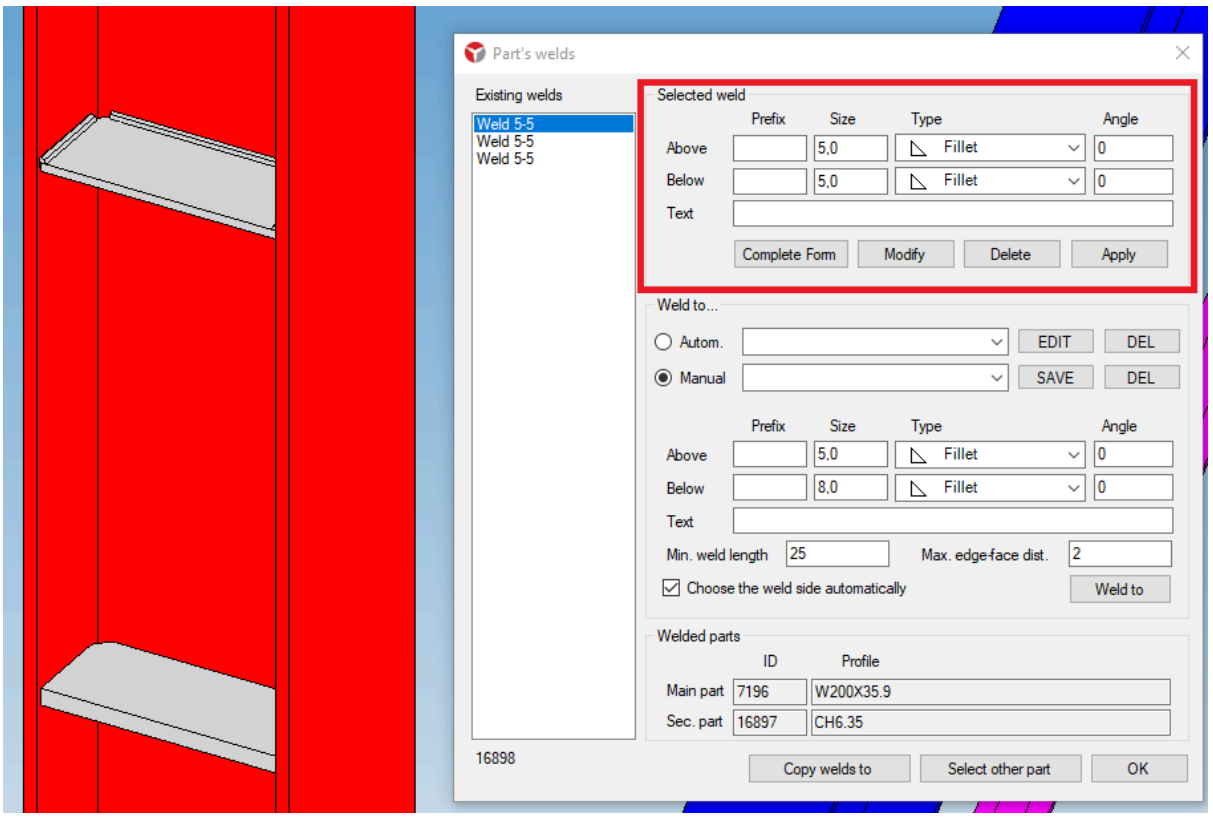
TSteel can graph the weld to show where the weld line is considered.**The representation does not reflect the type of weld (fillet, notch, etc…), only the location of the weld.**

Edit welds

To edit welds, select a part and use the Part's Welds Form (shortcut 'W'). Alternatively, you can use the right-click menu on the part. The editing window is as follows:



The weld editing window is as follows:

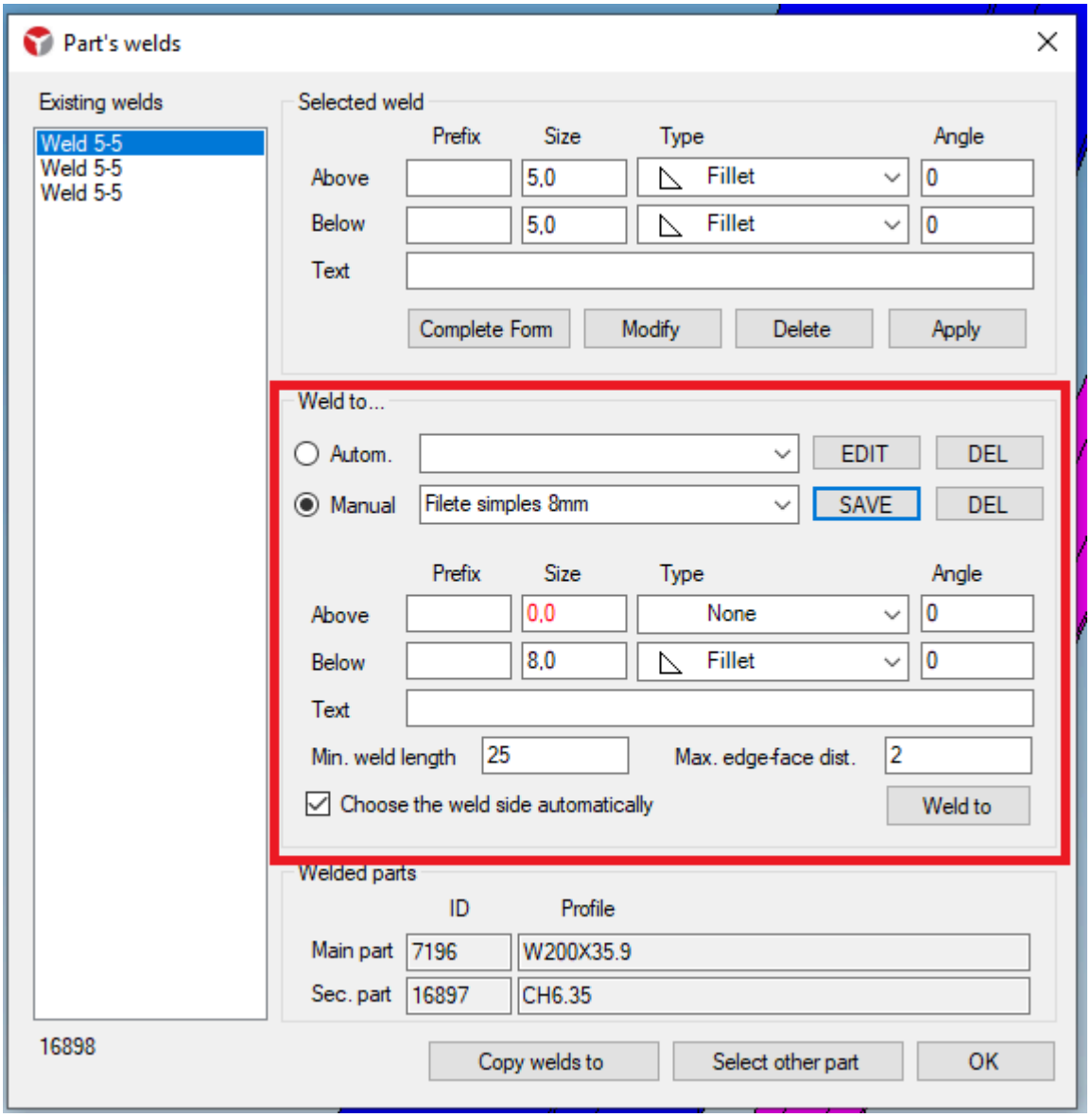


Select the weld you want to edit on the left. You can select more than one weld to edit simultaneously. Once the welds to be edited have been selected, in the marked area, change whatever is necessary: type of weld, dimension, angle when applicable and text.

Watch the video about weld editing

Create standard weld typologies and use them with ease

In the configuration area, shown below:



Fill in the name of the welding pattern, in the example above, I used “Single Fillet 8mm”, configure the welding. Note that I configured a simple 8mm fillet. And click**SAVE.** From now on, in the manual welding drop-down menu, you will have the 8mm single fillet weld available.

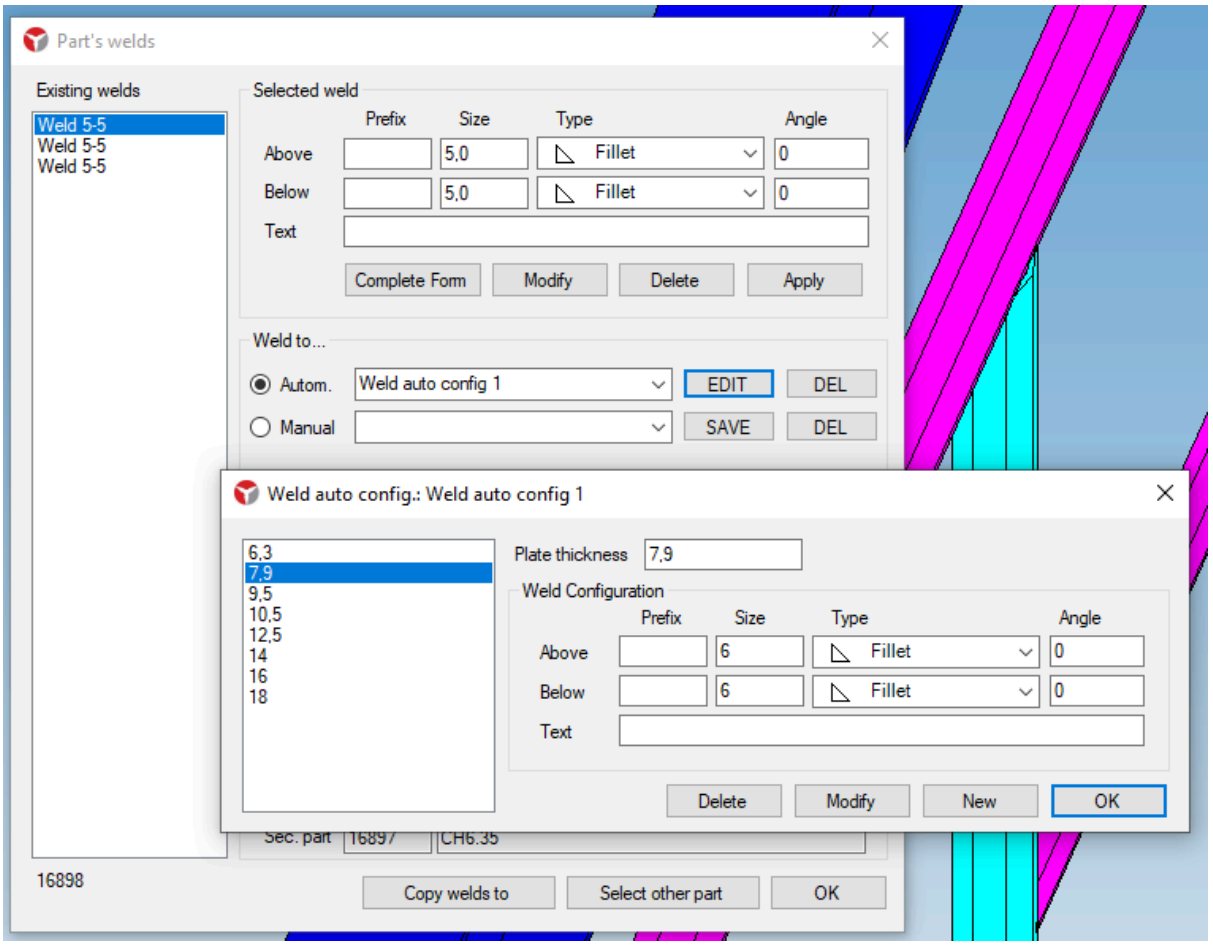
click in**Weld To**, select the part to be welded and see that a new weld will be created with this configuration.

You can create as many patterned welds as you want. Don't want this manual pattern anymore? Click on DEL and eliminate it from the options.

Create automatic welds depending on the sheet thickness

In the welds window, select and choose an automatic welding type:

Watch video about automatic welding



Note that for each thickness, shown on the left, there is a weld configuration. Using this automatic welding, TSteel will look for the thickness of the sheet being welded and create the weld according to this standard.

Finding missing welds

In the option “Find missing wels” (SHIFT-W), available in the welds menu or in the right mouse button menu, you can ask TSteel for locations where welds may be missing.

Watch the video on how to identify missing welds.

Assemblies

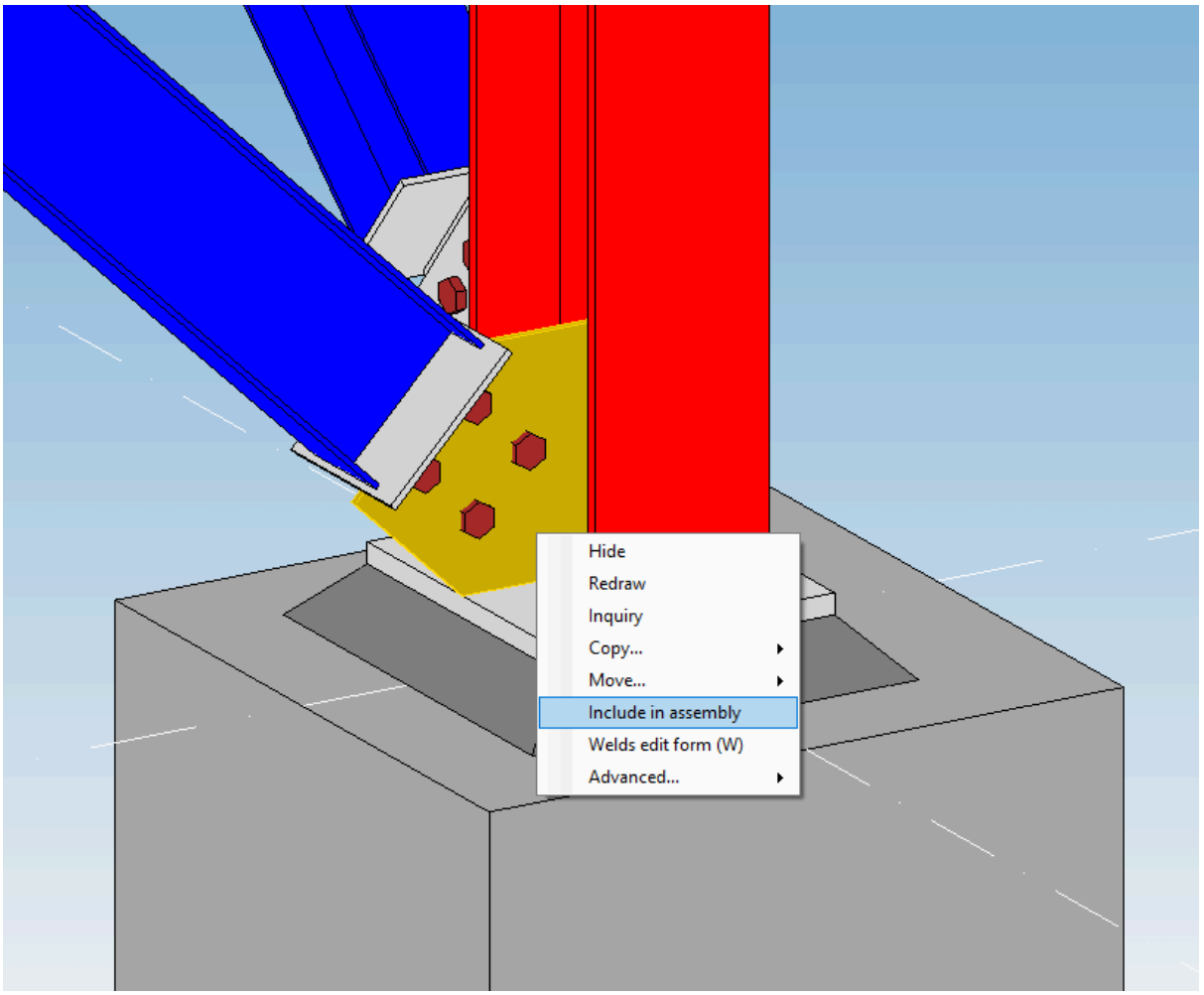
The sets (assemblies) are several parts welded together, forming a larger part that will be used for assembly. The final structure is the result of the assembly of all its assemblies.

How to add parts to a set

There are two ways to add parts:

Solderless inclusion

Right-click on a part to be included in the assembly and select the ‘Include in Assembly’ option:

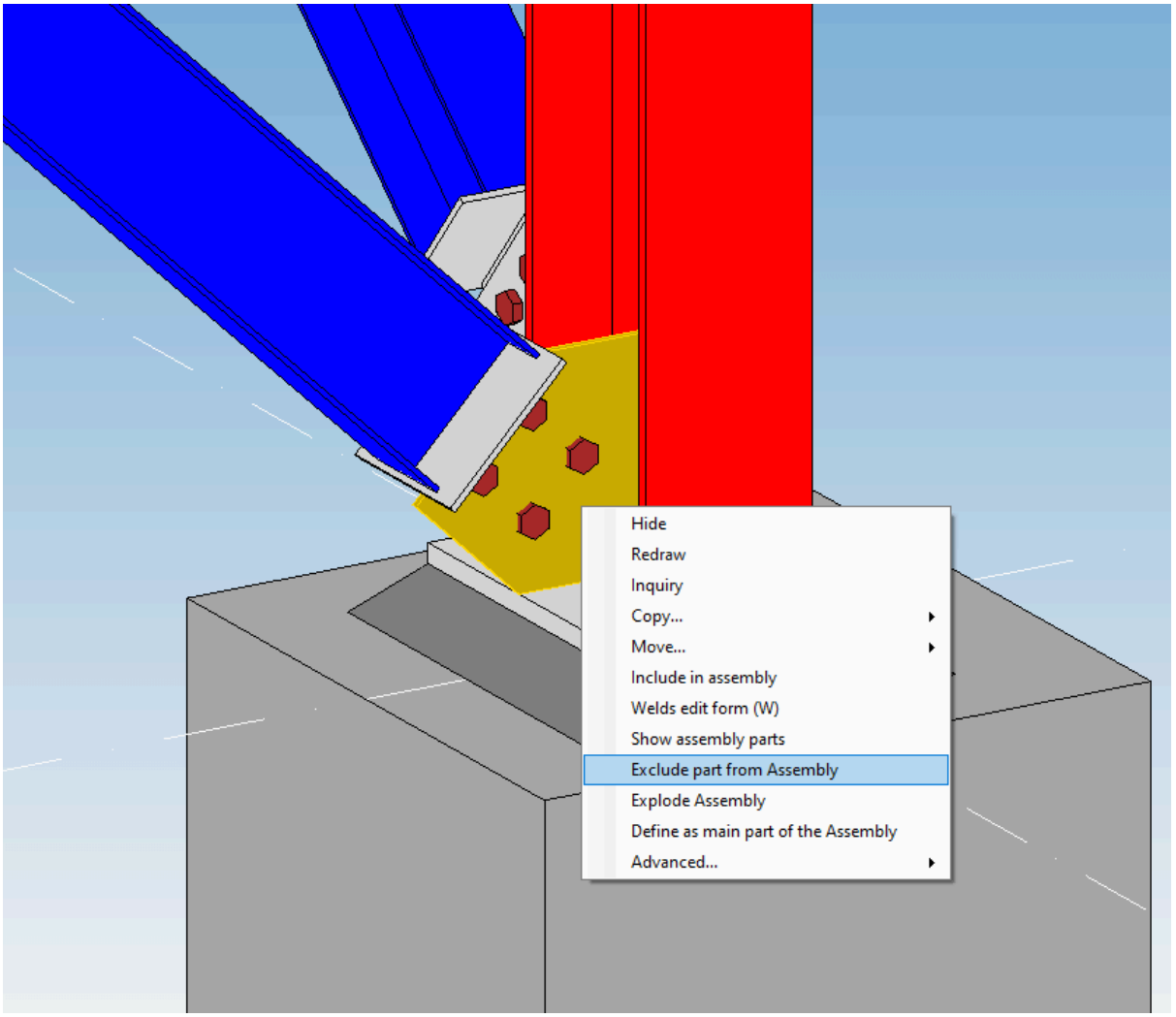


Automatic inclusion with soldering

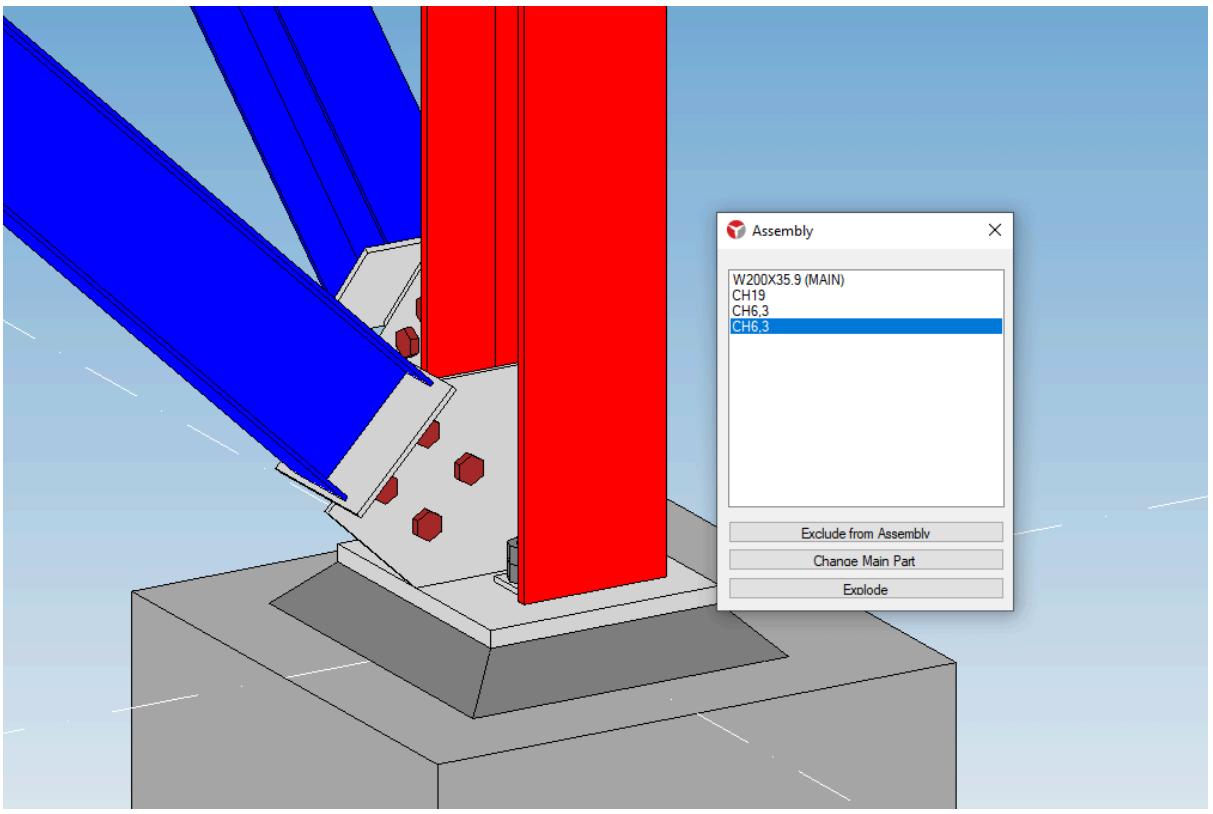
Whenever you create a weld between two parts, they automatically belong to the same assembly.*If you do not need the solder information in the model, use the solderless option above. Welds increase the size of the model, consume memory and slow down modeling when in large quantities.*

How to delete parts from a set

Right-click on the part you want to delete, and choose the option ‘Exclude part from Assembly’:



Another way is through the parts window. Click on the right button (see figure above) and choose ‘Show assembly parts’. You will see the following window:



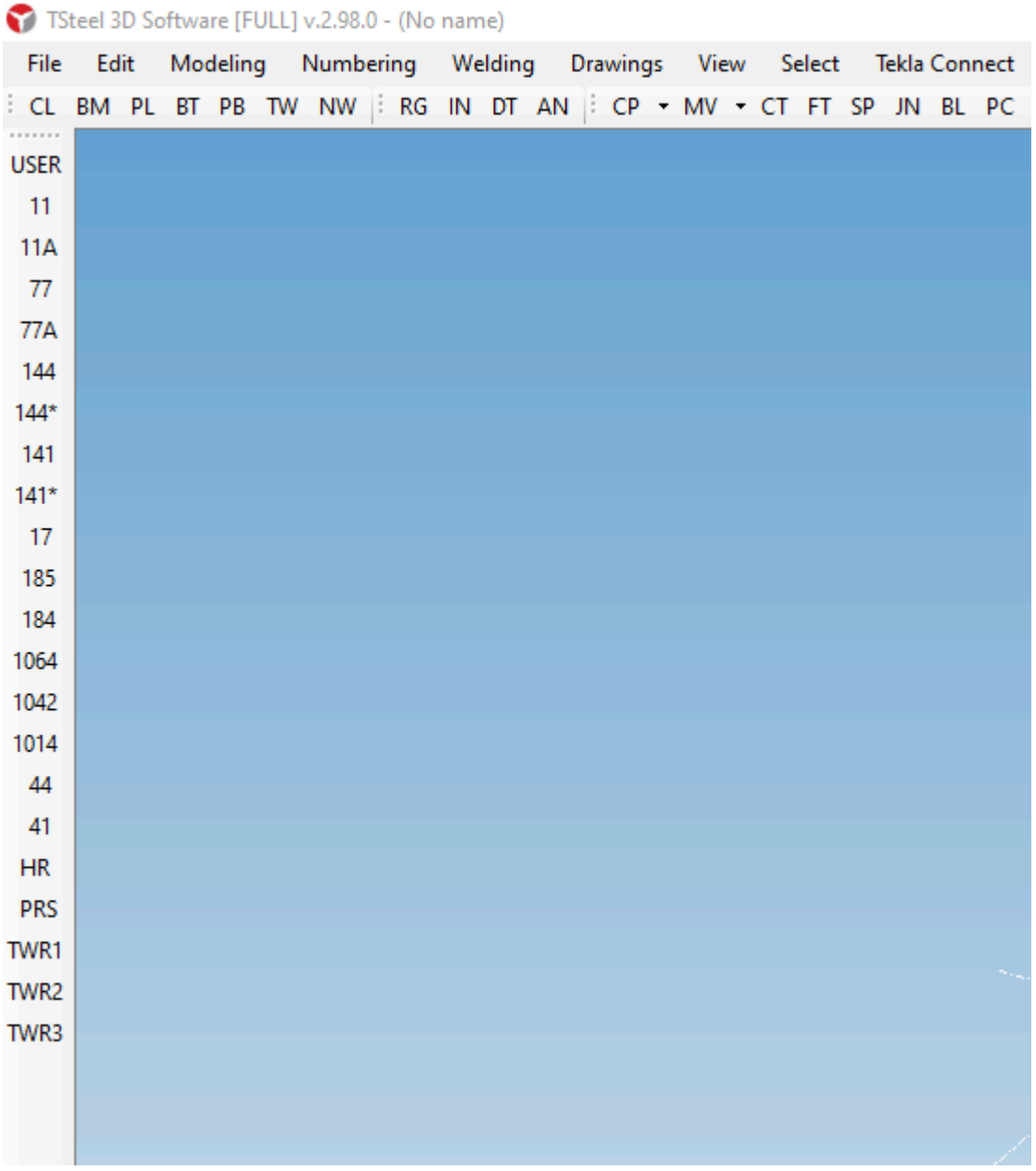
Through the parts window, you can select the parts you want to exclude and click on the ‘Exclude from assembly’ button.

Watch video on how to create and edit sets

Connection macros

Macros facilitate the creation of connections and structure types using input parameters.

The available macros are in the tool box on the left side of the screen:

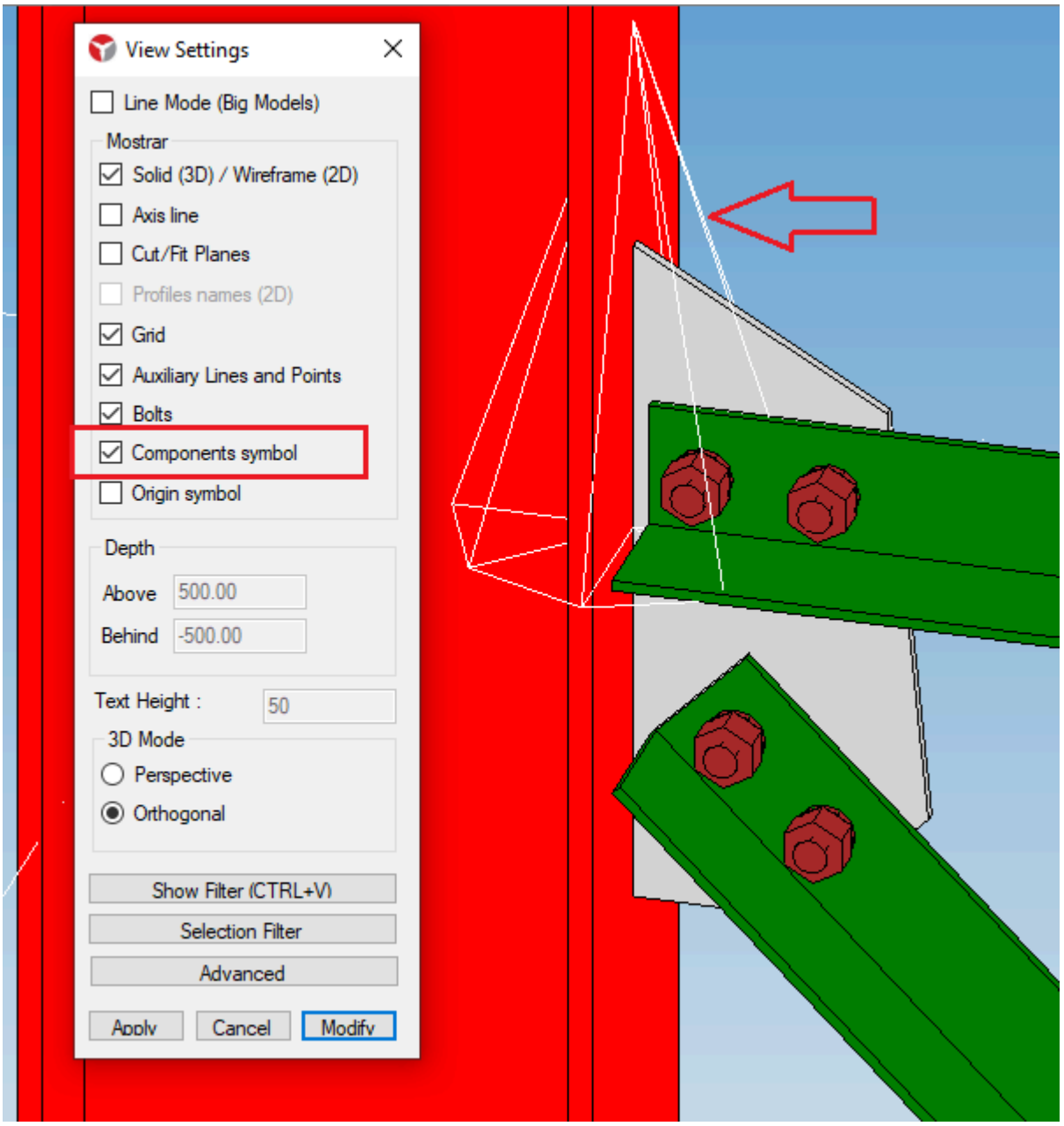


Each macro has a reference number.
By right-clicking on the macro, the macro configuration window opens. By clicking with the left button, the macro is executed.

How to edit macros

At the insertion point of each macro, the program shows a cone made of lines. This is the element you can select to delete the macro, or double-click to edit.

For macro symbols to be visible, the “Component Symbol” option must be turned on. See below an example of a macro symbol.

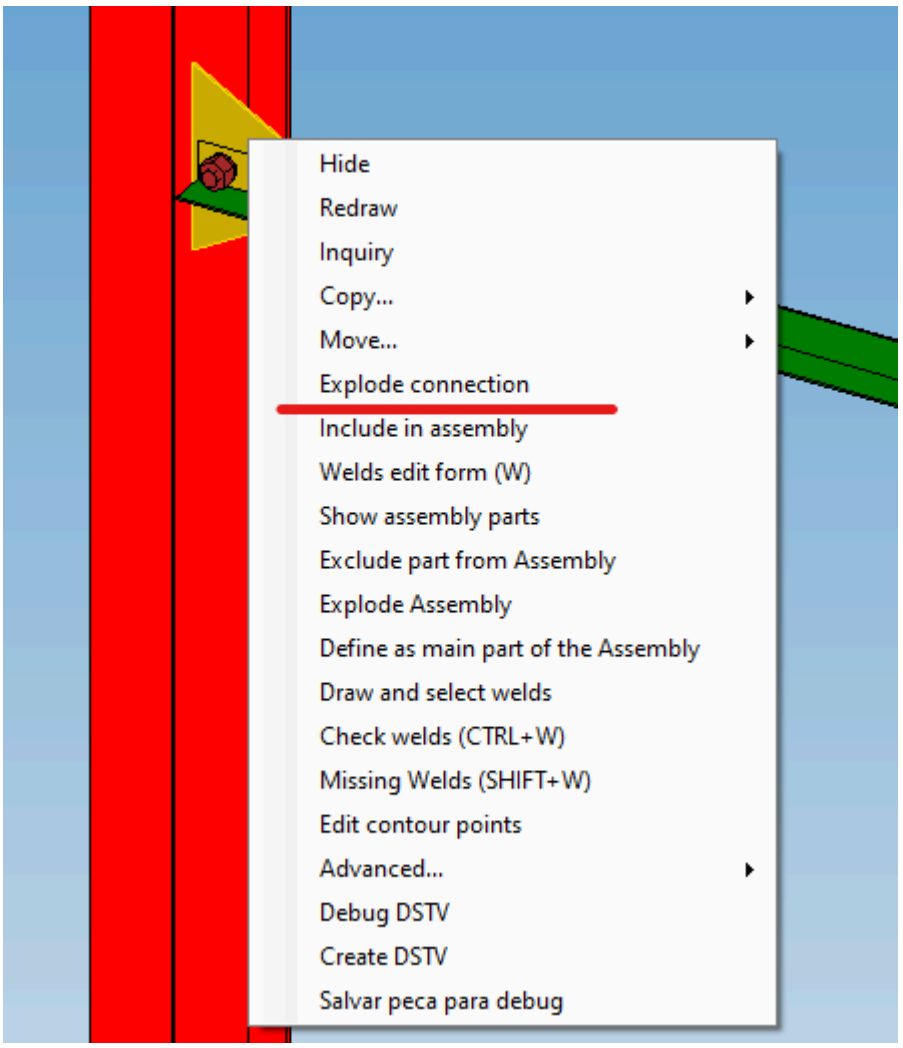


Explode Macro

Exploding a macro is to delete the macro without deleting the cuts, plates and screws created by the macro. You cannot edit the connection through the macro form, but you can make any manual changes you need.

You gain all the work done by the macro and can make the adjustments you need but are not available automatically.

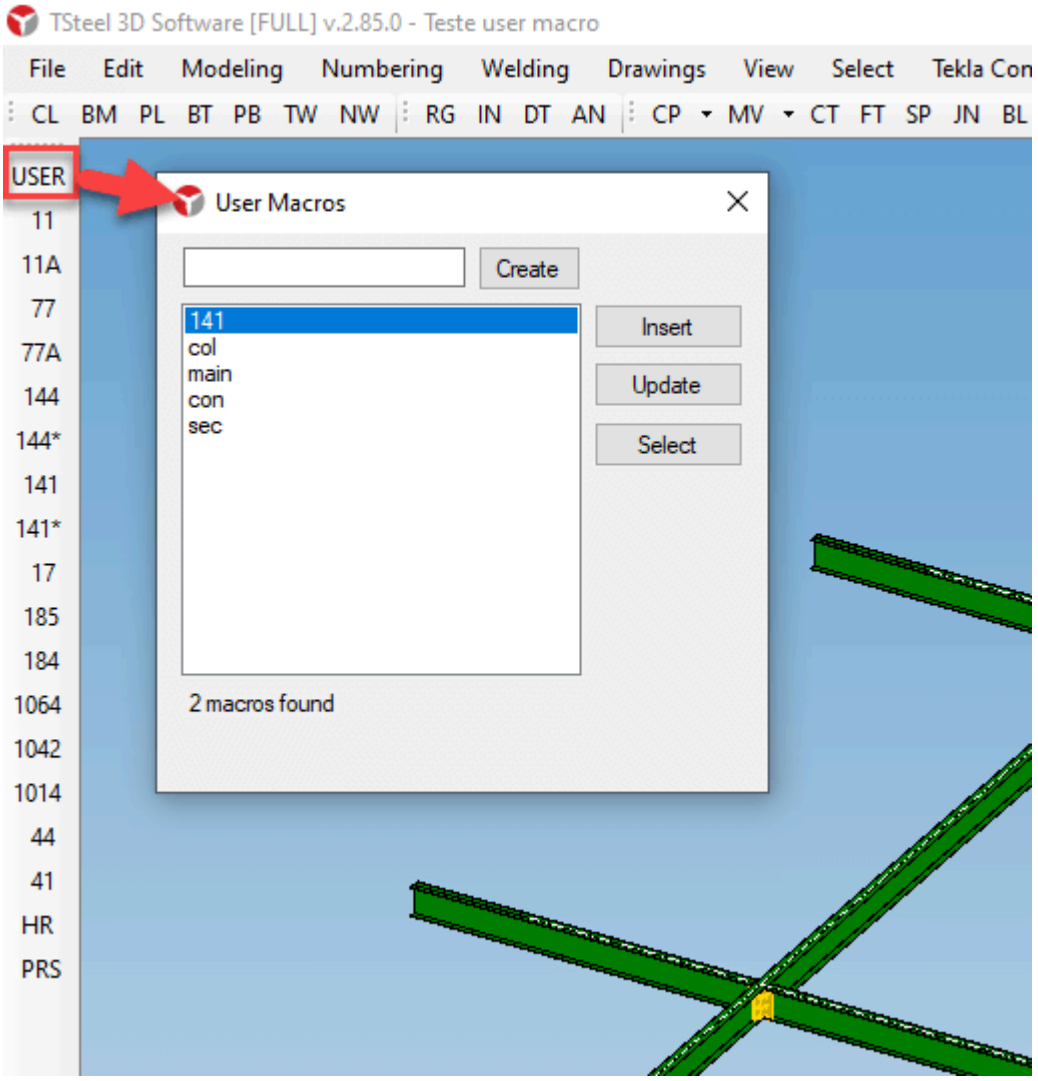
To explode a macro, right-click on a part of the macro (or on the macro icon) and choose “Explode connection”.



Use Macro - User Macro

When the user places a reference name in an already executed macro, this macro starts to have some additional characteristics:

1. You can run the macro with the parameters already defined.
2. You can change all macros that have the same name at once. Just change one of them and request an “Update”

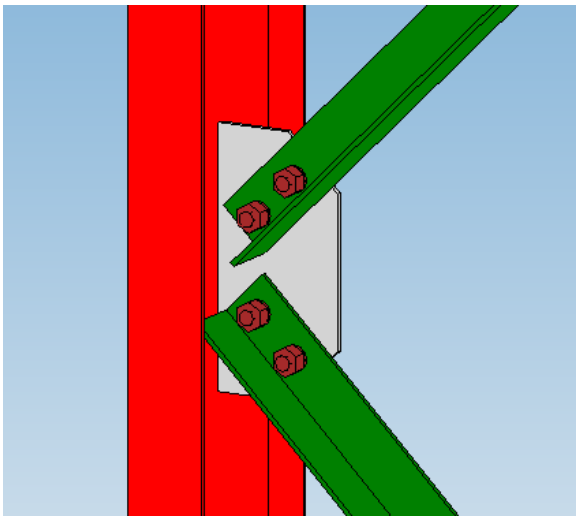
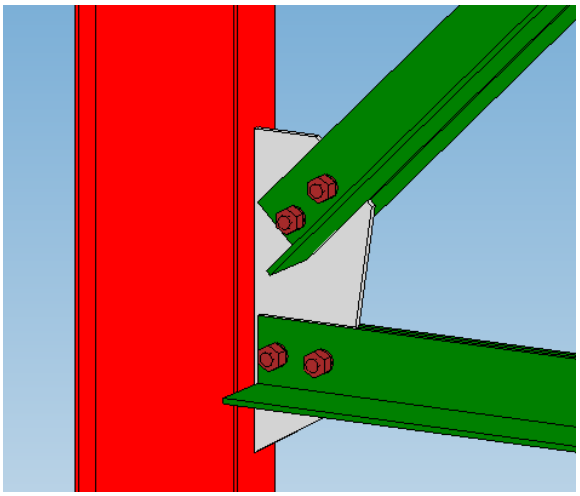


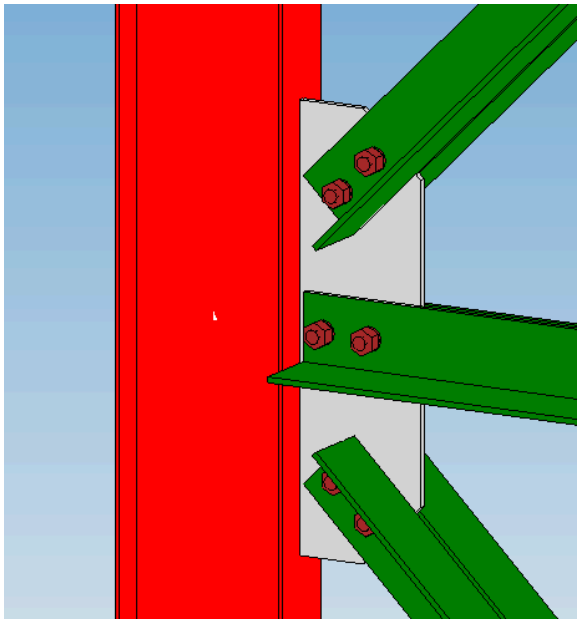
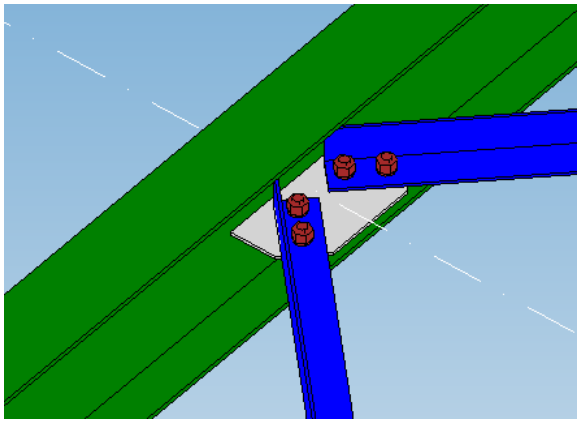
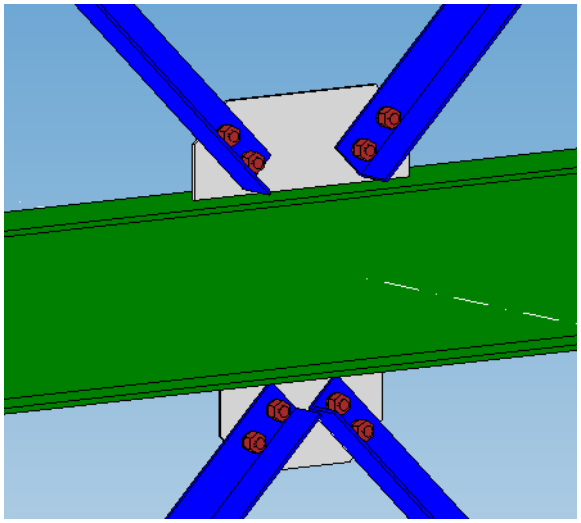
In the window you can create new user macros (just enter a valid name and click “Create”).
The “Insert” button runs new macros from the macro selected in the list.
“Update” updates all macros based on one selected by the user.
The “Select” button allows you to select a macro in the model and find out its name.

When the thickness field is empty, the program will choose the thickness automatically.

Macro 11 - Angles arriving in column or beam

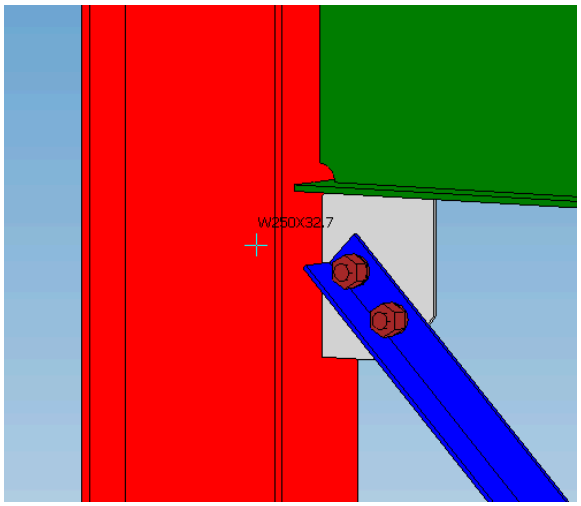
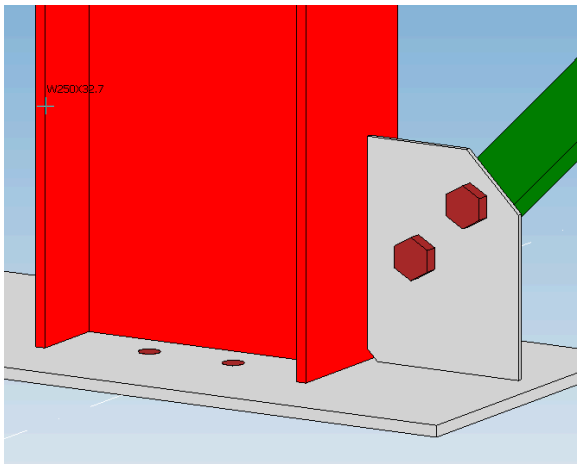
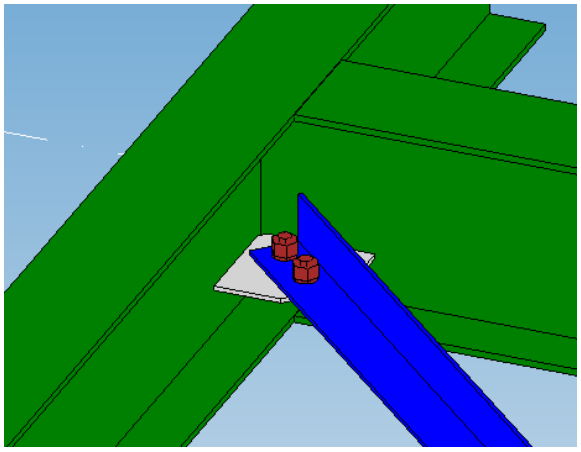
Some examples of connections made with this macro:

	Angles reaching the soul of a column
	Double angles (TWIN BEAMS) arriving in a column

	Angles arriving in a column
	Angles reaching the side of a beam
	Angles arriving in beam on the upper or lower flange

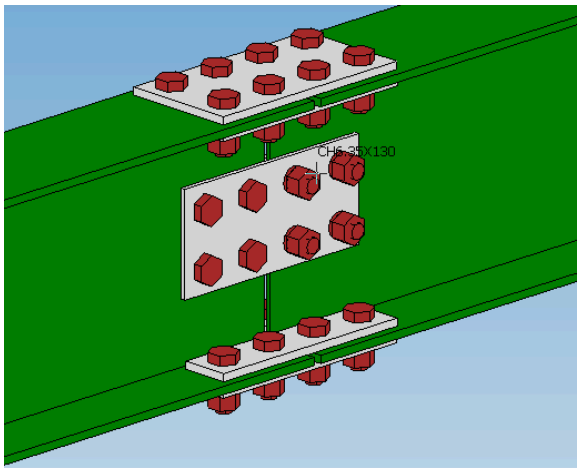
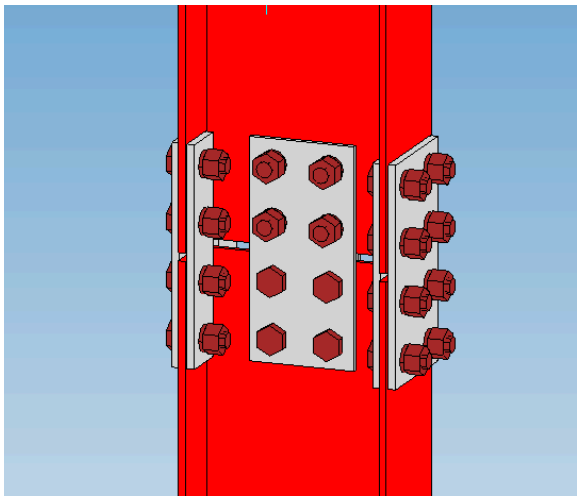
Macro 11A - Special case of macro 11

Some macro examples:

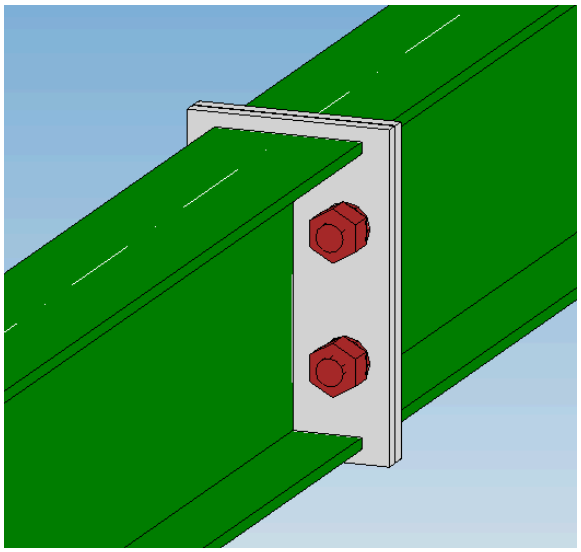
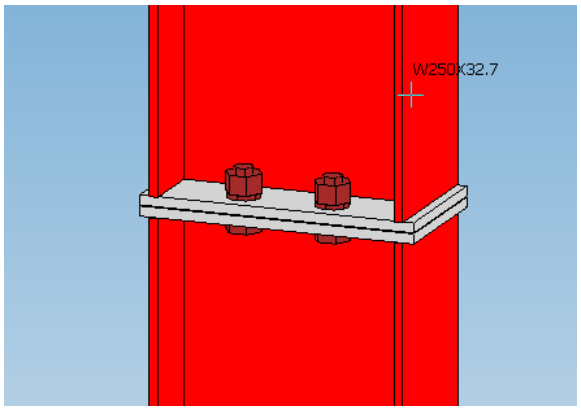
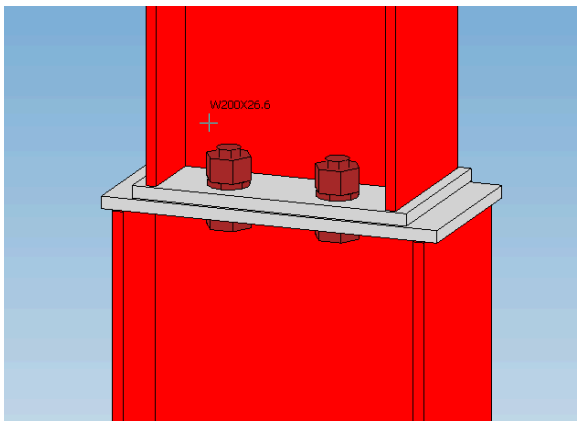
	Angle iron arriving in a “corner between column and beam
	Arrival of angle iron on column base
	Angle bar arriving at the meeting of two beams

Macro 77 - Splice with I or W profile splint

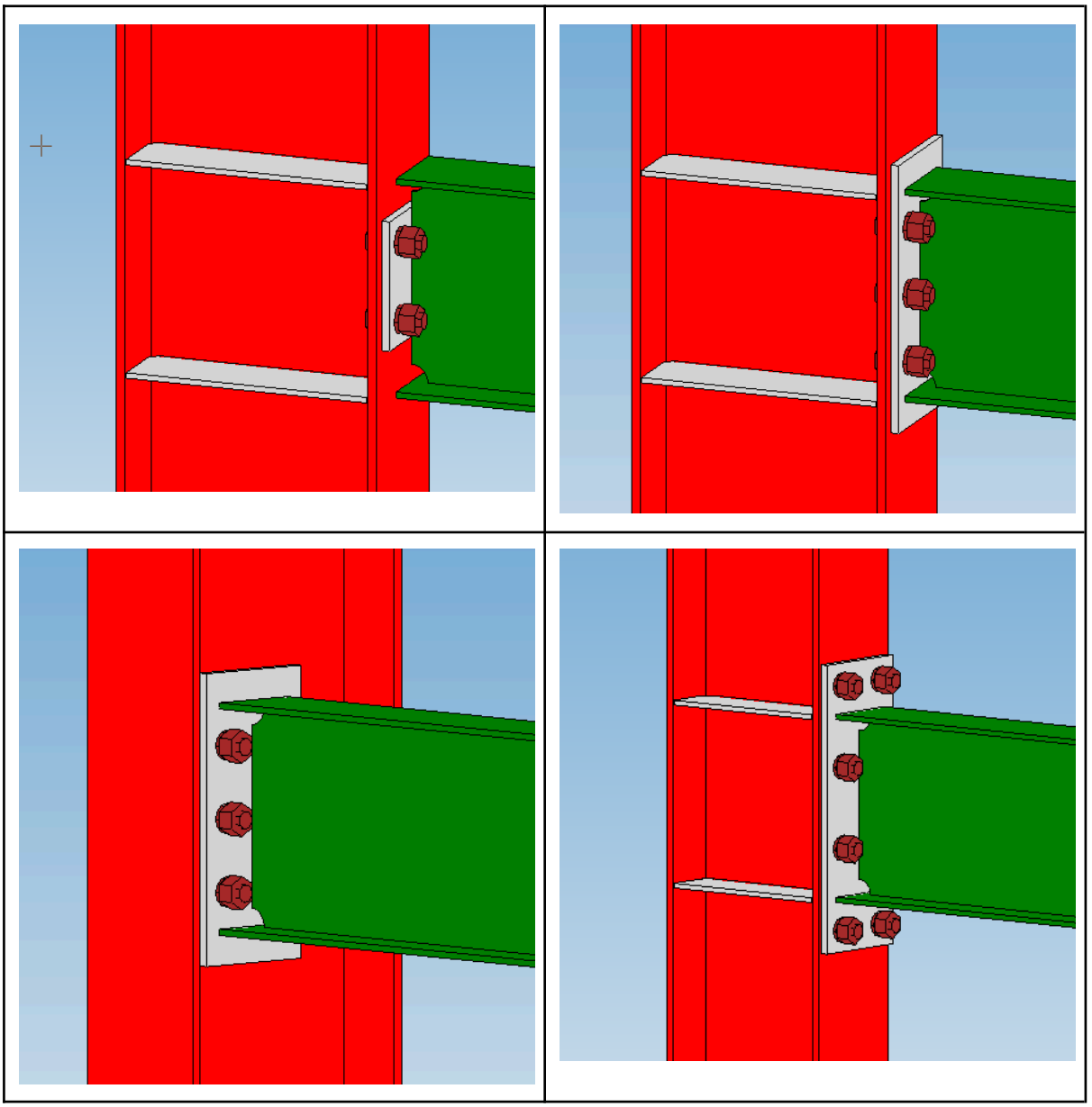
Macro examples:

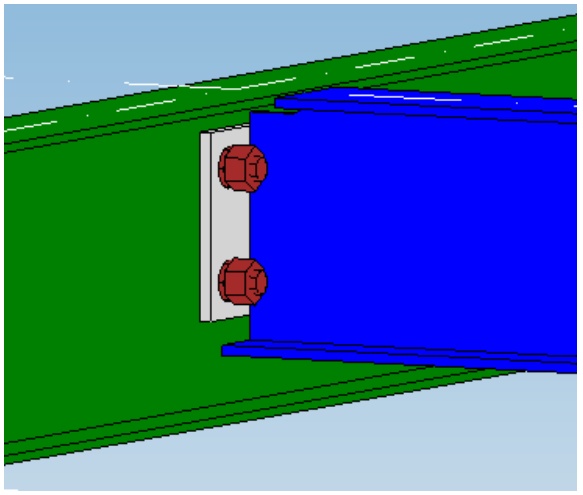
	Splice between W beams of the same gauge
	Splicing of columns

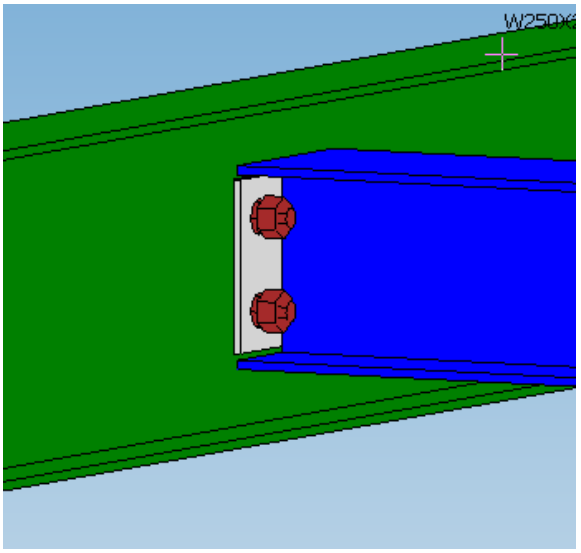
Macro 77A - Splicing with top plate for I or W profiles

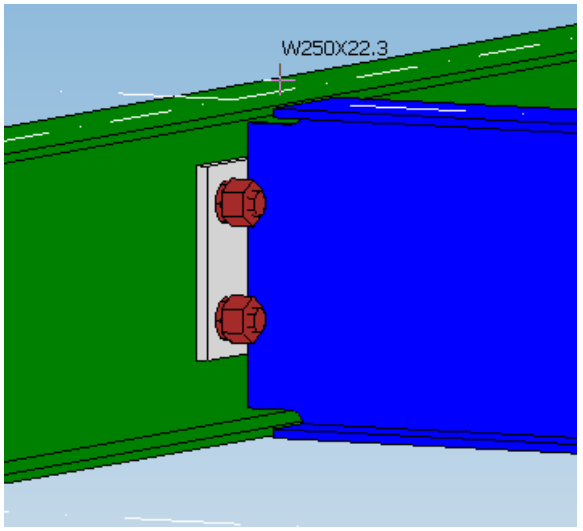
	Beam splicing
	Column Splice
	Splice columns with different gauges

Macro 144 - Connection with top plate

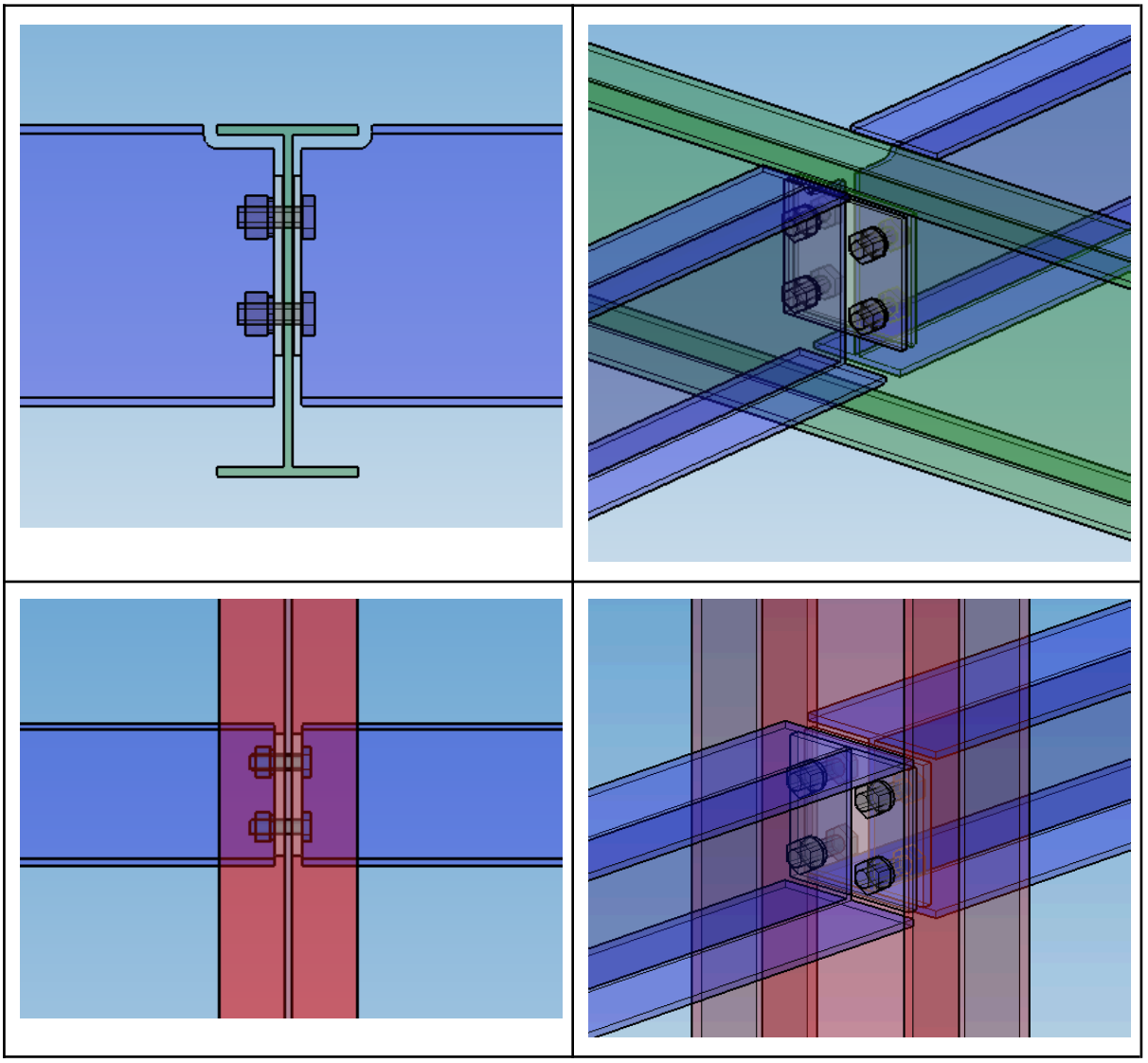




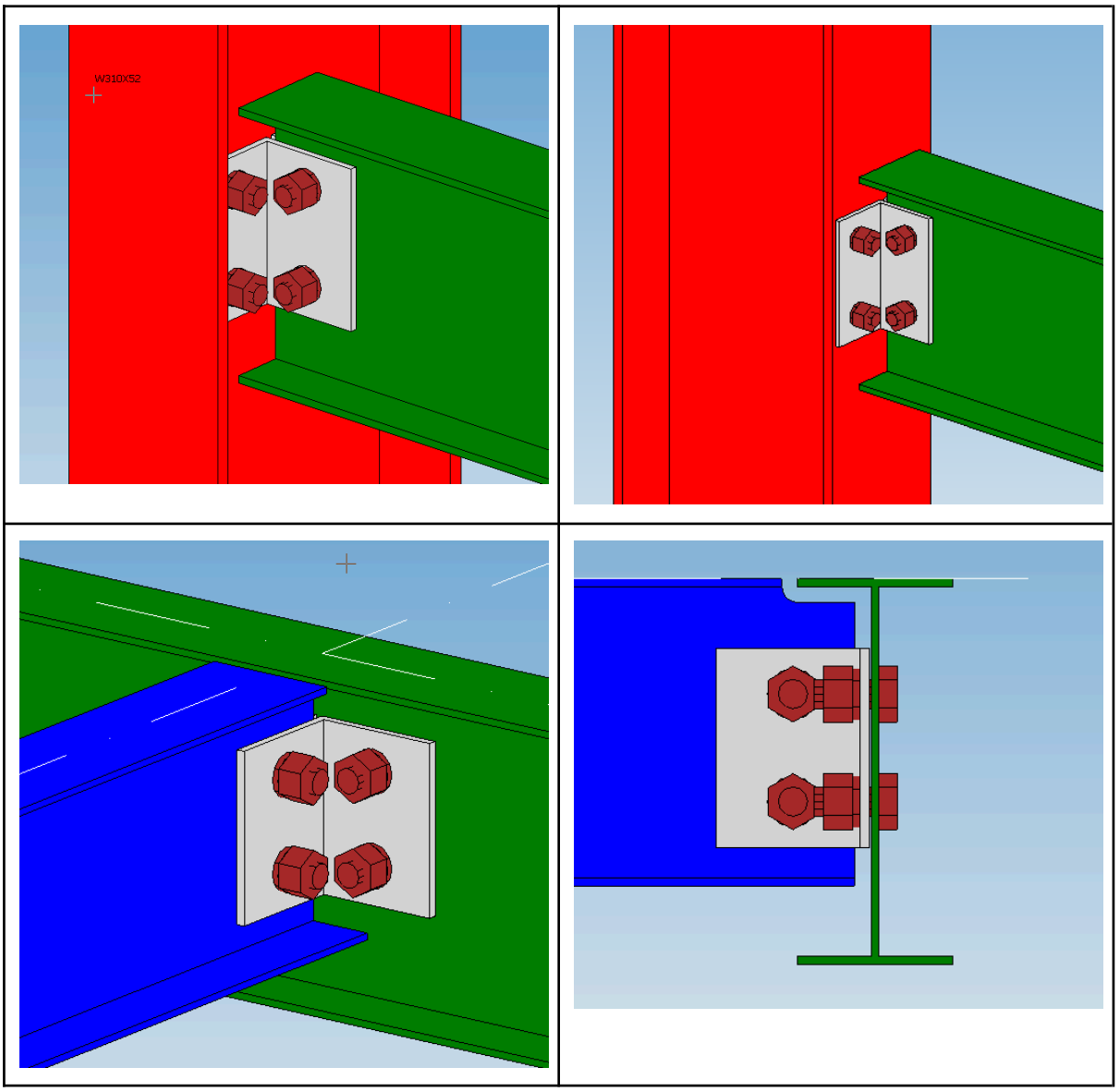



W250X22.3

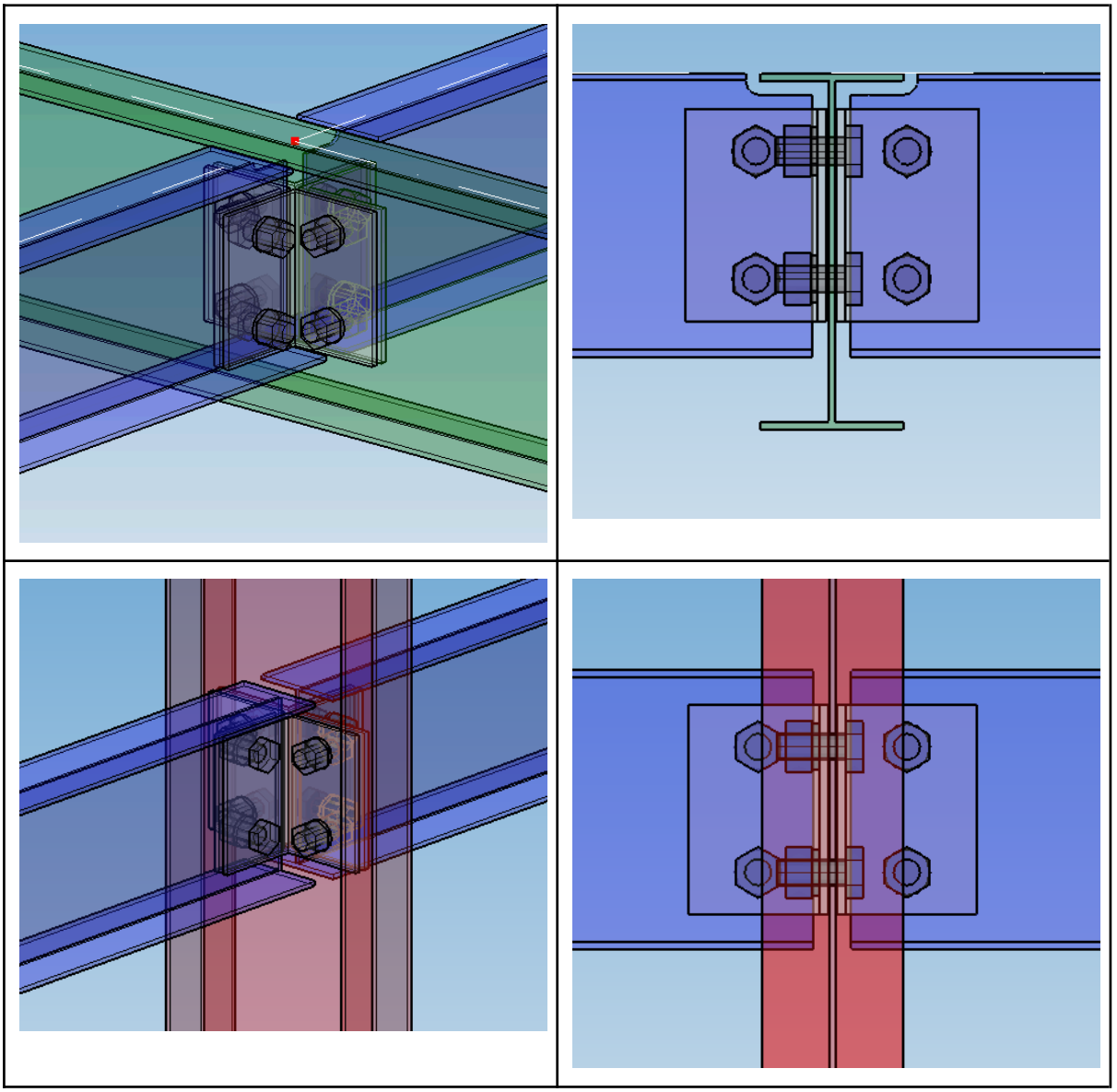
Macro 144* - Special case of 144



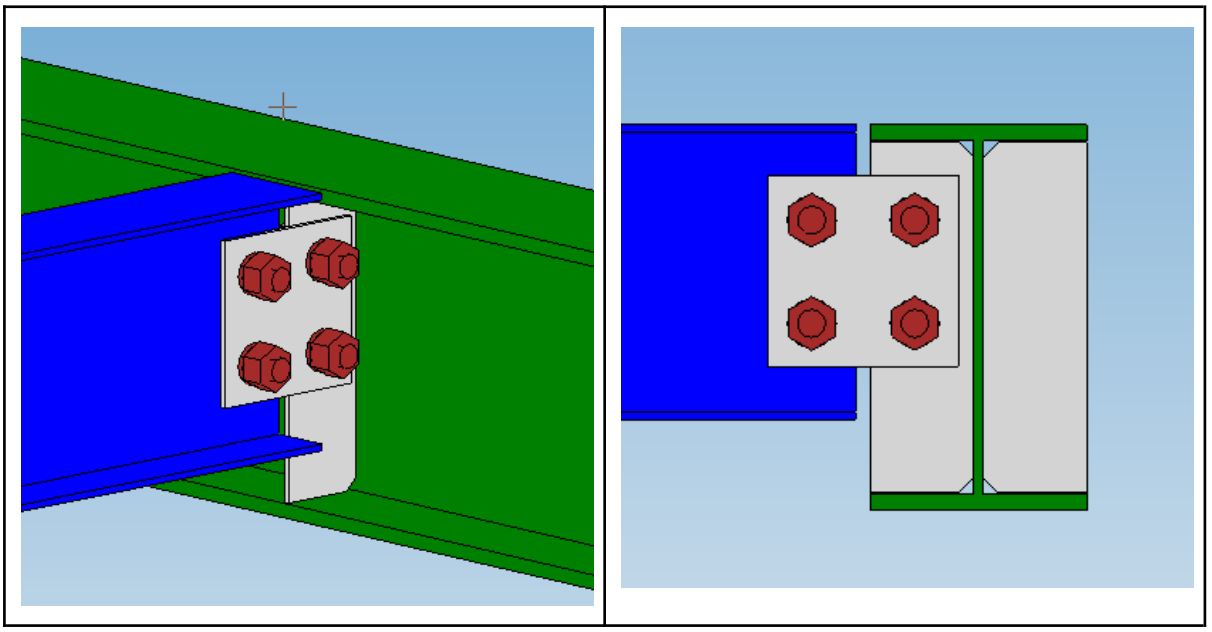
Macro 141 Connection with angles



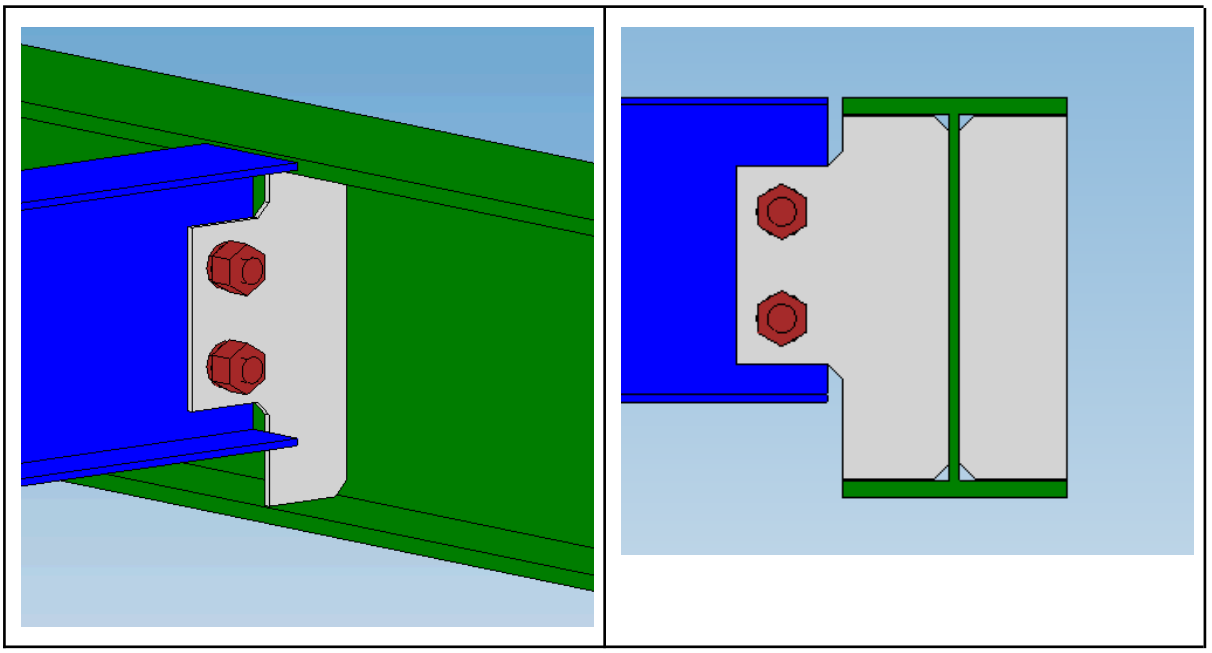
Macro 141* (Special case 141)



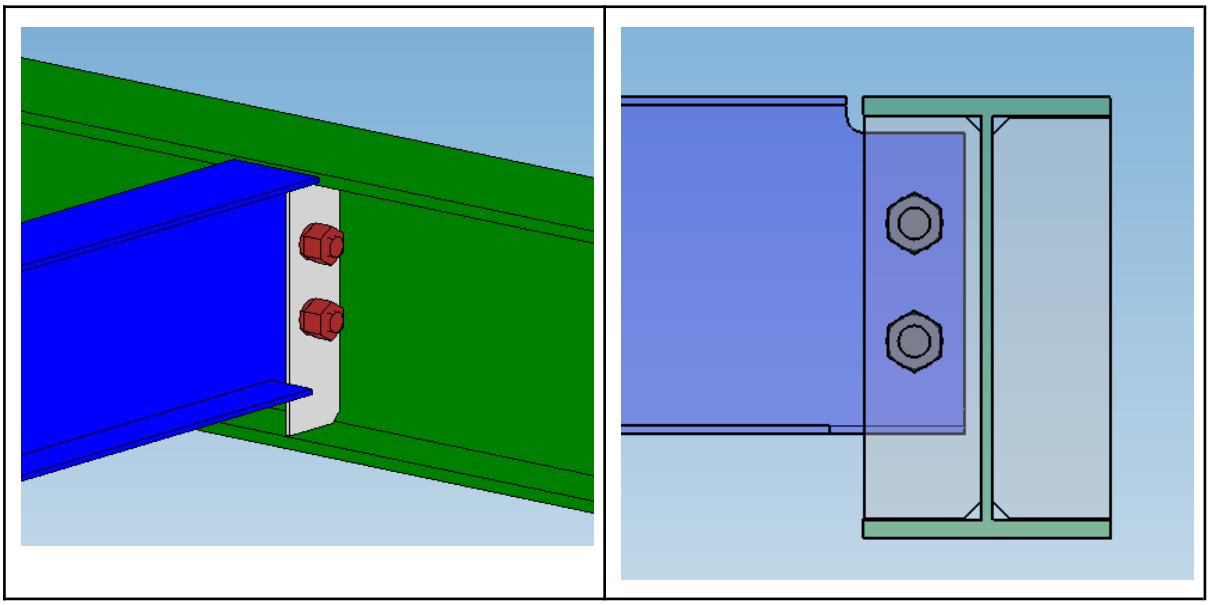
Macro 17 - Connection between beams



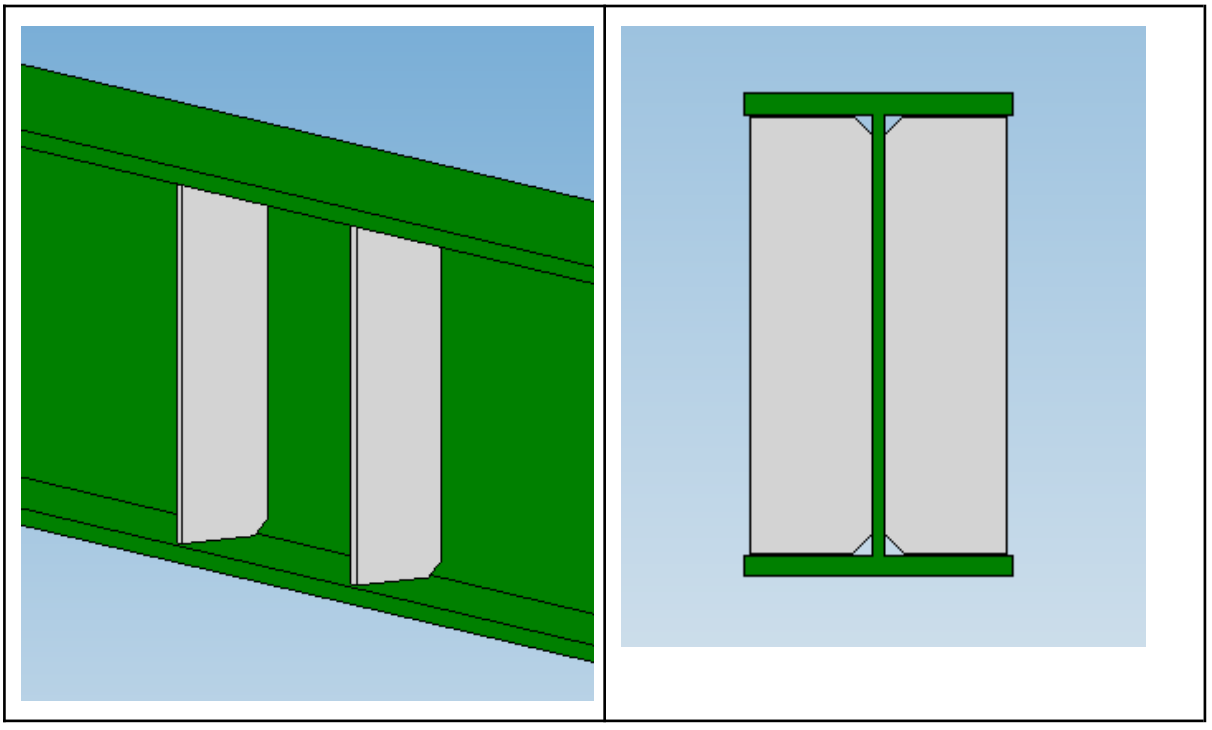
Macro 185 - Connection between beams



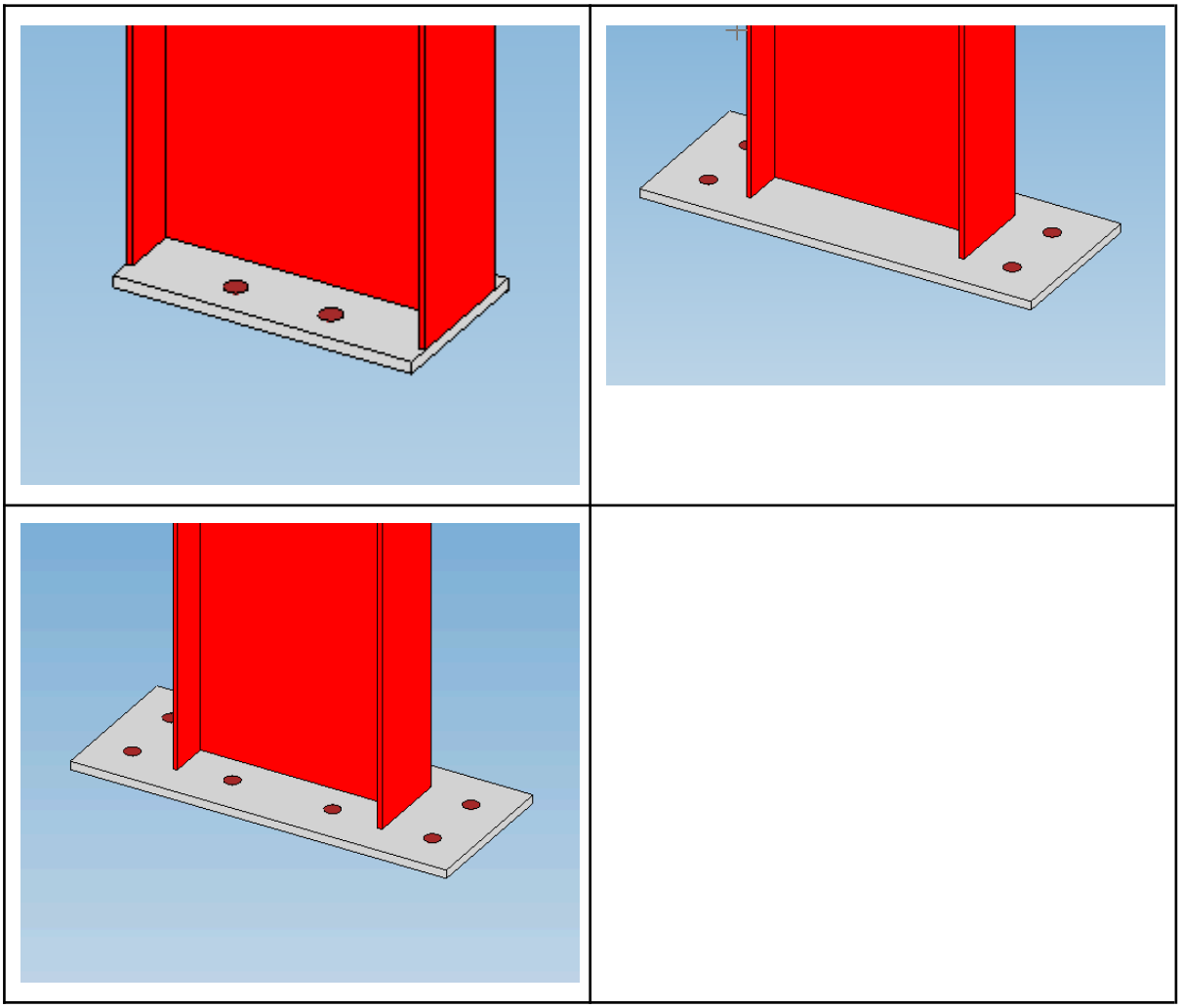
Macro 184 - Connection between beams



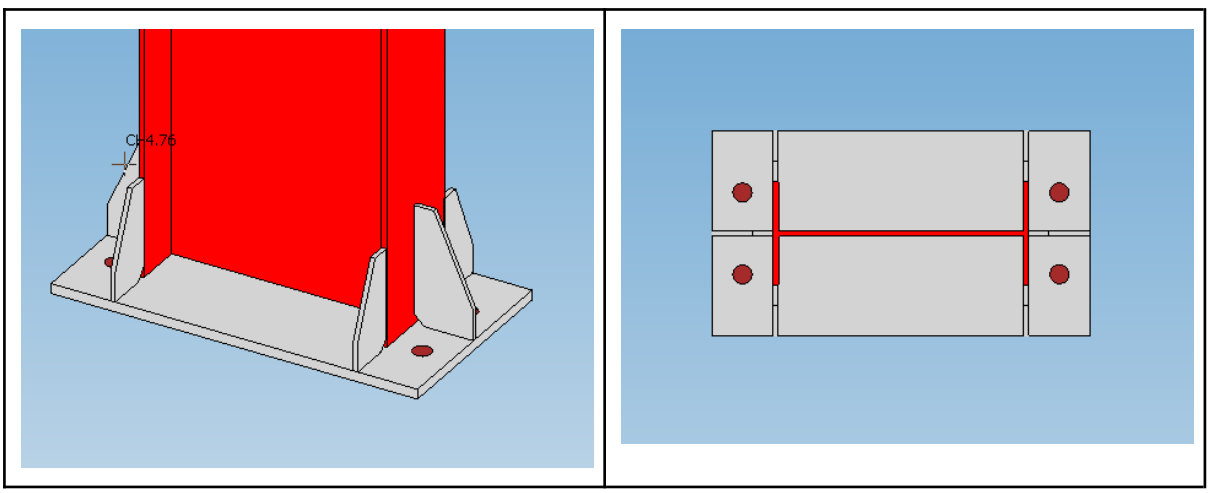
Macro 1064 - Stiffeners



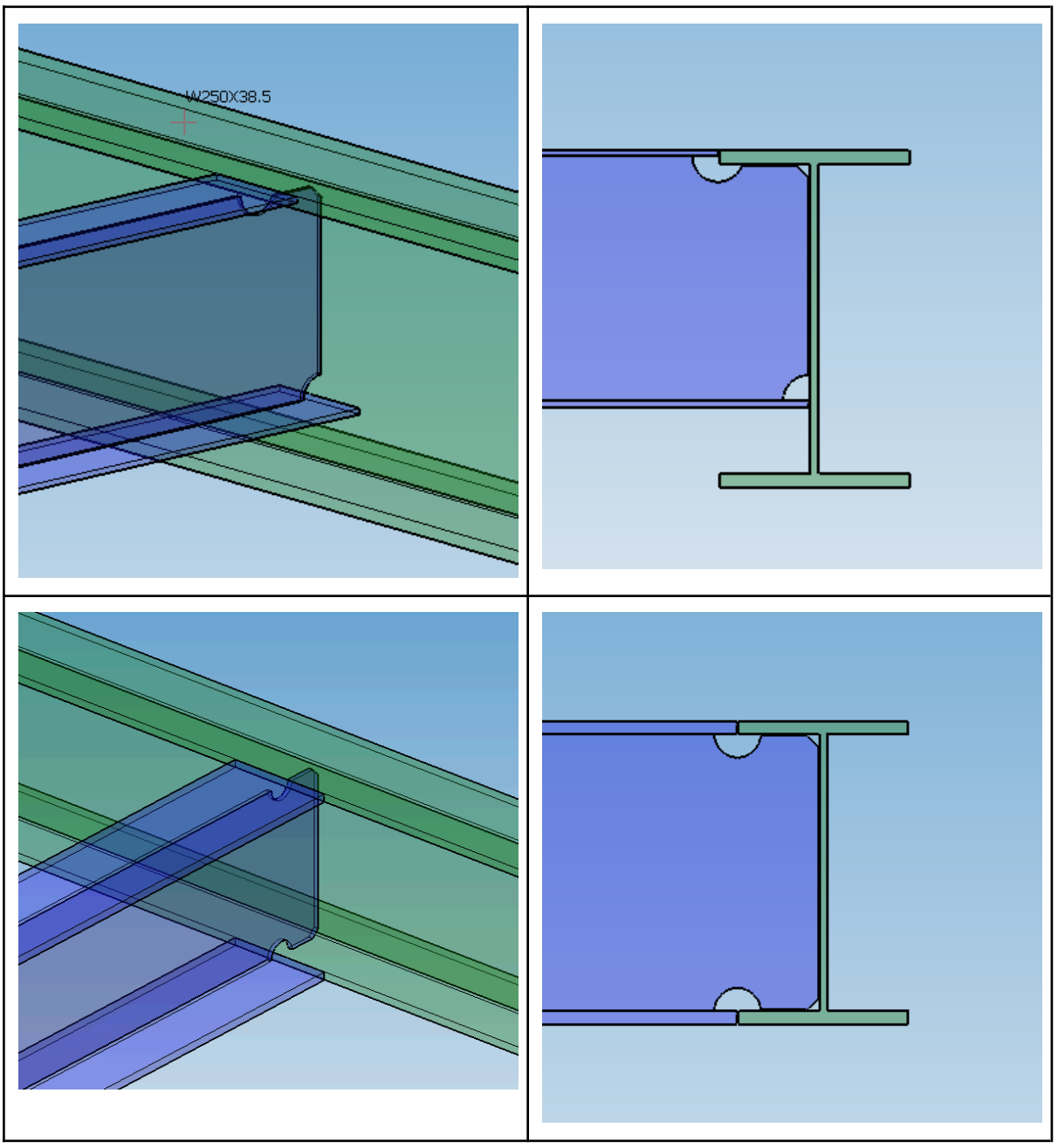
Macro 1042 - Column base

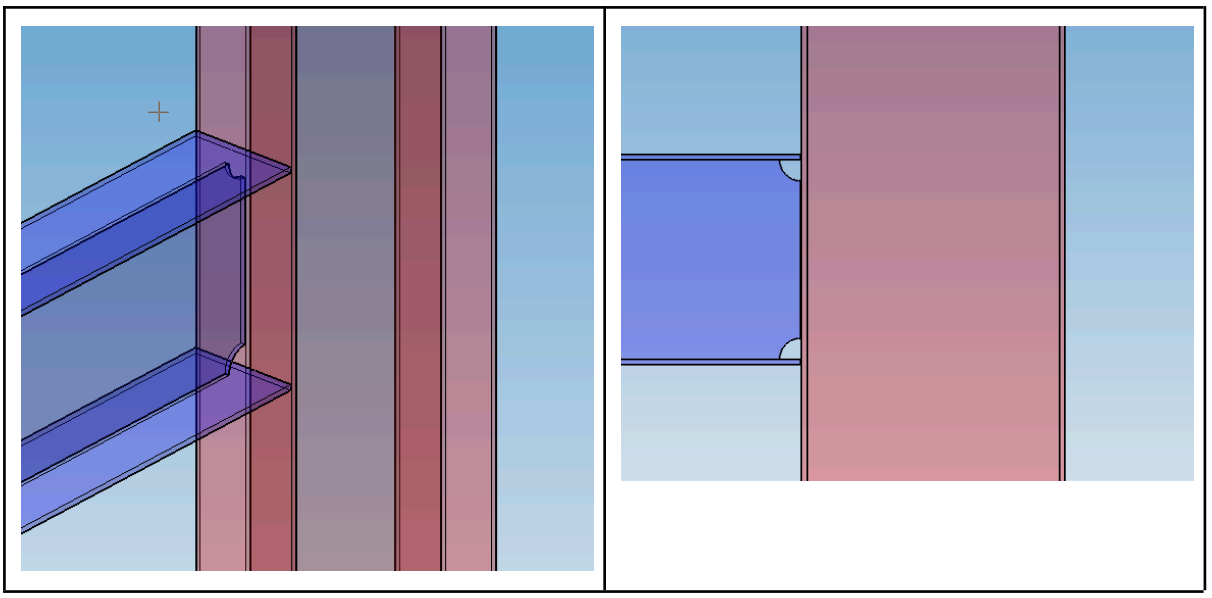


Macro 1014 - Column base with stiffener

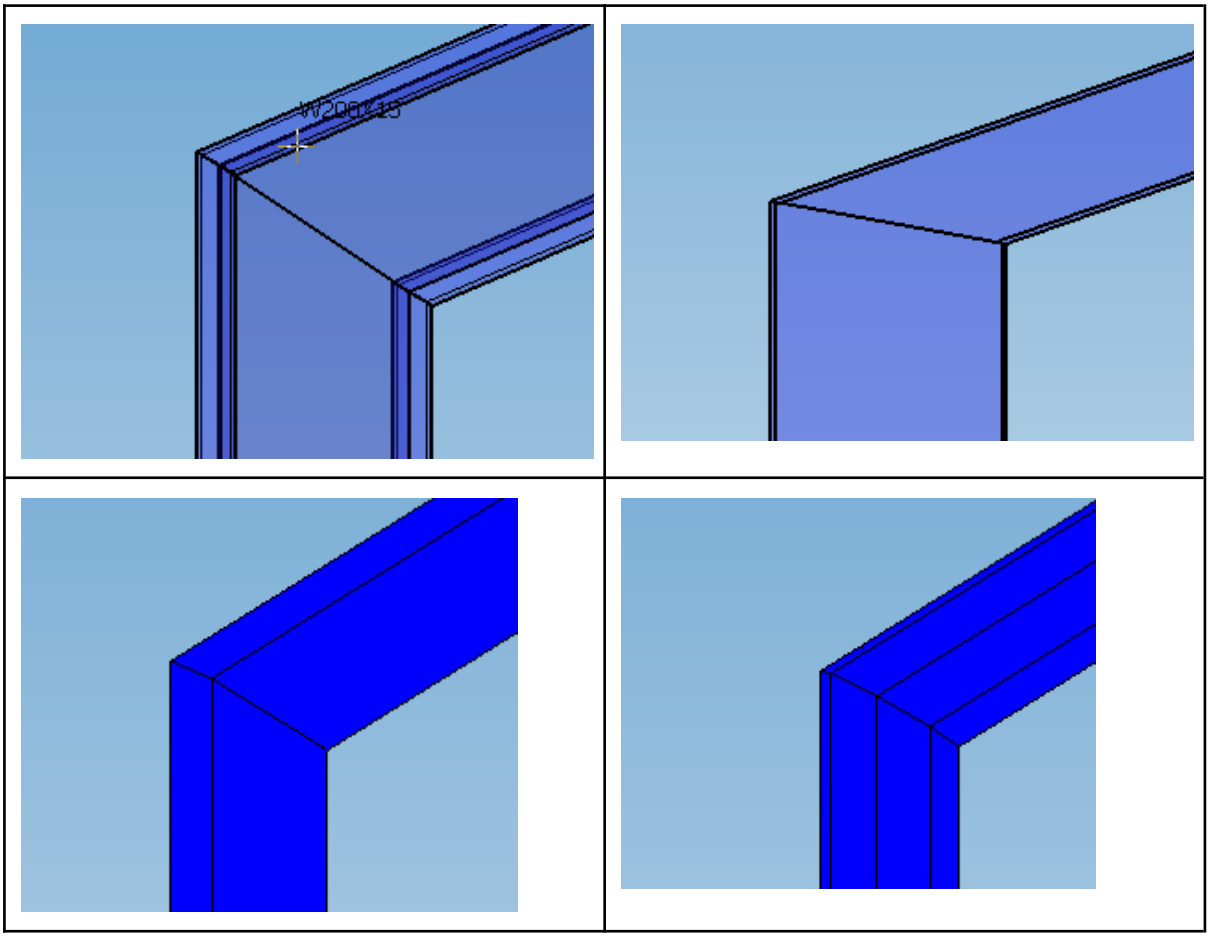


Macro 44 - Welding cutouts



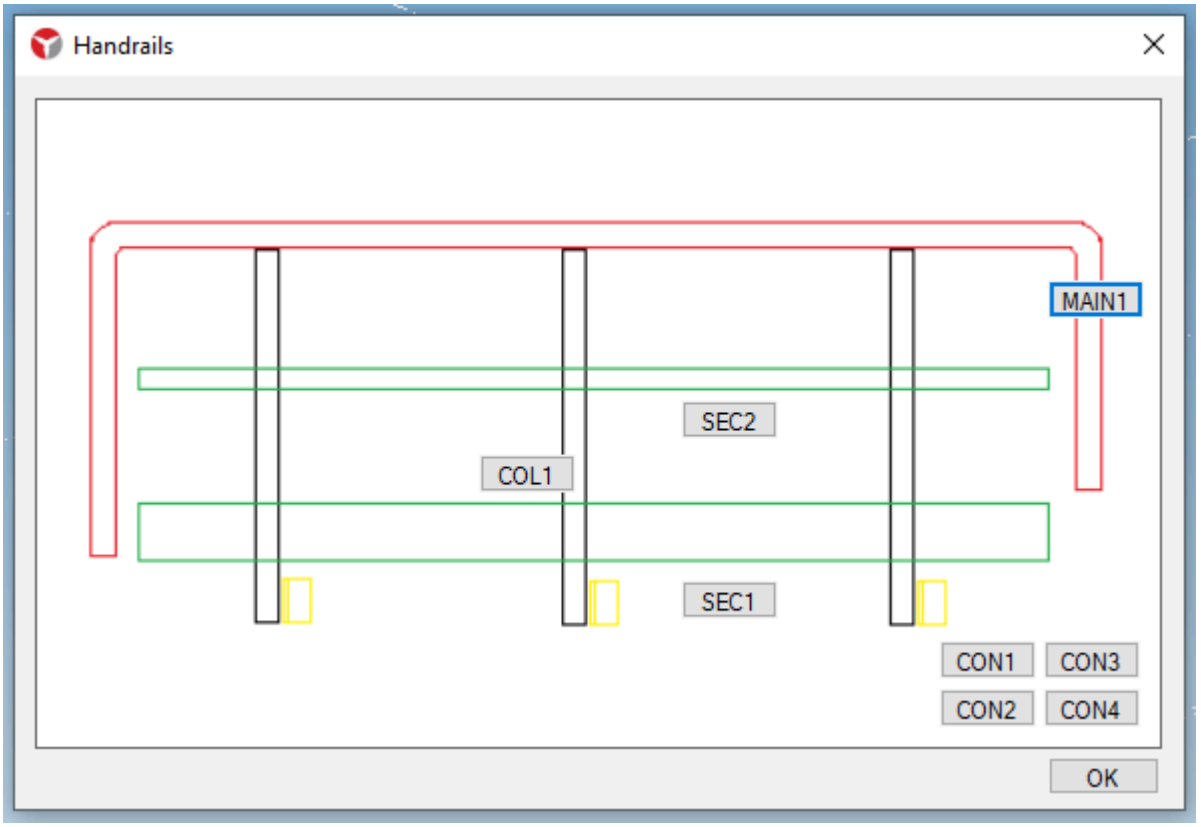


Macro 41 - Miter adjustment



Macro HR (Handrail = Bodyguard)

The railing macro brings together several intermediate macros:

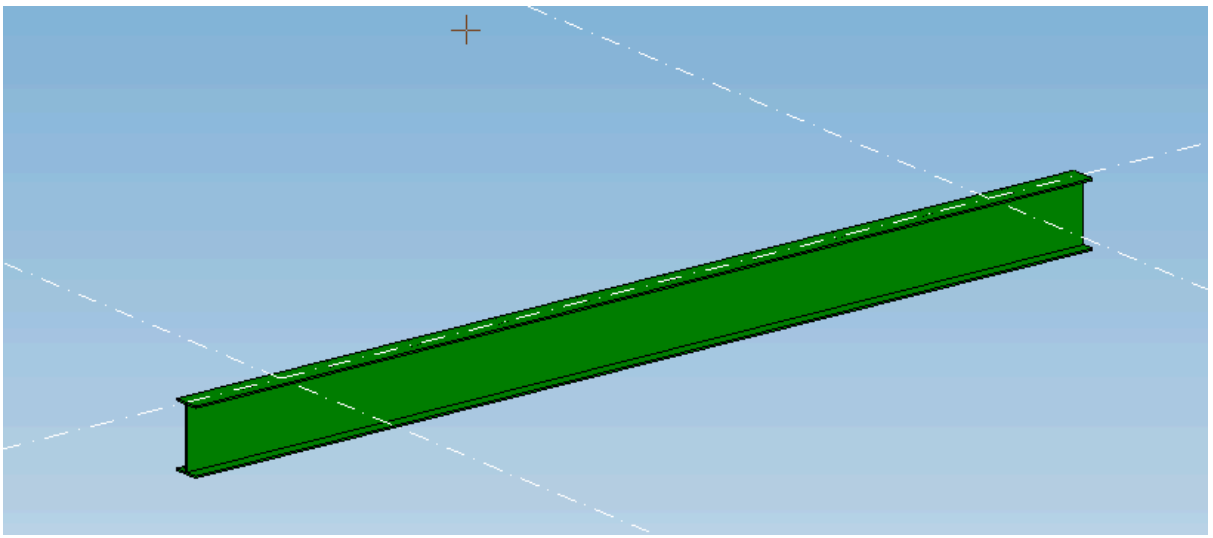


The construction of the guardrail is carried out in stages:

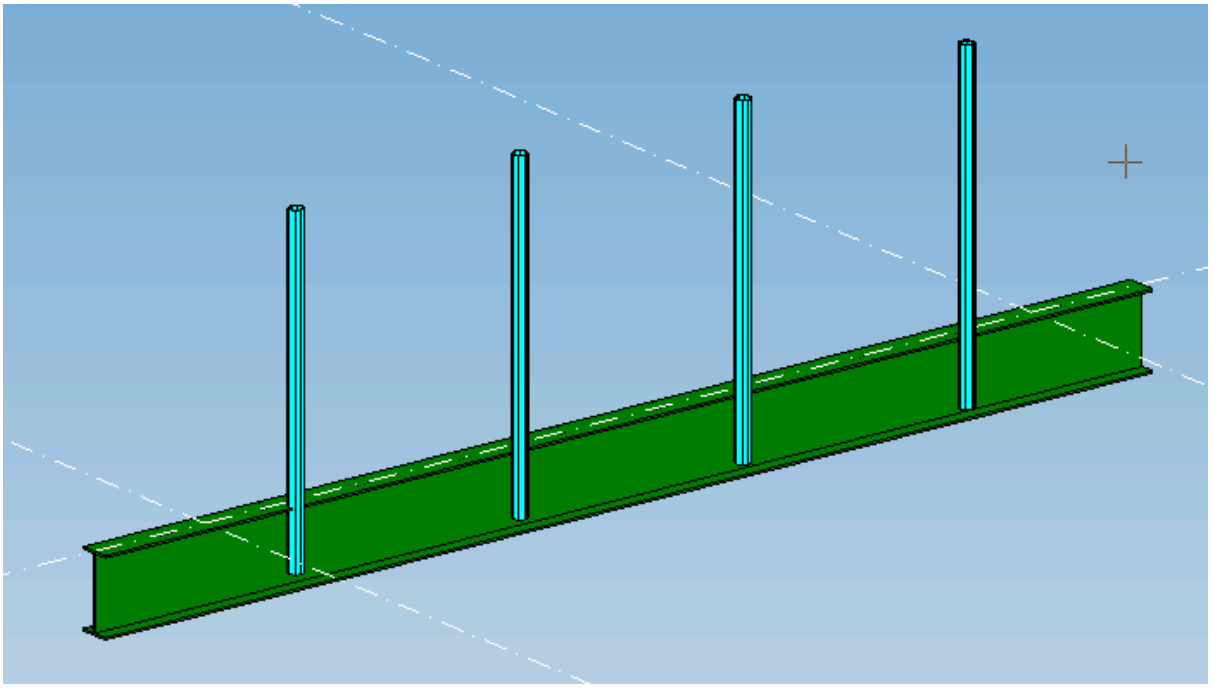
1. Defines the amounts (COL1)
2. Makes connections to the uprights in the structure (CON1, CON2, CON3, CON4)
3. Define a barra principal (MAIN1)
4. Includes footer (SEC1)
5. Includes secondary bar (SEC2)

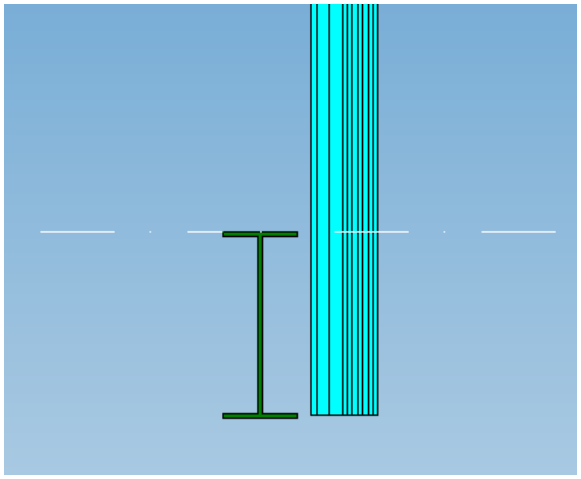
Installing a horizontal beam guardrail step by step

We have the following beam where we will model a guardrail.

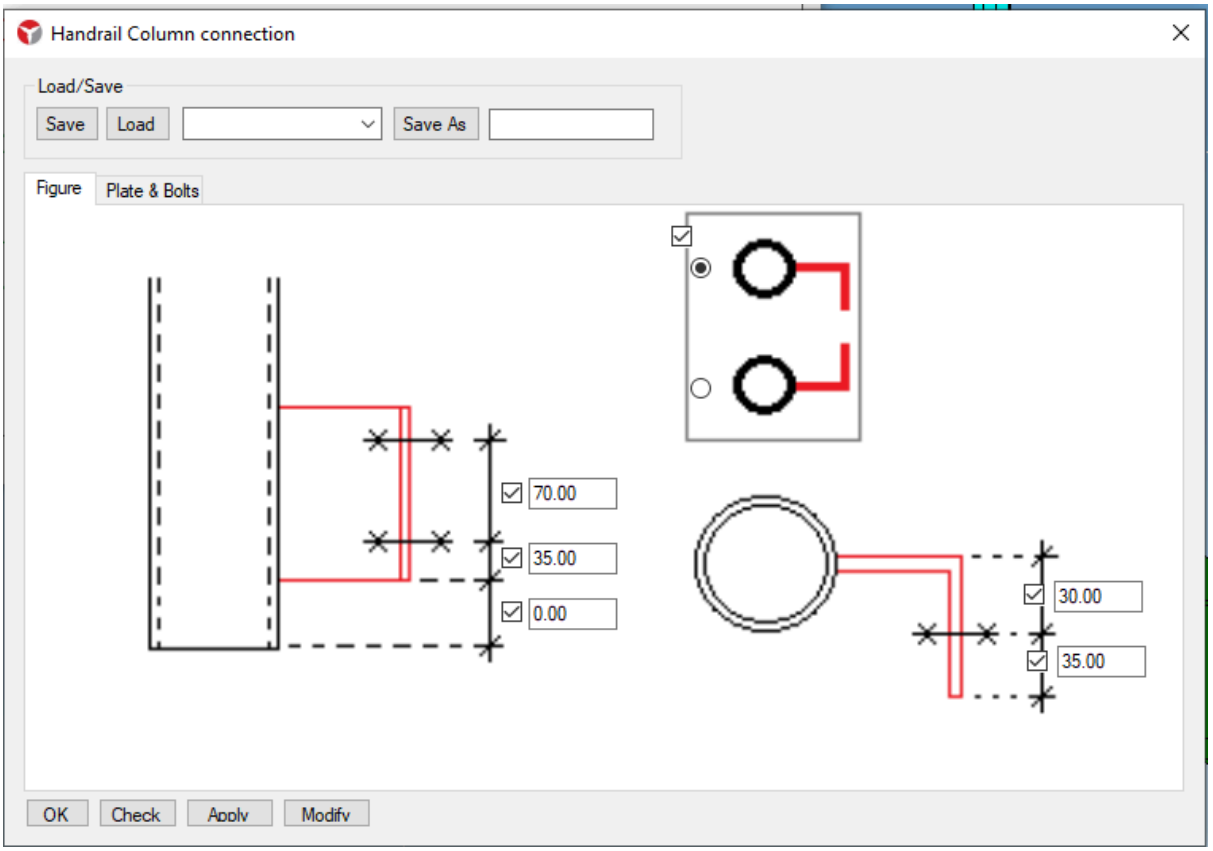


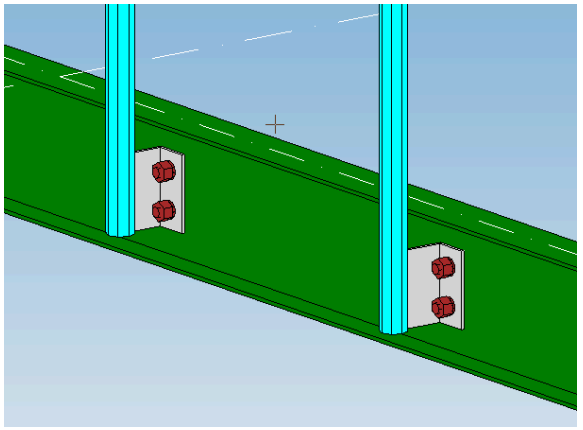
Let's click on COL1 and select the GC start and end points. The default suggestion appears like this:



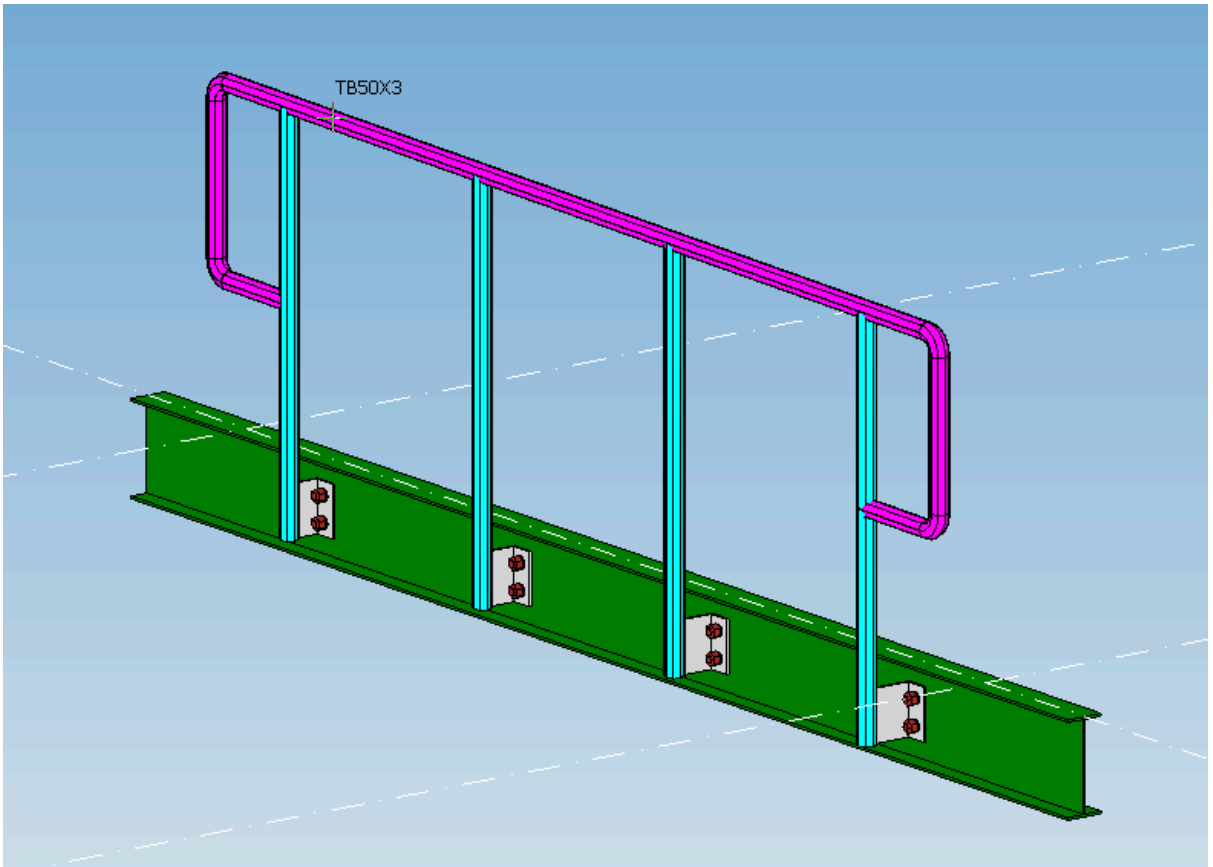
	Let's adjust the macro parameters so that the column is correctly distanced in relation to the beam.

Let's connect the uprights to the beam. As I am using tubes for the uprights, I will use CON3.



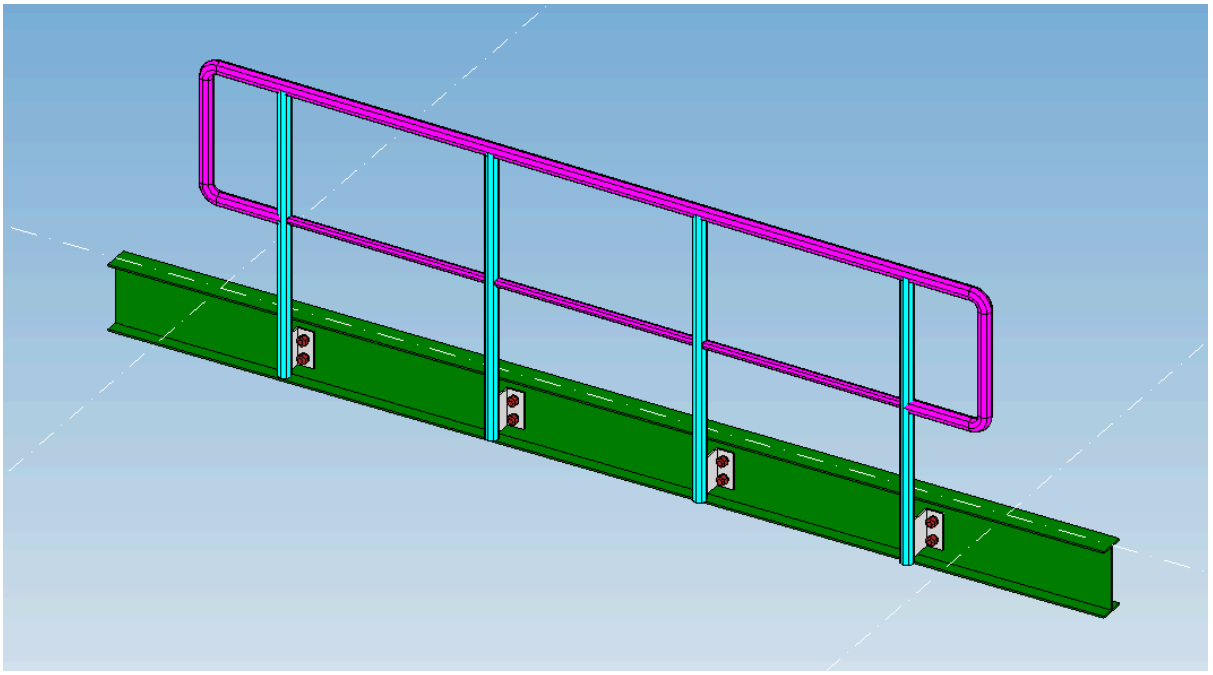
	Select the beam once and then the uprights one by one. The advantage of connecting the uprights individually is that you can configure these connections individually to avoid any interference.

Now let's insert the main bar through the MAIN1 macro.

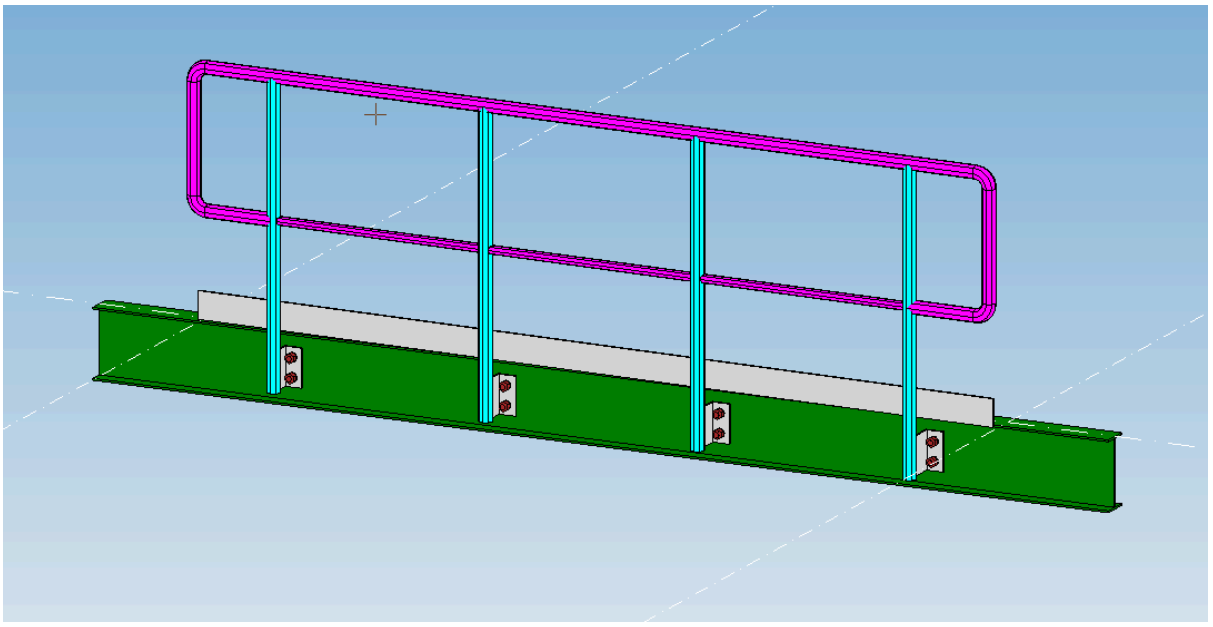


It is important that you explore all the variants that this macro has and that allow the construction of different types of guardrails.

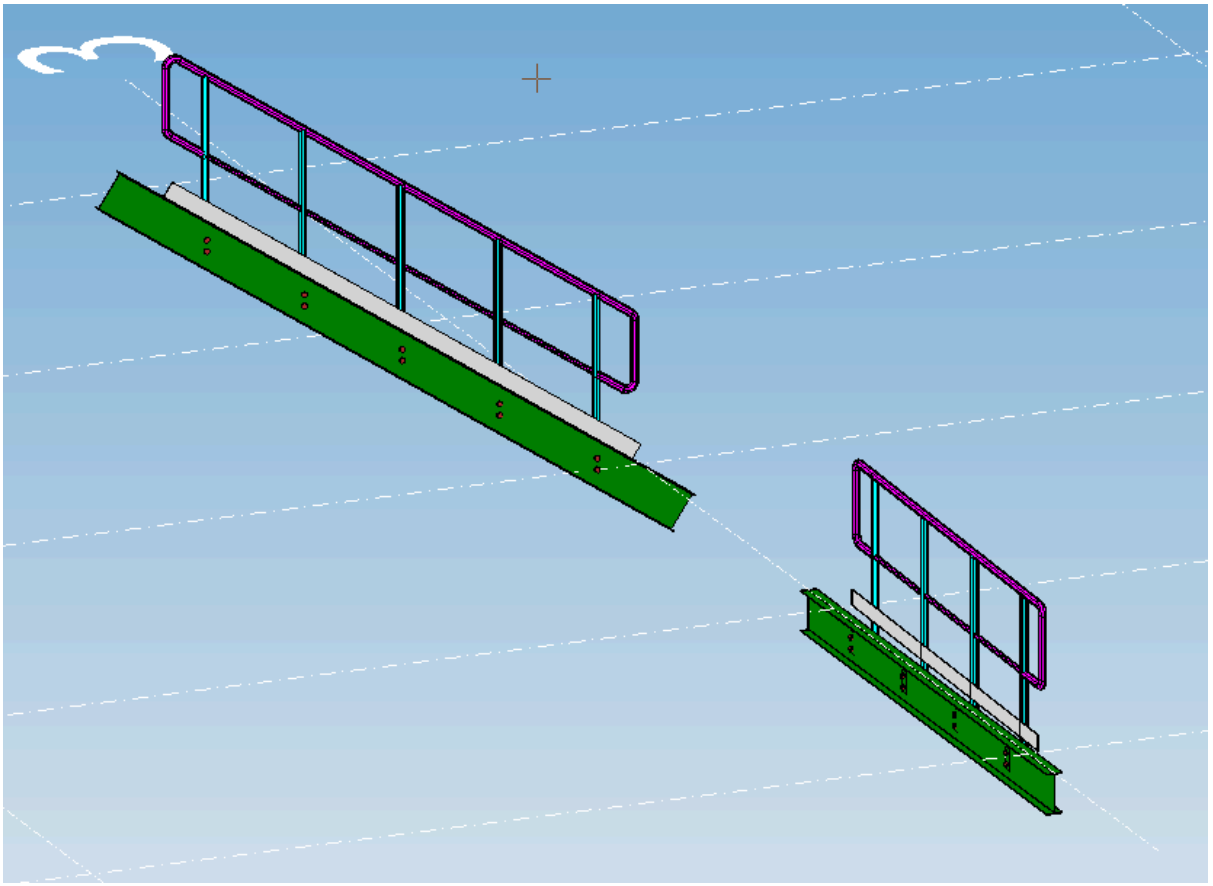
Now click on the SEC2 macro and then select any railing amount:



Lastly, click on the SEC macro to include the footer:



With the same step by step, on an inclined stair beam:



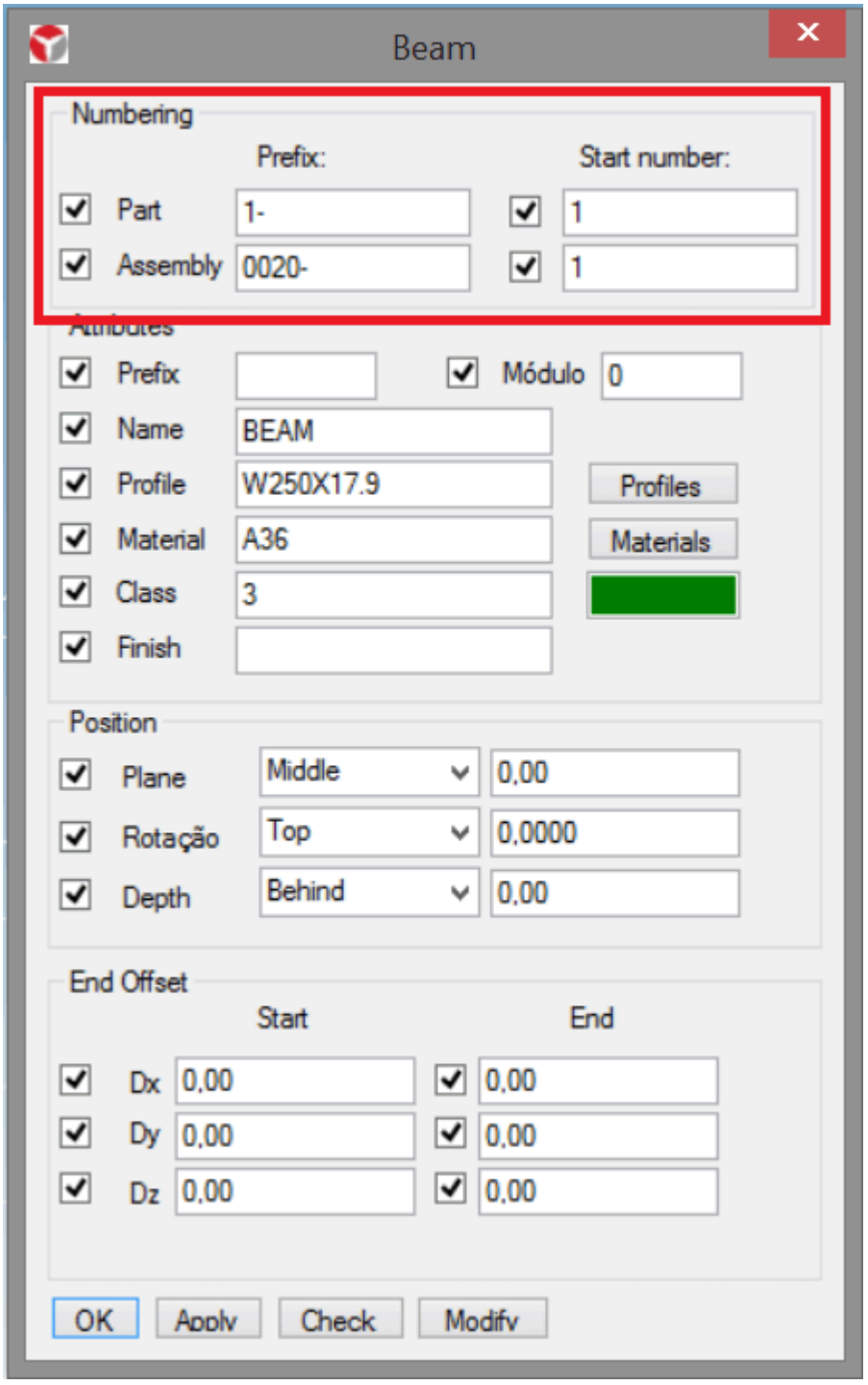
Numbering of parts and assemblies

Metal structure detailing needs to identify each part with a brand. Numbering is the process of giving marks to pieces.

Watch the video on how part numbering works

Equal pieces will have the same brand, to facilitate manufacturing, and obviously, different pieces can never have the same brand.
The user defines the numbering parameters, as we will see later, and the program will take care not to create the same marks for different parts.

The first and most important parameter for organizing numbering is the numbering series. Each series has a prefix and a starting number. For example, if the prefix is “PL-” and the starting number is 1, the pieces will be numbered PL-1, PL-2, PL-3, etc…
You can easily modify the numbering series of a part using the properties windows:



Numbering criteria

Parts with the same brand have the same:

- Typology (beam, column, plate, polybeam)
- Gauge (same thickness for sheets)
- Material
- If the part already has a mark from a previous numbering process, 3 different situations can occur:

    - One piece has a brand and the other does not: the pieces will be compared and if they are the same they will receive the existing brand of the first piece
    - The two pieces already have the same brand: The pieces will be compared, if they are the same they will remain with the brands. If they are not, one of the pieces will receive a new brand
    - The two pieces already have different brands: They will not be compared and their previous brands will be maintained.
- Phase (optional)
- Name (optional)
- Finishing or “Finish” (optional)
- Class (optional)
- Geometry
- Screws

What happens when there is a conflict between brands?

When the user modifies previously marked parts, or when importing new Tekla parts that already contain marks, the numbering process may find different parts with the same mark.
When a brand conflict is detected, the program will choose which pieces will maintain the previous brand and the others will receive new brands.

How are new brands created?

Whenever the numbering process needs to assign a new tag, it searches for an available number using the following criteria (you can configure this criteria):

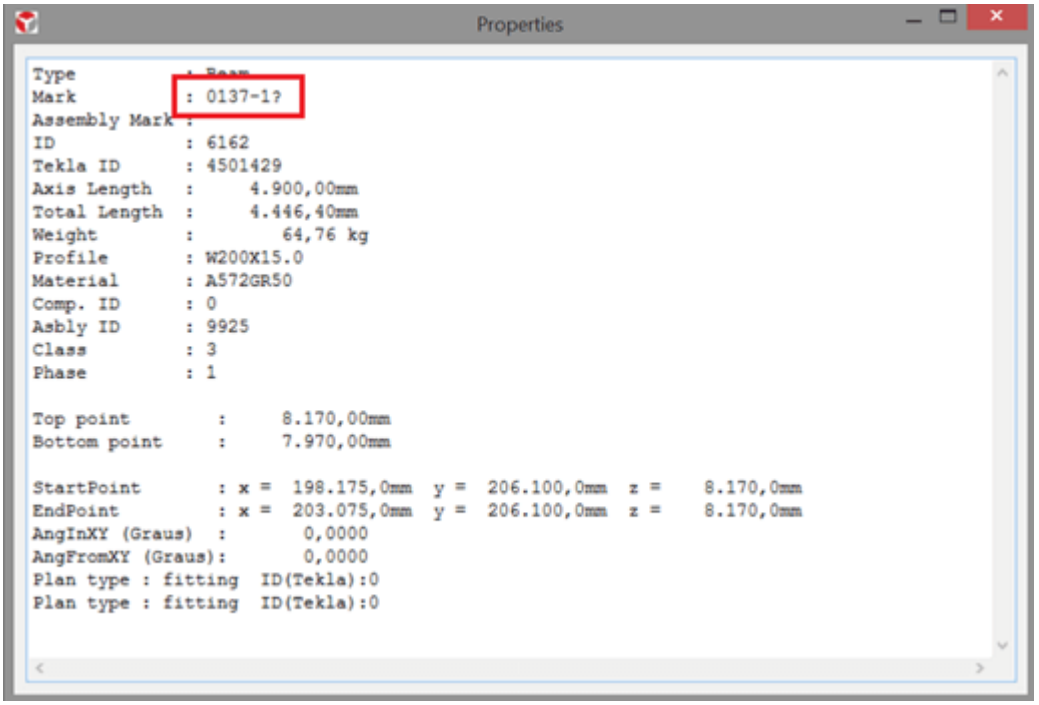
- Search for the first available number. In this case, previously used part numbers and now available will be used again.
- Find the largest number ever used in the series and add 1.

When an overlap of numbers occurs

Imagine, for example, a series for beams with the prefix “V” and starting with 101 for the first floor, 201 for the second, and so on.
If the first floor has more than 100 beams, the program will try to use V201, V202, and so on. It turns out that these marks may already be in use on the second floor beams.
If the program detects that a series has an insufficient range of numbers, it will warn the user to change the numbering series.

What happens when the user changes an already marked part?

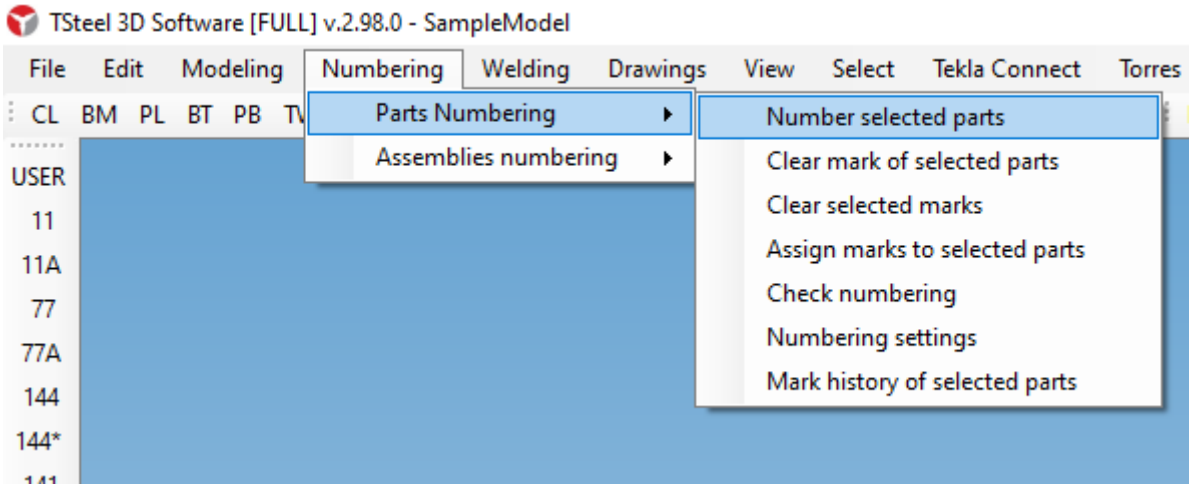
After any modification to already marked parts, the program will mark this part as changed and require a marking check. If you request an “Inquiry” (properties) of this part, you will see a question mark next to the brand. This question mark indicates that the marking needs to be verified.



Changes that require a markup check are:

- Some Boolean part is moved, changed, added, or deleted;
- Some cutting or adjustment plane is modified;
- Modification in name, material, phase, class, finish (Finish);
- Change in the numbering series (prefix or initial number);
- Some bolt group is changed, moved, added, or deleted;
- A Join or Split command was made with the part;
- The start and end points (Start/End points) have been modified;
- Any contour point has been modified, added or deleted.

Numbering menu

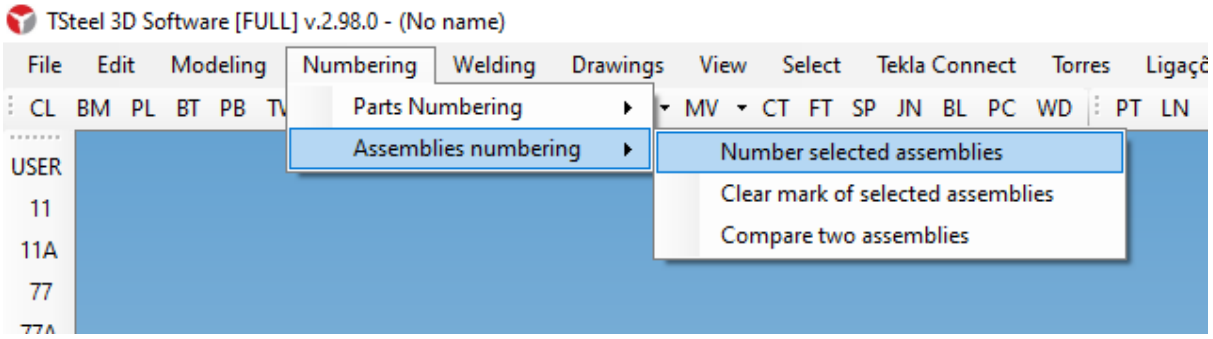


Number selected parts: The numbering process will identify which pieces are the same. The numbering criteria will be:

- If there are no old brands, the entire group will receive a new brand;
- If there are already marked pieces, they will remain with the original marks (if possible) and the pieces not yet numbered will receive a new mark.

Clear mark of selected parts: The selected pieces will have their marks deleted, they will remain as pieces that have not yet received marks.
Clear selected marks: All pieces that contain the selected brands will have their brands excluded.
Assign marks to selected parts: Allows the user to manually define the marking of one or more parts. Before assigning the mark, the program will check whether it is possible to assign the new mark while maintaining the integrity of the mark.
Check numbering: Allows the user to compare parts and find out if the parts are the same or not. This process does not alter the pieces, it only serves as a consultation.
Numbering settings: Allows the user to configure the part numbering criteria.
Mark history of selected parts: Helps monitor changes in parts brands

Set numbering



The numbering of the sets follows the same philosophy as the numbering of parts. To number sets, all pieces must be numbered.

Designs

You need to be in a saved model (already has the directory structure created) to start creating drawings. Drawings are saved in the “Drawings” folder of your model and have the extension “.tsdwg”.
To open or create new drawings, use the “Drawings” menu.

Watch a video about general drawing concepts

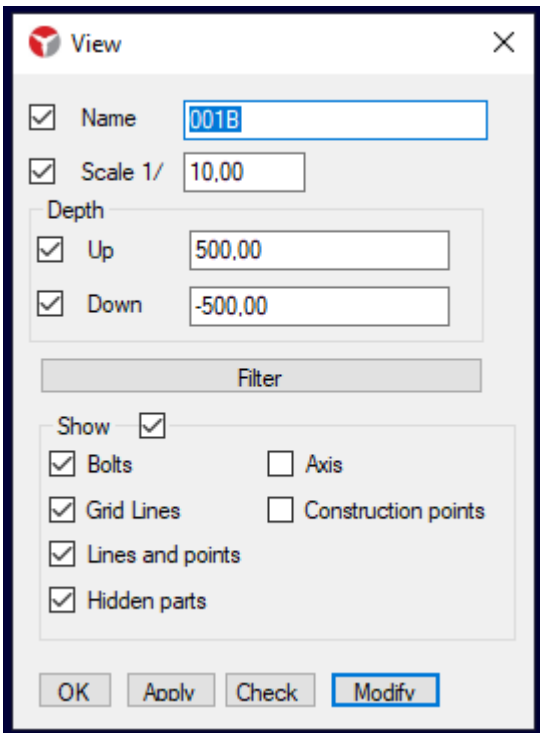
Assumptions of 2D drawings

- The drawings are on a 1:1 scale. Views, details and sections will be drawn at the chosen scale.
- As the drawing is at a 1:1 scale, the texts always have their final size in mm, regardless of the scale used in the view or detail.
- A drawing is made up of one or plan views of the model, separated into VIEWS (or frames). Each VIEW has its own depth configuration, part filter, drawing scale, etc…
- The model's flattened drawings are updated every time the drawing is opened. This means that drawings are always synchronized with changes to the model.

Settings of a VIEW in the drawing

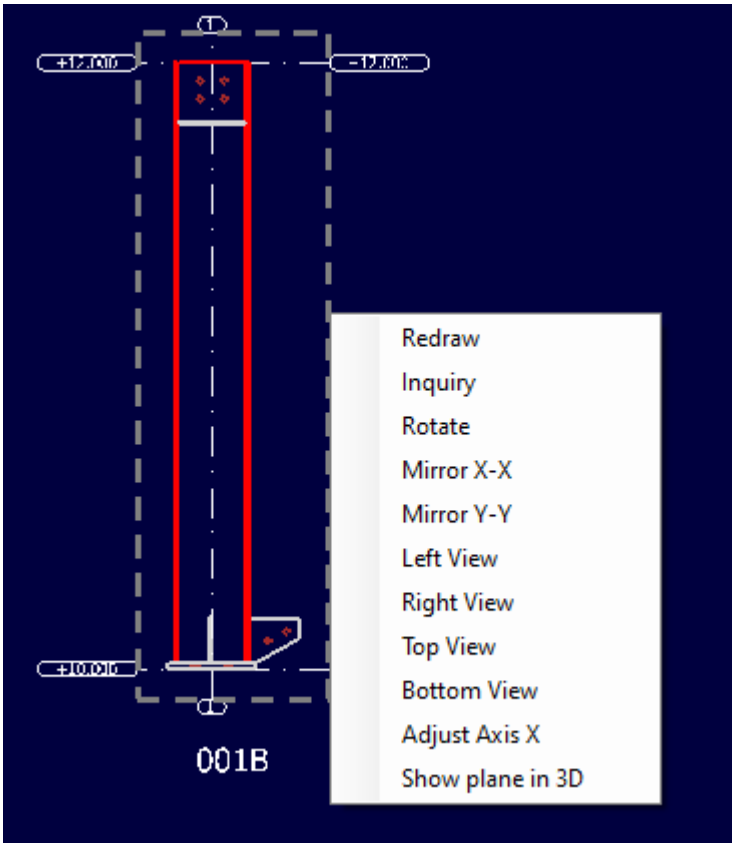
Each VIEW is marked by a dashed line box. This line is not exported to the final DWG file, it only serves as an auxiliary drawing entity. Through the board you can:

- Right-click and access a drop-down menu
- Click and drag the corners of the frame to change its size and drawing area
- Move the frame by dragging with the mouse
- Double click to open the Vista configuration window



In the window it is possible to configure the name of the view (which appears in the drawing), the scale, the depths (same concept as the flat modeling views), the part filter and a configuration of what should or should not be displayed in the frame.

The drop-down menu (right-click on the board) has:



- **Redraw:** Redesigns the view (updates model changes)

- **Inquiry:** View properties
- **Rotate:** Rotate the frame 90 degrees
- **Mirror:** Inverts the frame relative to the X or Y axis
- **Show** Plane in 3D: Creates a representation of the view plane in 3D

Types of Frames

There are the following ways to create a new board:

- **Model view:** Created from a model plan. It can be used to create a plan view of a platform, the elevation of a shaft, etc…
- **Isolated part:** Once a part is chosen, TSteel 3D creates a frame with the main plane of the part and draws it
- **Set:** Once an assembly is chosen, TSteel 3D creates a table with the drawing of the main view of the assembly. The choice of the main design plan for the set takes into account the main part of the set.
- **Cut:** Created from an existing frame, creates a view with a section
- **Detail:** Reproduces part of an existing frame, used to show larger-scale details.

How to include cuts

TSteel 3D creates cuts automatically, the user simply needs to enter the cutting line and its direction.
Each cut will be a new view, within a new frame, and therefore may have its own scale settings, filters, depths, etc...

How to add details

TSteel 3D creates the details automatically, the user simply needs to inform which area of the drawing they want. Just like the cuts, the detail will be in its own frame and can have its personalized configuration.

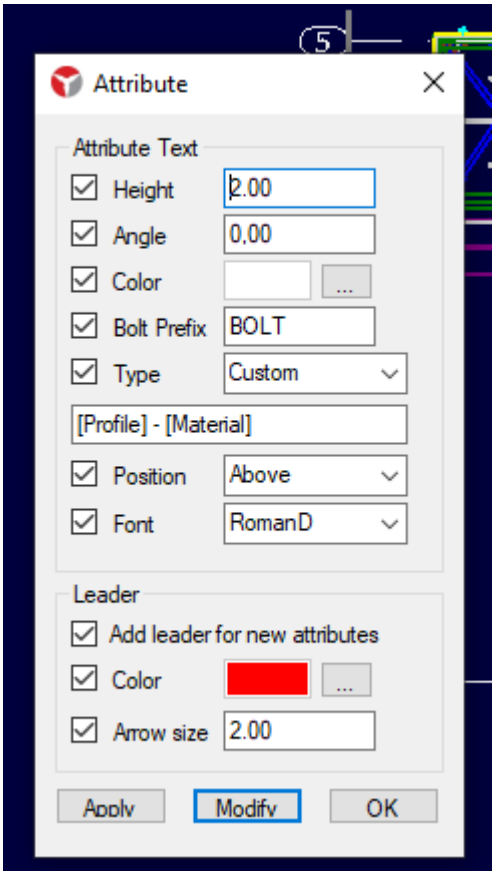
Watch video with cuts and details in the drawings

What are attributes

These are the properties of the parts, such as gauge, material, part brand or assembly brand. Attributes placed in the drawing are updated as the model changes. If a part changes its marking, the 2D drawing will be updated automatically.

Watch video on how to insert and configure attributes

To create custom attributes, use the “Custom” type and below a line of text where we can configure how we want the attribute text. Always use the attribute name (same as in the checkbox above) in square brackets.



How to create different types of quotas

TSteel 3D creates linear dimensions (Horizontal, Vertical or aligned) and angle dimensions. See the video below how to create quotas:

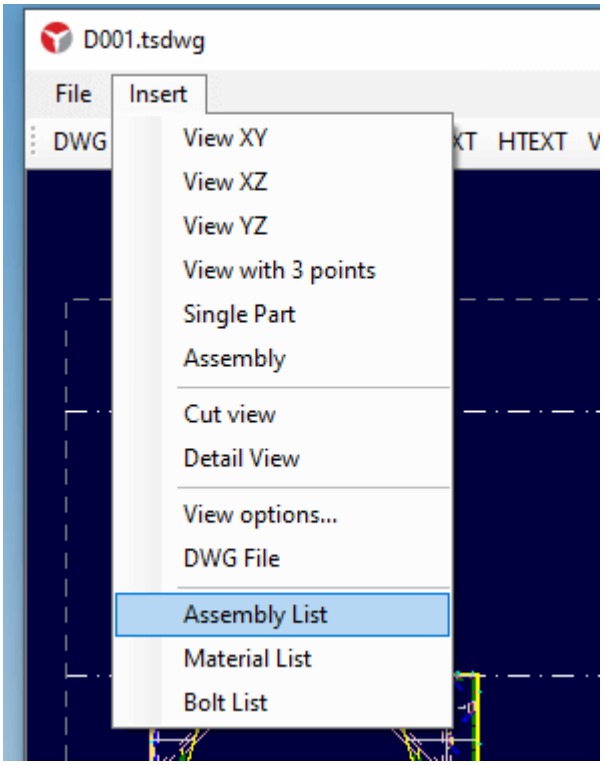
Watch video about dimensions in the drawing.

Material list in drawing

There are currently 3 types of lists available to insert into drawings:

- Set list (used in detail drawings)
- Bill of Materials (summary of materials, used mainly in executive projects)
- Screw List

To insert the material list into the drawing, use the Insert>Assembly List menu.



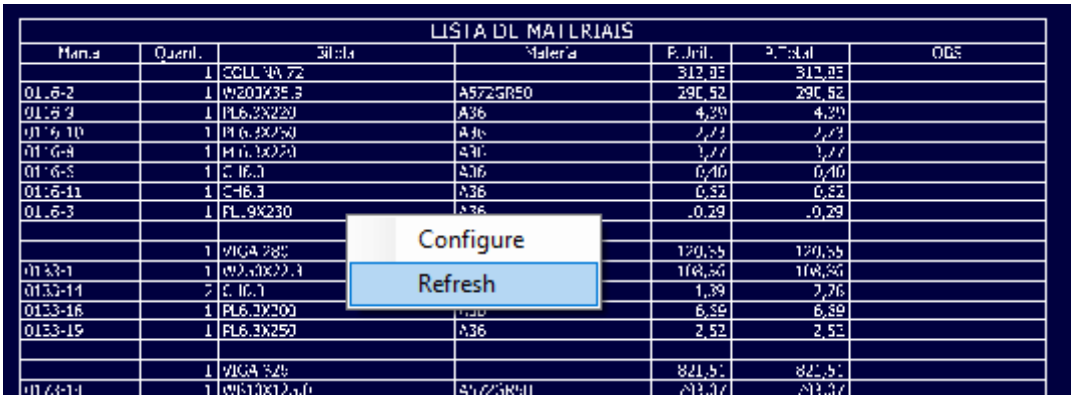
Set List

TSteel 3D will search which Assemblies are in the drawing and generate the bill of material. See the example below:

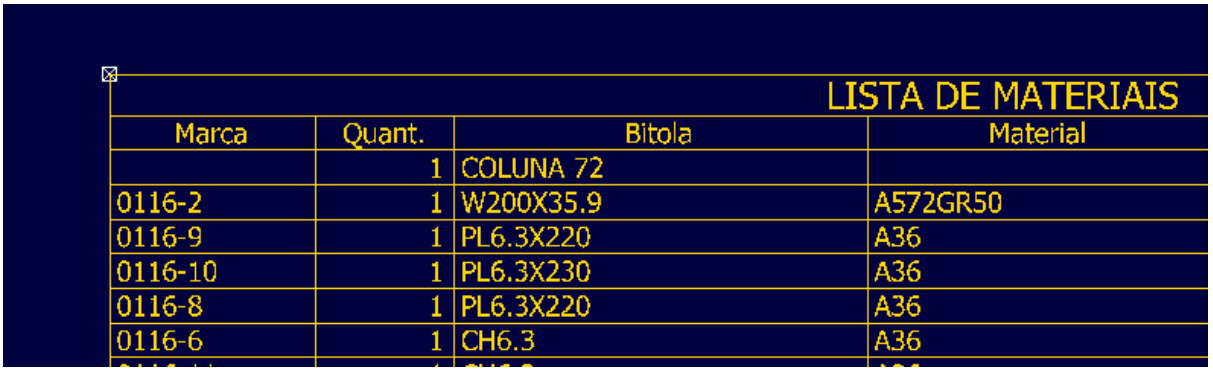
LISTA DE MATERIAIS

Marca	Quant.	Título	Material	P. Unit.	P. Total	OBS
	1	[illegible]		312,05	312,05	
01.6-2	1	W200X35.9	A572GR50	290,52	290,52	
01.6-9	1	PL6.3X220	A36	4,29	4,29	
01.6-10	1	PL6.3X250	A36	2,73	2,73	
01.6-8	1	PL6.3X220	A36	3,77	3,77	
01.6-5	1	C 16.3	A36	0,40	0,40	
01.6-11	1	C16.3	A36	0,52	0,52	
01.6-3	1	PL.9X230	A36	.0,29	.0,29	
	1	VIGA 28C		120,55	120,55	
0133-1	1	W530X92.0	A572GR50	108,50	108,50	
0133-14	2	C 16.3	A36	1,39	2,76	
0133-16	1	PL6.3X200	A36	6,59	6,59	
0133-19	1	PL6.3X250	A36	2,52	2,52	
	1	VIGA 32A		821,51	821,51	
0173-14	1	W610X101.0	A572GR50	703,37	703,37	
0173-12	2	C 12.5	A36	0,17	12,34	
0173-13	2	C 12.5	A36	6,08	12,16	
0173-11	2	C16.3	A36	1,02	2,05	
0173-10	3	C16.3	A36	0,94	1,89	
	1	[illegible]		305,26	305,26	
01.8-1	1	W200X35.9	A572GR50	290,52	290,52	
01.8-5	1	C 16.3	A36	0,55	0,55	
01.8-6	1	C16.3	A36	0,39	0,39	
01.8-10	1	PL8X340	A36	7,20	7,20	
01.8-7	1	PL6.3X220	A36	4,86	4,86	
01.8-9	1	C16	A36	0,65	0,65	
01.8-12	1	PL8X200	A36	2,89	2,89	
01.8-8	1	PL6.3X200	A36	3,37	3,37	
01.8-1	1	W410X38.8	A572GR50	10,27	10,27	
01.8-11	1	PL8X235	A36	3,28	3,28	
01.8-2	1	W410X38.8	A572GR50	8,29	8,29	
01.8-4	1	PL.10X230	A36	12,01	12,01	
		Peso Total			1.599,95	

The list is recalculated and updated whenever the drawing is opened or when you request it through the Refresh menu:



List settings, including titles, formatting and column order, are available in the same right-click menu, under the “Configure” option.
To move the material list in your drawing, click anywhere in the list to select, then click and drag the insertion point in the upper left corner.

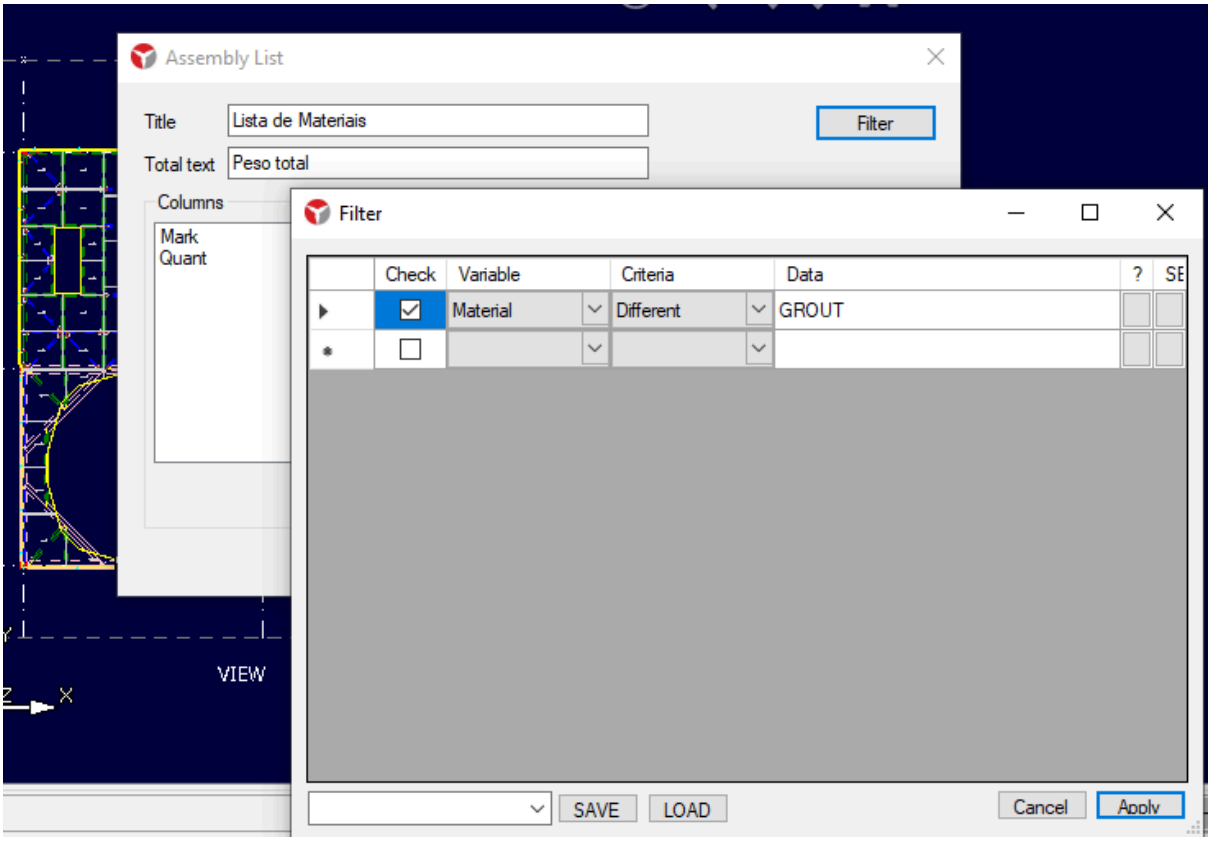


LISTA DE MATERIAIS

Marca	Quant.	Bitola	Material
	1	COLUNA 72	
0116-2	1	W200X35.9	A572GR50
0116-9	1	PL6.3X220	A36
0116-10	1	PL6.3X230	A36
0116-8	1	PL6.3X220	A36
0116-6	1	CH6.3	A36

List of materials

The list of materials considers the entire model. You can adjust what needs to be shown, right-click on the list, select “Configure” and use the filter.



Example of list of materials:

Lista de Materiais					
Bitola	Dimensão	Material	P.Unit.	P.Total	OBS
W150X13.0	234,32	A572GR50	12,36	2.895,77	
W150X22.5	34,97	A572GR50	22,77	796,09	
W200X15.0	40,81	A572GR50	14,56	594,40	
W200X19.3	31,37	A572GR50	19,70	618,03	
W200X26.6	14,28	A572GR50	26,85	383,42	
W200X35.9	66,73	A572GR50	35,87	2.393,83	
W250X22.3	27,88	A572GR50	22,69	632,51	
W250X25.3	5,18	A572GR50	25,59	132,54	
W310X32.7	5,97	A572GR50	33,05	197,23	
W310X38.7	16,70	A572GR50	39,01	651,72	
W310X52.0	5,97	A572GR50	51,95	310,32	
W410X38.8	18,55	A572GR50	39,49	732,53	
W410X46.1	11,50	A572GR50	46,47	534,47	
W460X52.0	17,95	A572GR50	51,33	921,49	
W530X101.0	23,08	A572GR50	100,34	2.315,35	
W530X82.0	5,79	A572GR50	81,03	469,49	
W610X125.0	12,79	A572GR50	123,98	1.586,15	
Peso total				16.165,33	

Screw List

The list of screws uses the entire model. You can also apply filters to include just one phase, for example.

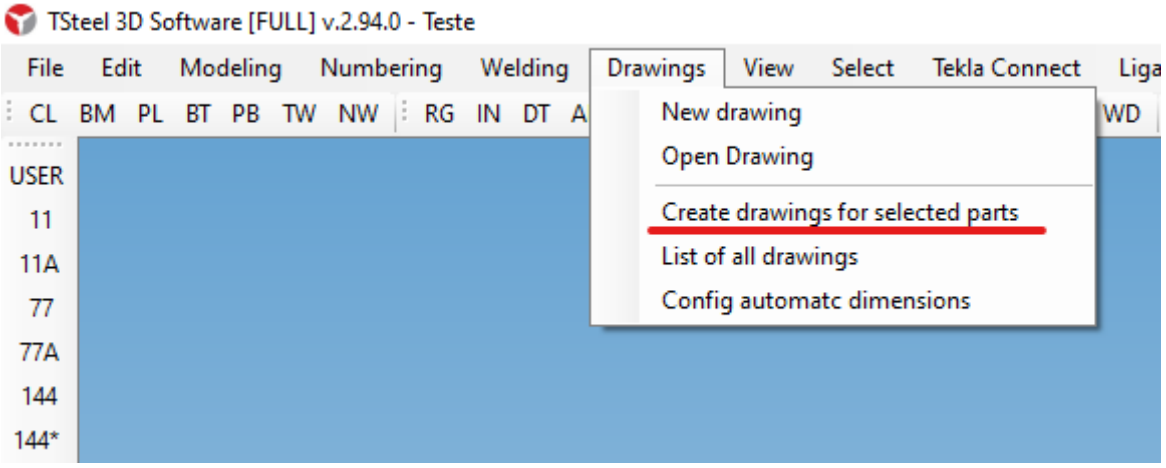
Lista de Parafusos			
Bitola	Material	Quant.	OBS
ARR 1	STANDARD	9	
ARR 1/2	STANDARD	160	
ARR 3/4	STANDARD	410	
ARR 5/8	STANDARD	1588	
PARAF 1/2X1.1/4	A325	160	
PARAF 1X1.3/4	A325	9	
PARAF 3/4X1.1/2	A325	32	
PARAF 3/4X1.1/4	A325	15	
PARAF 3/4X1.3/4	A325	64	
PARAF 3/4X2	A325	279	
PARAF 3/4X2.1/4	A325	20	
PARAF 5/8X1.1/2	A325	1120	
PARAF 5/8X1.1/4	A325	18	
PARAF 5/8X1.3/4	A325	446	
PARAF 5/8X2	A325	4	
PORCA 1	HEAVY	9	
PORCA 1/2	HEAVY	160	
PORCA 3/4	HEAVY	410	
PORCA 5/8	HEAVY	1588	

Automatic part drawings

TSteel automatically creates part drawings (also called manufacturing sketches).

Automatic drawings, created using an algorithm that solves most cases, are subject to the need for manual adjustments. In any case, we consider that all automatic drawings need verification.

To do this, select all the parts you want (use filters if necessary), and click**Drawings > Create drawings** for selected parts

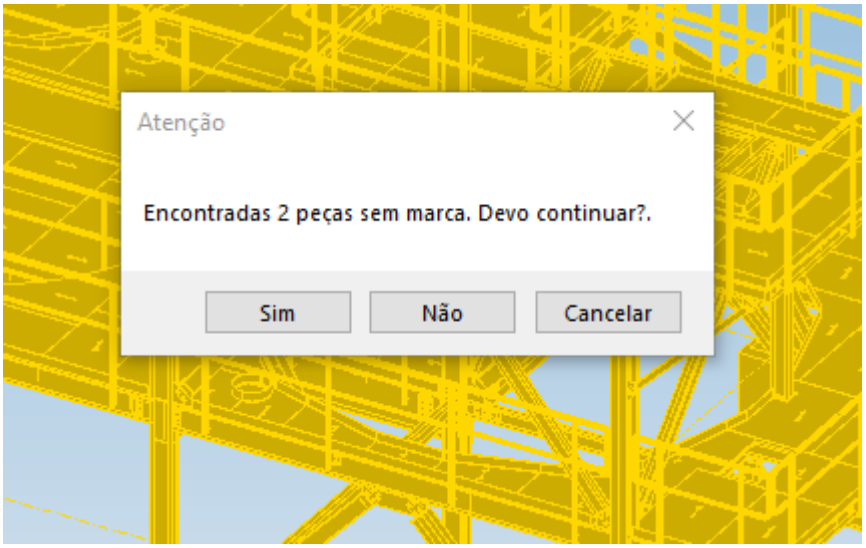


Given the command, TSteel will perform the following tasks:

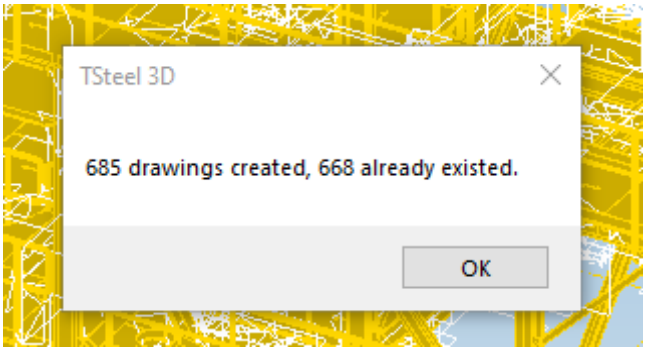
- Identify all brands of selected parts

- For each part brand, create a drawing named after the part brand.**If a drawing with this name already exists, it will be overwritten**.
- Unmarked parts will not have their drawings created, and you will receive a warning of how many unmarked parts were found.

Notice of unmarked parts among those selected to extract drawing:



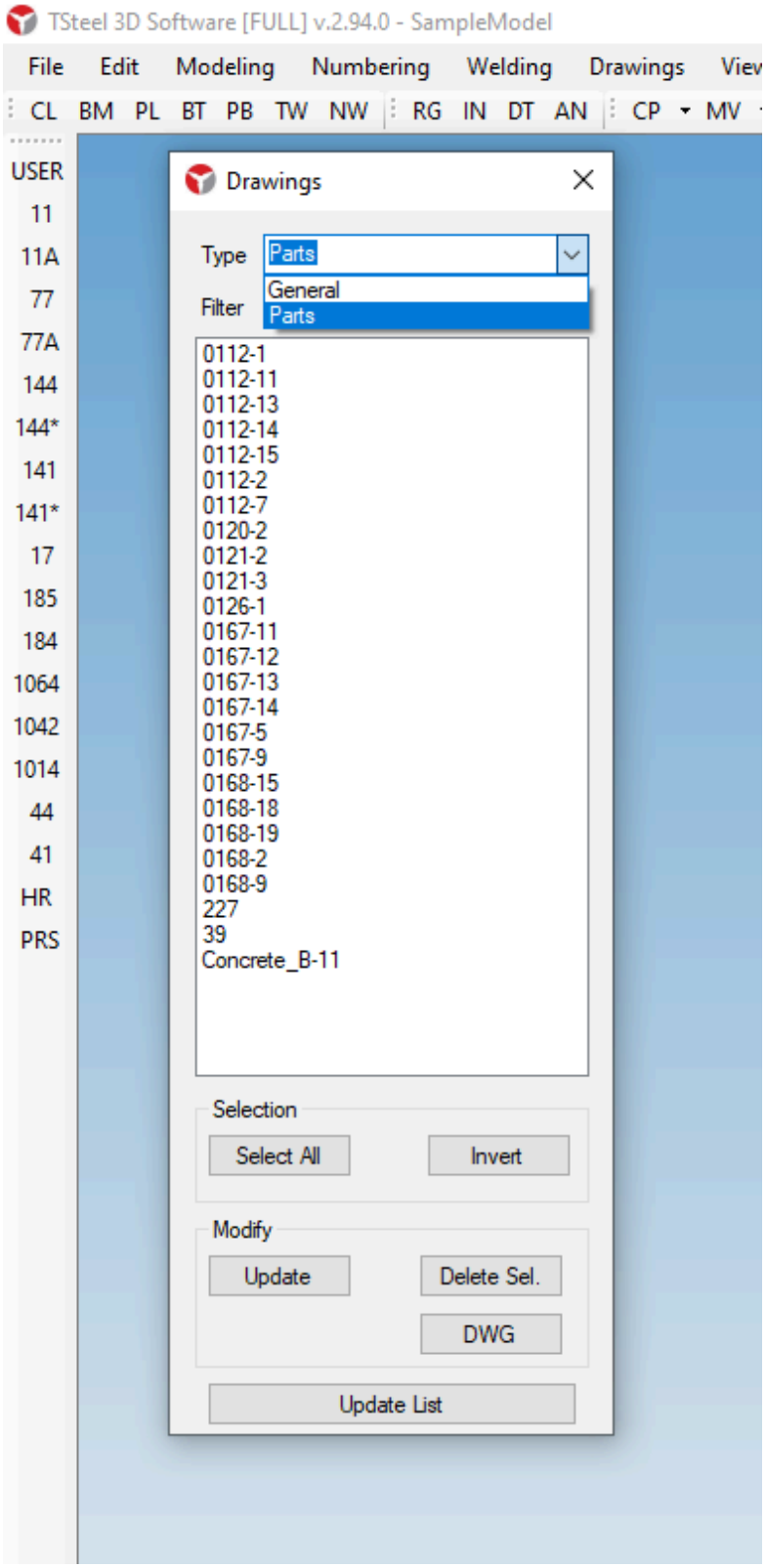
Depending on the number of designs to be created, the process may take some time. At the end, you receive a message about how many drawings were created:



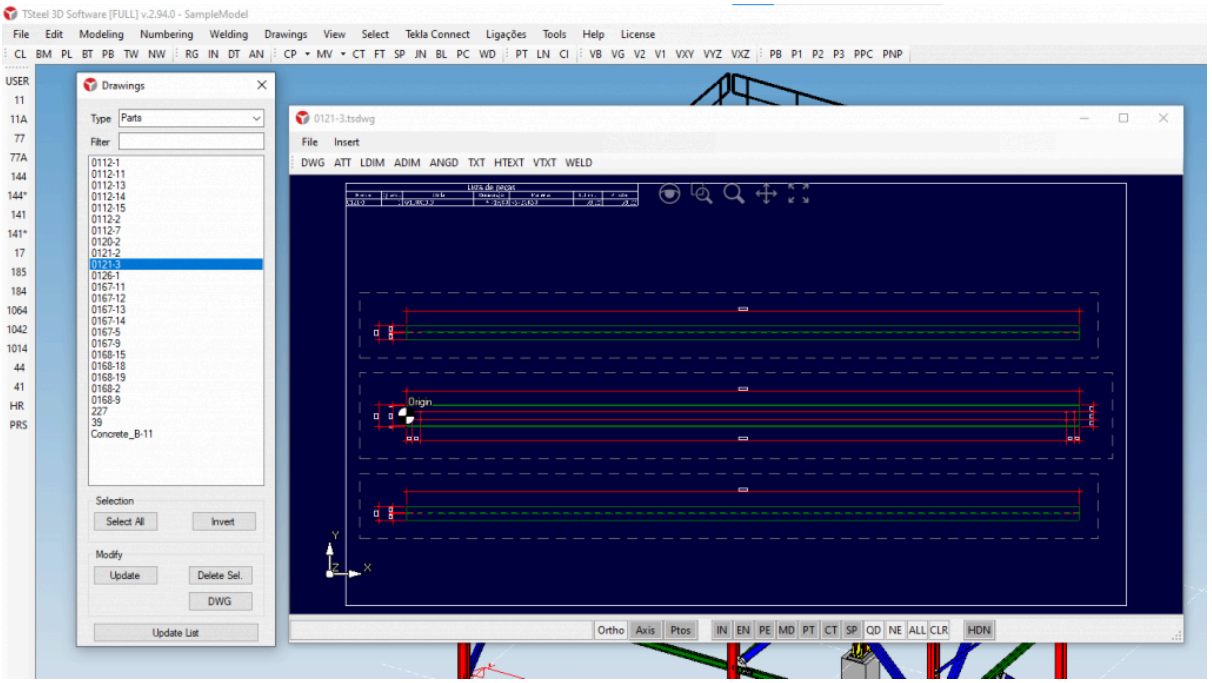
Note that existing designs are not replaced. If you want to create drawings from scratch, you must first delete the existing drawings and then have them created again.
Automatic part drawings are created in a special directory**\Drawings\Parts**

How to inspect created drawings

In the Drawings menu, there is the All Drawings command, which opens a window with a list of available drawings. Choose whether you want to see part drawings or general drawings.



You can use the filter to narrow down the list of designs shown and help you find what you're looking for. When you click on one of the items in the list, it opens the drawing.



As you select new designs,**the previous drawing is automatically saved** and the new drawing is shown.

The stamp of automatic drawings

Currently the stamp is generated using the following format sizes: A4 (flat or upright, A3 lying down. If the design does not fit into any of these formats, TSteel will lengthen the format to fit the design.
If you need to redo the stamp after adjusting dimensions and views, do the following:

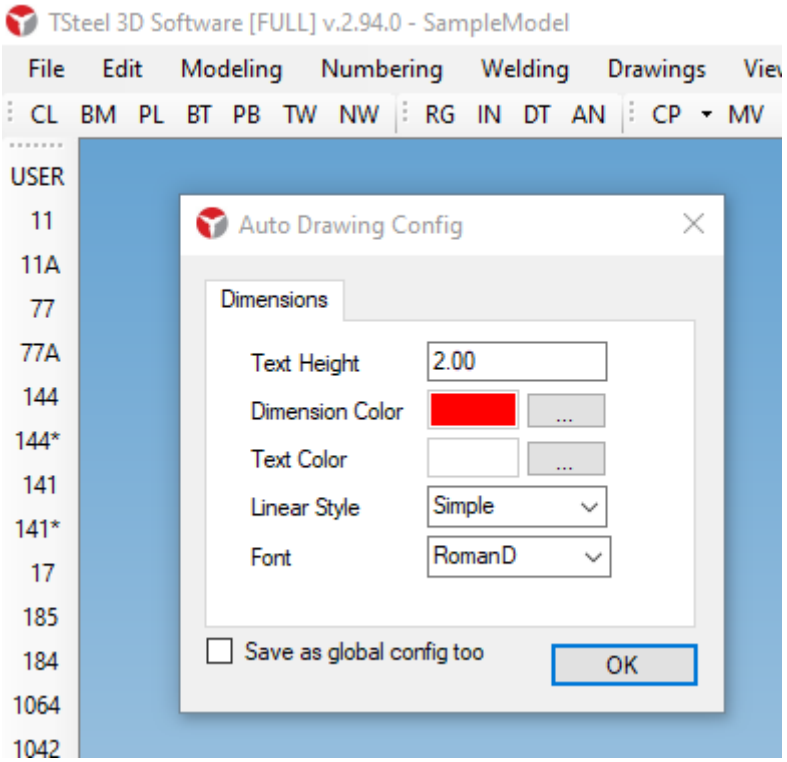
1. Select and delete the stamp outline and material list
2. Wow menu**Insert>Automatic Part** Format and TSteel will insert a new stamp and list.

Scale of drawings

Currently the drawings are inserted at a 1:10 scale. Changing the scale must be manual.

Configuring automatic quotas

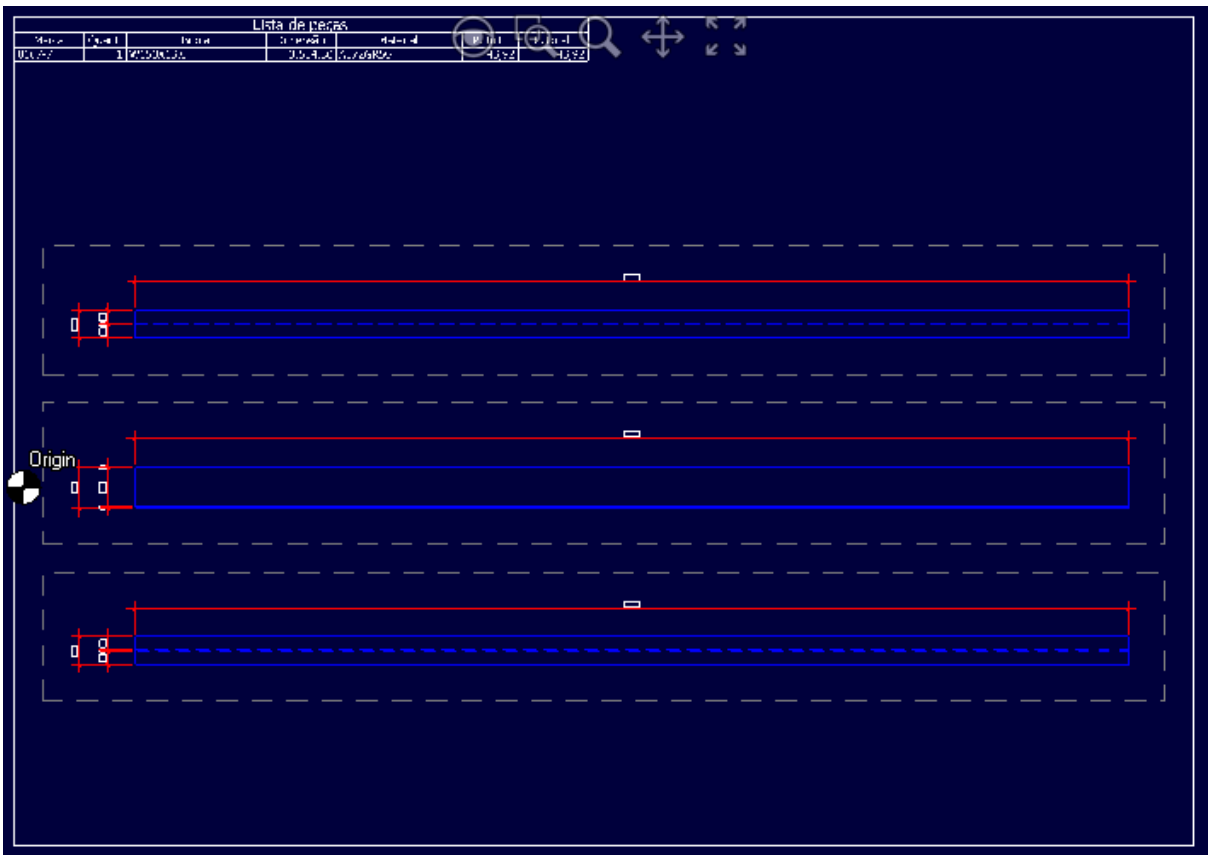
The configuration of automatic dimensions is in the Drawings>Config automatic dimensions menu, which opens the following window:



When you change the settings and click OK, TSteel saves a file in the model with the settings (AutoDrawing.config). If you select the option to save as global configuration, TSteel saves this file also in the c:\TSteel3D directory and it will be used as the global default.

Top and bottom view of beams in automatic sketches

For beams, automatic drawings create 3 views, see below:



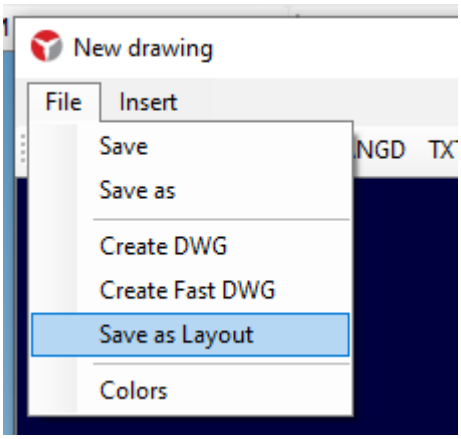
For the top view, there needs to be a depth adjustment to exclude the bottom table. Therefore, the drawing only shows holes and cutouts for the upper table. The same goes for the bottom view.

This criterion resolves the vast majority of cases, but may raise doubts when reading the drawing in special cases. For these special cases, manual adjustment will be necessary.

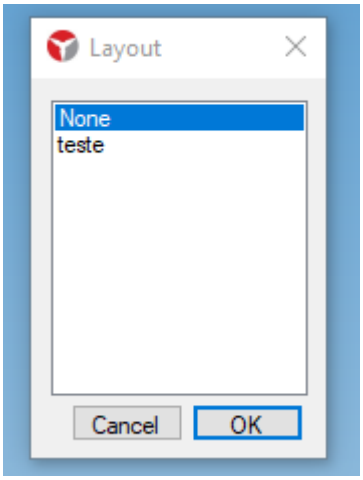
Configuring formats and stamps

To create a standard format and stamp for drawings, for example for A1, A4 drawings, etc…

1. Open a new empty drawing
2. Insert a DWG file with the format and stamp you want to use. Remember that not all entities in a DWG can be imported by TSTeel
3. Save the file as Layout. Save in the Drawings directory, otherwise the layout will not be found.



After creating a Layout, whenever you open a new design, TSteel will offer a list of available Layouts.



In the example above, only a layout named “test” was saved.

Watch a video with the creation of drawing layouts.

Plate DXF files

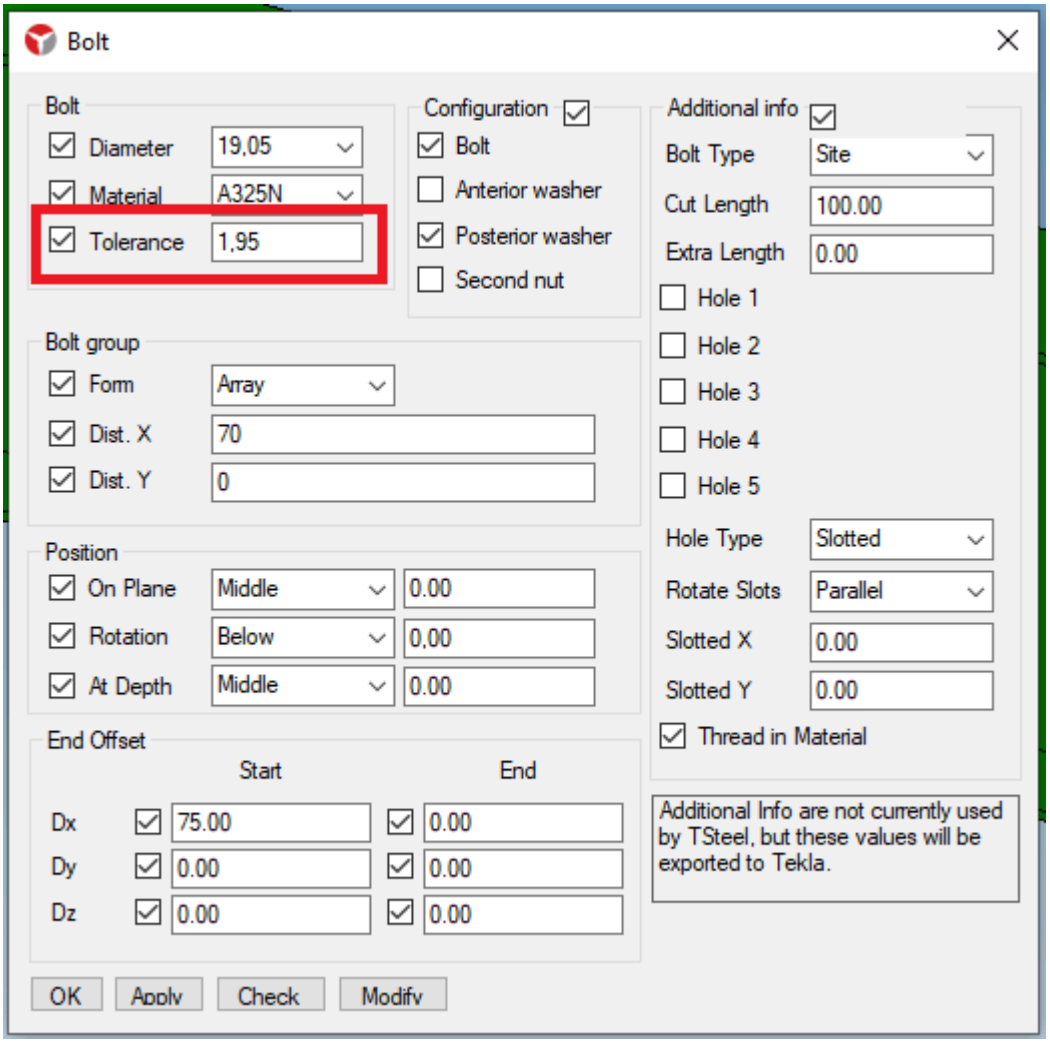
TSteel 3D creates DXF sheet metal files automatically.

What parts are considered sheets?

All parts created as ContourPlate and beams (not polybeams) with flat bar gauge.
It is up to the user to select the parts they want to create the DXF file.

How are the holes represented?

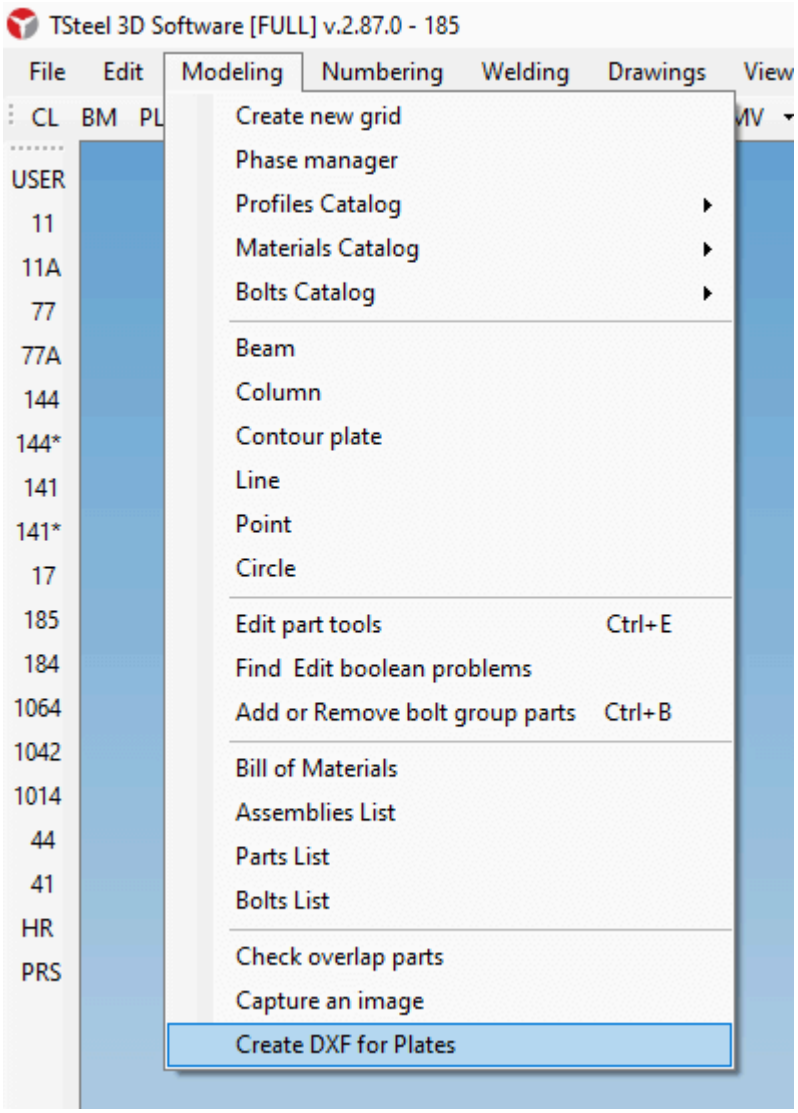
The diameter of the hole drawn will be the screw diameter + the registered tolerance.



For the case above, a 21.0mm hole will be drawn for the 3/4″ screw.

How to generate DXF files

For DXF files to be created, the parts must be marked. If, among the selected parts, any unmarked part is found, TSteel 3D will give a warning and stop the operation. Tags are necessary so that files can be identified. Each DXF file will be named after the brand of the plate.
DXF generation is in the Modeling menu:



Tsteel 3D will show a warning if it was unable to create the DXF of a sheet. All files created will be**inside the DXF directory**, inside the model's main directory.

Attention : Old DXF files will be overwritten. Create a selection and creation strategy in stages so that your files are organized.

Configuring the DWG of your drawings

From version 2.95 onwards, drawings began to be exported to DWG with different layers.

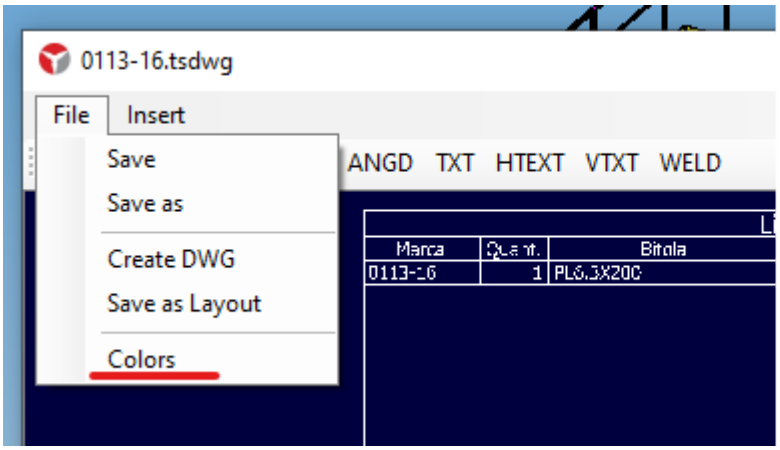
What layers are created?

For the parts (structure), each class will have its own layer. For the other parts of the drawing, the layer division will be as follows:

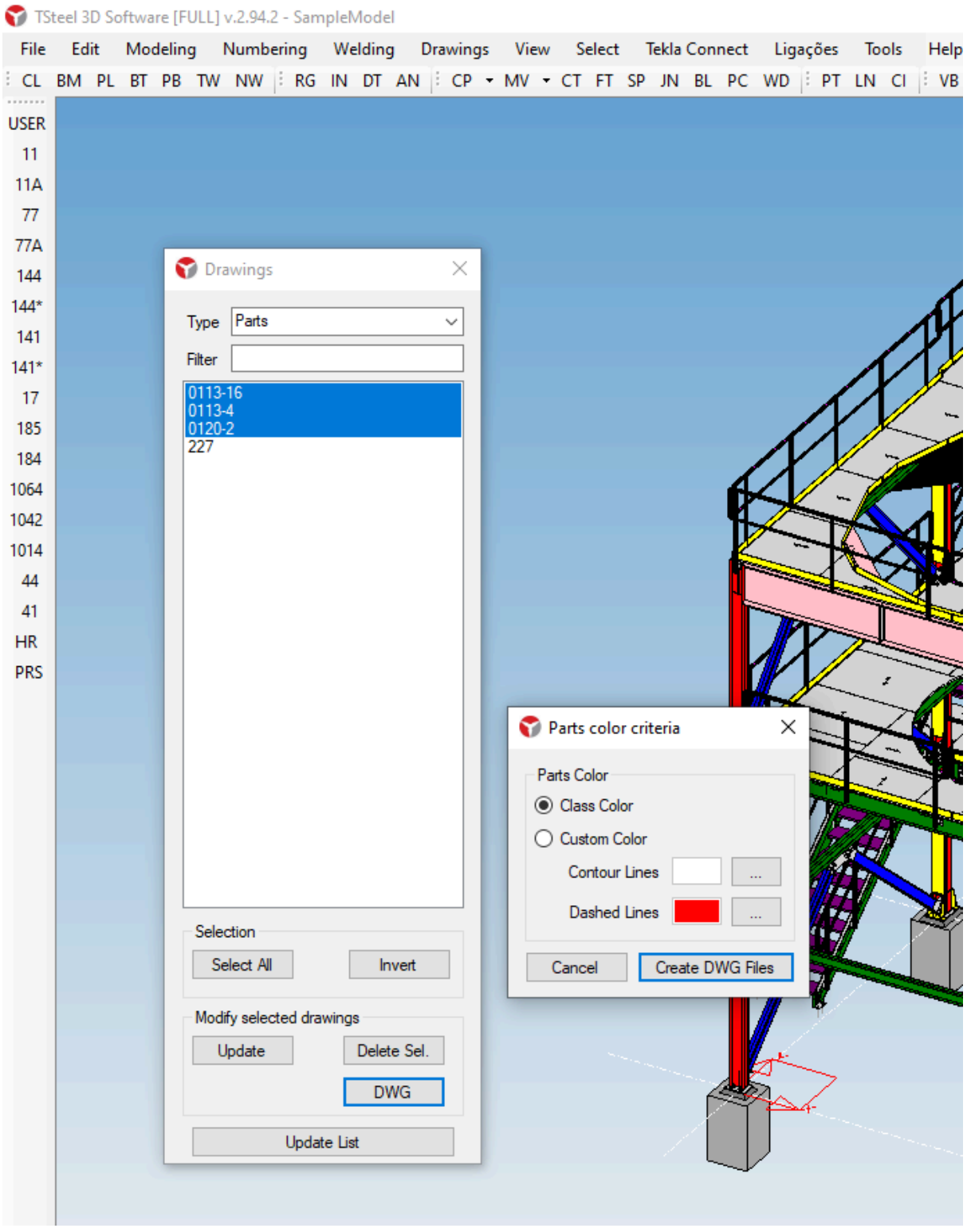
- Default: will have all entities that do not fit into the other categories
- Dimensions: All drawing dimensions
- Text: Contains texts and attributes of the parts
- Welds: Weld symbologies
- Bolts: Contains the centerlines of the bolts. The representation of the hole will be along with the layer of the part that contains it.

Color criteria

The text and line colors will be the same as what you see in the drawing, regardless of the layer it is on. The colors of the parts depend on the color configuration of the drawing.



When you use the graphics window and request the creation of several DWGs at once, TSteel will ask you which color criteria you want.


TSteel 3D Software [FULL] v.2.94.2 - SampleModel
File Edit Modeling Numbering Welding Drawings View Select Tekla Connect Ligações Tools Help
CL BM PL BT PB TW NW RG IN DT AN CP MV CT FT SP JN BL PC WD PT LN CI VB
USER
11
11A
77
77A
144
144*
141
141*
17
185
184
1064
1042
1014
44
41
HR
PRS
Drawings
Type Parts
Filter
0113-16
0113-4
0120-2
227
Selection
Select All
Invert
Modify selected drawings
Update
Delete Sel.
DWG
Update List
Parts color criteria
Parts Color
Class Color
Custom Color
Contour Lines
Dashed Lines
Cancel
Create DWG Files

Export and import

DWG files

To configure how to export your drawings to DWG, see the drawings chapter. There you will find how to configure layers and colors.

Here we will talk about exporting/importing the model to a DWG file.

Your 3D model can be exported to DWG and the parts will be as “Meshes” in the DWG file. In other words, it is possible to see and use a 3D model within Autocad.

When importing, all elements can be used as a reference. It is very useful, for example, to import an architectural plan to use as a location reference for your model.

Imported DWG files are not incorporated into your model, meaning you need to import the reference DWG whenever you need it. If you need it to be part of your model, see below for how to embed a DWG into your model.

Problems generating DWG files

The DWG format is exclusive to Autodesk, which in turn has no interest in sharing with the market how to read and write in this format. The software market has created libraries capable of getting around this problem and can read and write in DWG format.
TSteel 3D uses one of these routine libraries to read and write DWG. There are cases, in more complicated DWG files, that our routines are not able to read correctly. In other cases, the routines don't even work due to the way other Autodesk applications have already been installed on your machine.

When the problem is with the file

To find out if the problem is in the DWG file, which contains some entity or configuration that cannot be read, we do the following:

1. Create a new DWG file, with just a few lines as if it were a GRID.
2. Import this simple dwg file.
3. If TSteel 3D was able to import, the problem is with the file and not with your computer's installations.
4. Start by exploding the blocks in your DWG to see if it works.
5. In a second step, copy only the parts that interest you to a new DWG file.
6. This process is trial and error, you have to be patient.

When the problem is not with the file, but with the computer

There are cases where the DWG import and export routines do not work. This depends on the settings and DLLs on your computer. To get around this problem, we provide an alternative program to make this link with DWG files, the**TsDWG**.

How to install TsDwg

Follow the following steps:

1. Download the ZIP file with the programin this link
2. Unzip the file**inside the C:\TSteel3D directory. After unzipping, there should be a directory C:\TSteeel3D\TsDWG**
3. Run the installer (Setup.EXE) inside the TsDWG created directory.
4. The program will install and place a shortcut on your home screen.

How to use TsDWG

When TSteel 3D needs to import or export a DWG file, and discovers that there is a TsDWG installed on the machine (through the existence of the TsDWG directory that we created), it will perform these operations using TsDWG.
For TsDWG to work when importing or exporting, it needs to be running. In other words, open the program before any DWG import or export.

IFC

Embed IFC or DWG files in the model

We saw previously that TSteel 3D**does not save imported elements as IFC or DWG in your models**. You are required to import the reference files each time you open your model.

If you want to save reference files with your models, do the following:

1. In a new model, import your IFC or DWG file;
2. Change the scale or move them as needed so they are in the right location;
3. Select the entities you will need (remember not to add more reference data than you need) and export with the “TSteel3D ref file” option (reference file in TSteel 3D format)
4. Open your model and import this reference file we just created. Now, these entities will be saved along with your model.

These entities, imported as a TSteel reference file, will be incorporated into your model and will be loaded along with the model.

Advantages of TSteel 3D Ref File

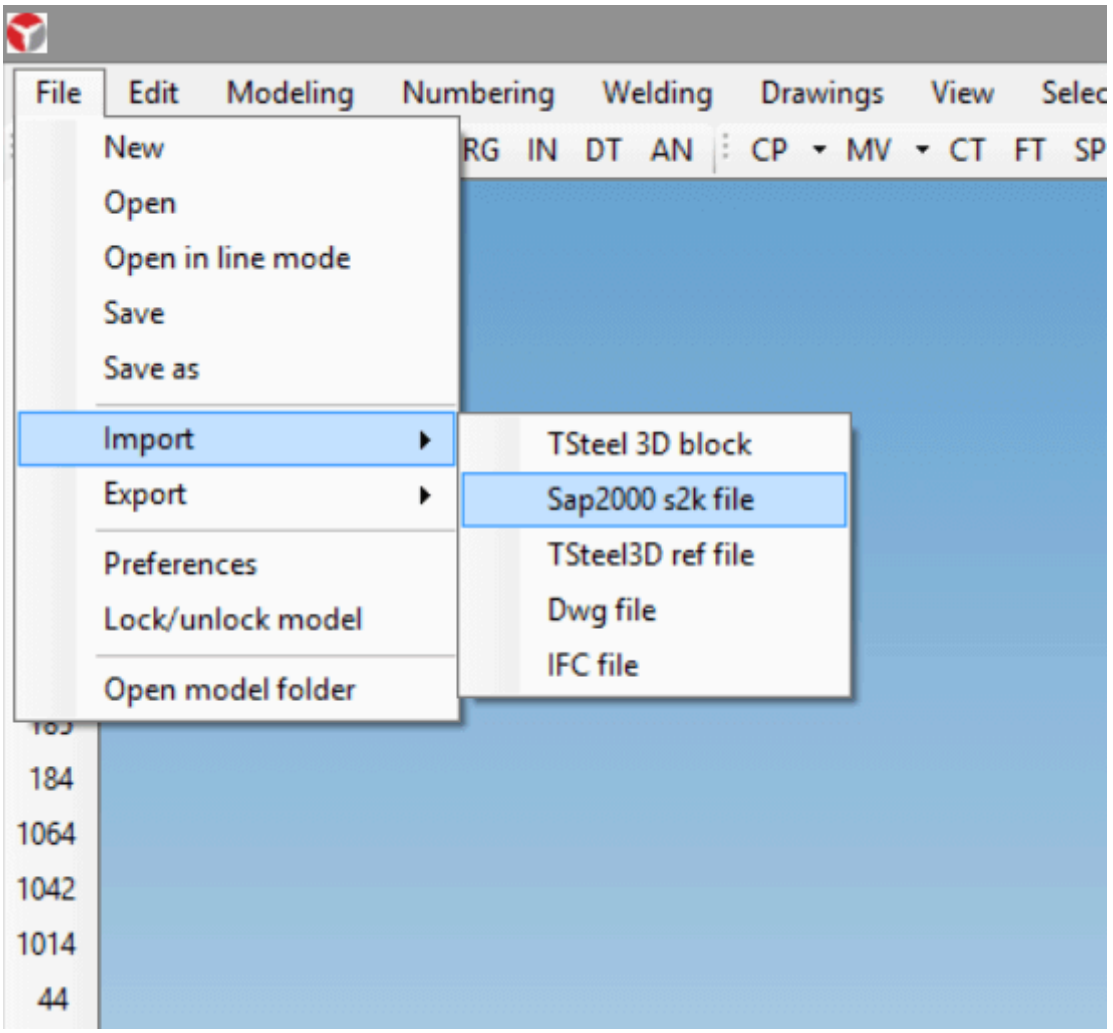
As we saw previously, the TSteel 3D reference file is a way of including external entities in your model, but that's not all.

The file format is optimized, taking up less disk space (and less memory when loaded) and being faster to render.

Keep in mind that saving large reference files along with the model can harm the software's performance.

Sap2000

SAP import is direct and fast, using the S2K interoperability file.



The S2K file is a text file created by SAP 20000 and has maintained compatibility over the years. Therefore, importing S2K files works with all available SAP versions.

Importing from SAP 2000, unlike DWG and IFC, creates native elements in TSteel 3D. If necessary, TSteel creates profiles and materials in catalogs during import, so that your model is consistent with SAP 2000.

Tekla

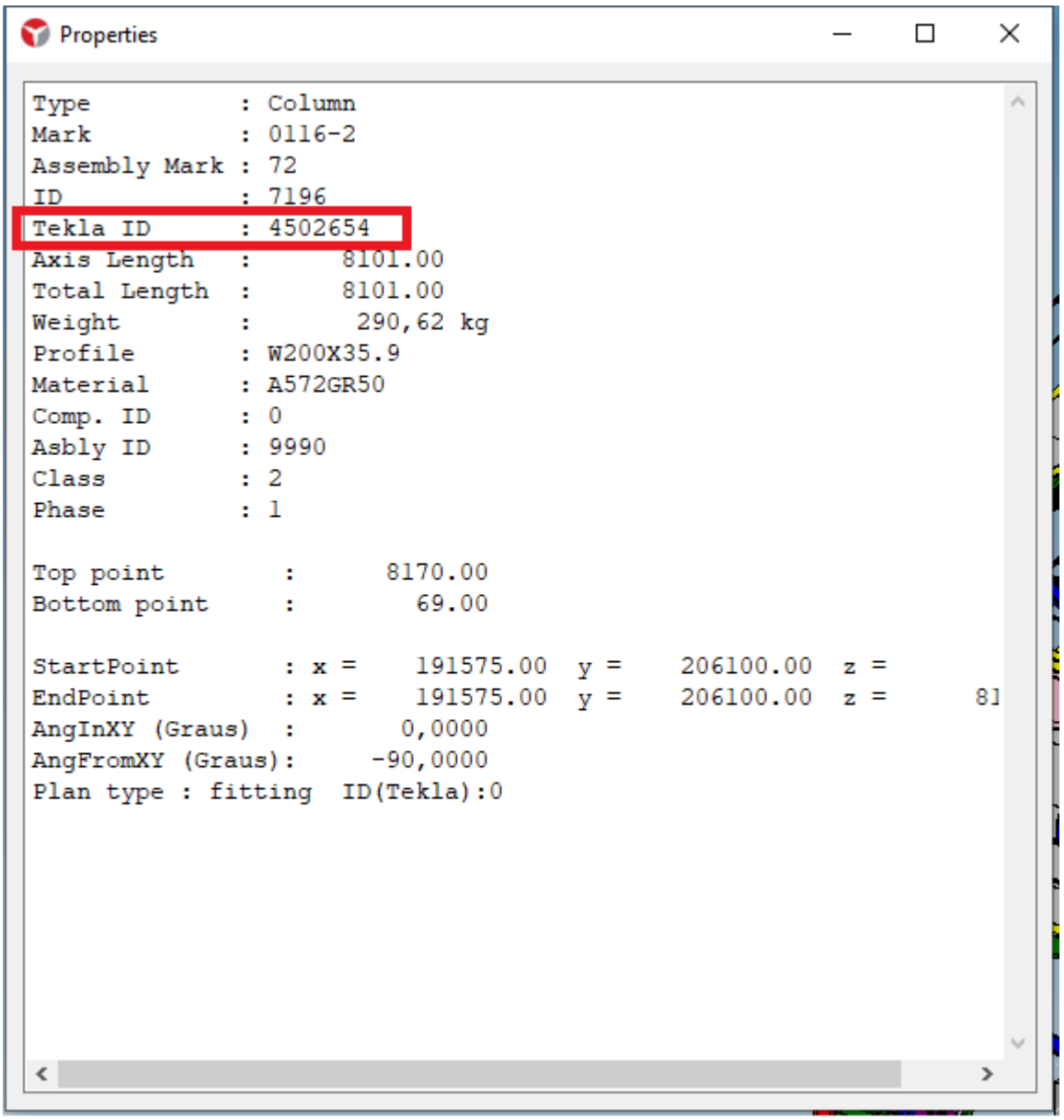
TSteel imports and exports a 3D model to Tekla. To do this, it connects through the Tekla programming API. This requires that both programs (Tekla and TSteel) are running on the same machine.

To export a column, for example, it sends commands to create the part (including profile, material, finishing, etc...) and then includes the cutting planes and cuts. The export to Tekla is therefore a new modeling based on the existing elements in your TSteel model.

Importing is the reverse process. TSteel reads all the information for each part and creates new parts based on this information.

Model synchronization

When you import a Tekla model into TSteel, it stores the ID of each part. You can check this ID in the part information window.



The advantage of this is that when you export this model to Tekla again, TSteel knows that this part came from a Tekla model. So, if it is present in the model during export, TSteel does not export the part, it synchronizes it. This synchronization is to find the differences and perform only these differences.

A simple example. You have imported a column from Tekla into TSteel. In TSteel, you made any modification (gauge, cutout, repositioning, etc…). TSteel will only perform the necessary operations on the existing part to keep the two in sync.

Export from TSteel to Tekla

TSteel 3D cannot create new gauges or new materials in Tekla, a limitation imposed by Tekla's own API. Therefore, it is necessary to check whether the necessary gauges and materials exist before starting the export process. An easy and quick process.

It is possible to export the profile catalog through *.LIS files. Remember to be careful not to overlap existing profiles. Are you in doubt? Carry out tests and progress little by little, seeing how Tekla behaves.
TSteel 3D helps you do this by creating a list of gauges and materials that need to exist in Tekla. This list is displayed in an iterative window that allows us to make the necessary correlations before starting the export.

Watch a video with an export.

Import from Tekla to TSteel

At the time of import, TSteel 3D identifies all existing profiles and materials in the Tekla model and then adjusts or creates these tables to receive the new model.
In other words, when you import a model, the process of creating new gauges and materials (when necessary) is automatic.

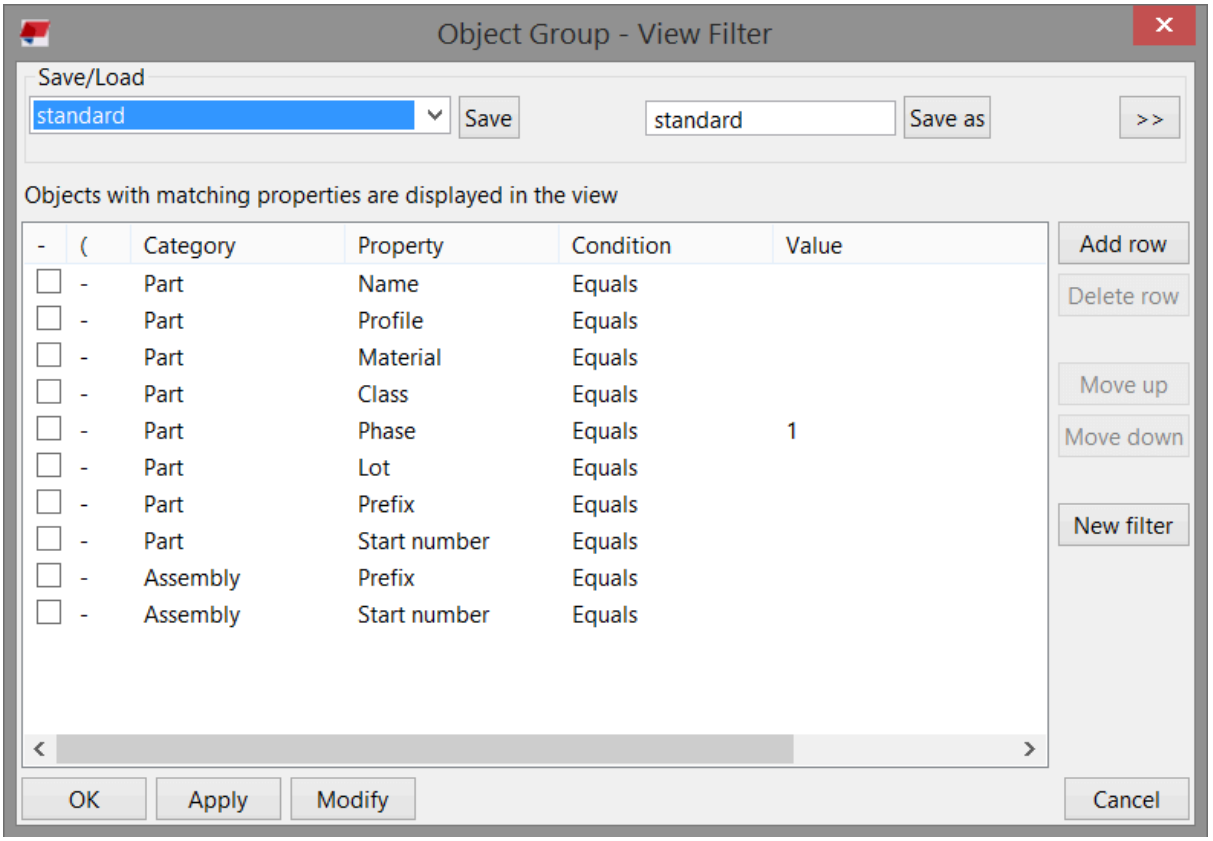
Careful: Older versions of the Tekla API did not provide access to the density value of materials, only the type. When importing from an old version, TSteel 3D automatically sets the density to 7,850 kg/m3 for steel and 2,400 kg/m3 for concrete. At the end of the import, TSteel 3D will issue a warning that these materials need to have their density checked manually. For newer versions, the entire process is automatic.

Checklist before importing

Before selecting the structure in Tekla that will be imported, it is important to check:

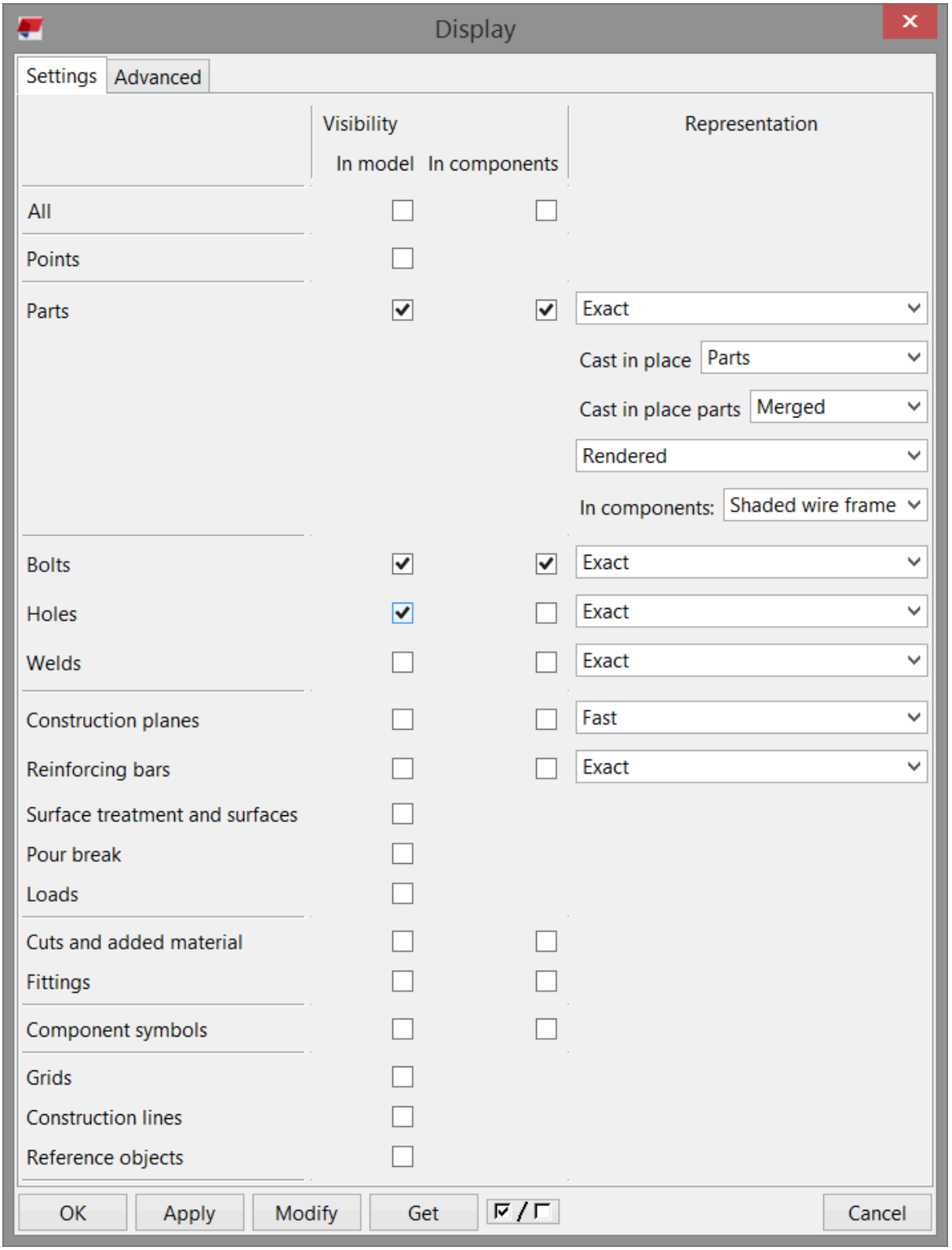
1. Display Filters

Make sure the object filter is not hiding any elements that should be imported.



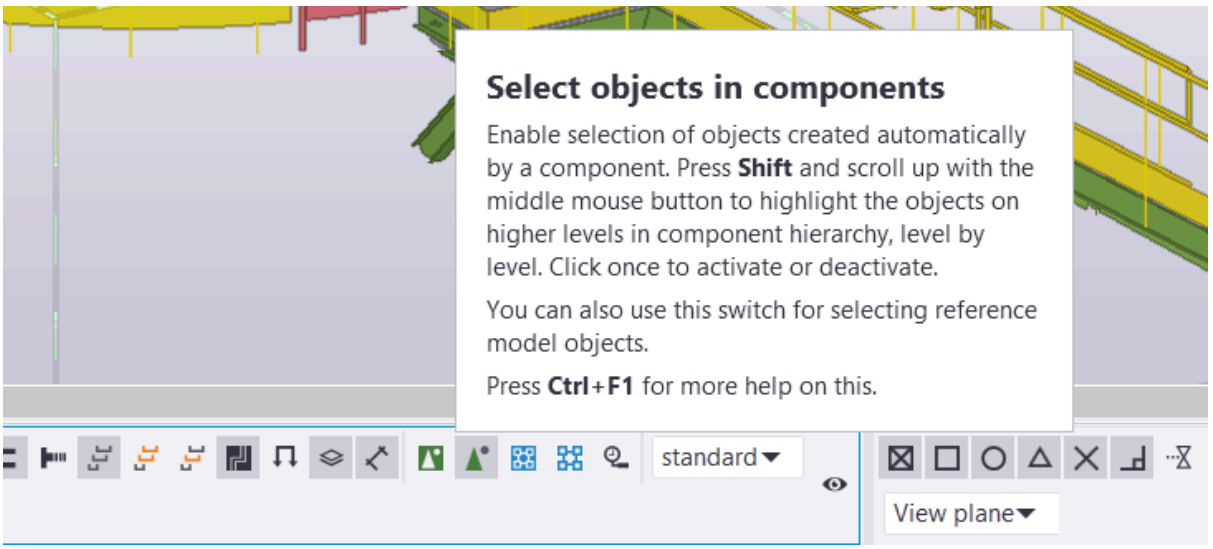
2. Object display

Check if you are displaying parts, bolts and holes.



3. Enable object selection within components

Always turn on the selection of objects belonging to components, this ensures that when you select a part of the structure, the parts and screws created by a component (macro) will be included in the selection and imported into TSteel 3D.



[Watch a video on how to import a Tekla model]

Configure connection with Tekla

TSteel 3D connects to Tekla via API (application programming interface), that is, when importing you need to have both programs open.

TSteel 3D automatically identifies the version of Tekla that is open and uses the DLLs that are installed on your computer. To do this, TSteel 3D needs to locate where the Tekla installation files are.

Where will TSteel 3D look for Tekla DLLs?

Typically in the default installation directory C:\TeklaStructures.
If you installed Tekla in another directory, just go into the settings menu and change.

What if I have more than one version of Tekla installed?

No problem, what TSteel 3D will do when it finds more than one version installed, is test all of them until one manages to connect.
We have already done connection tests from version 18 to version 2022, and they all worked without problems.

What if TSteel 3D cannot connect to Tekla

Possible reasons are:

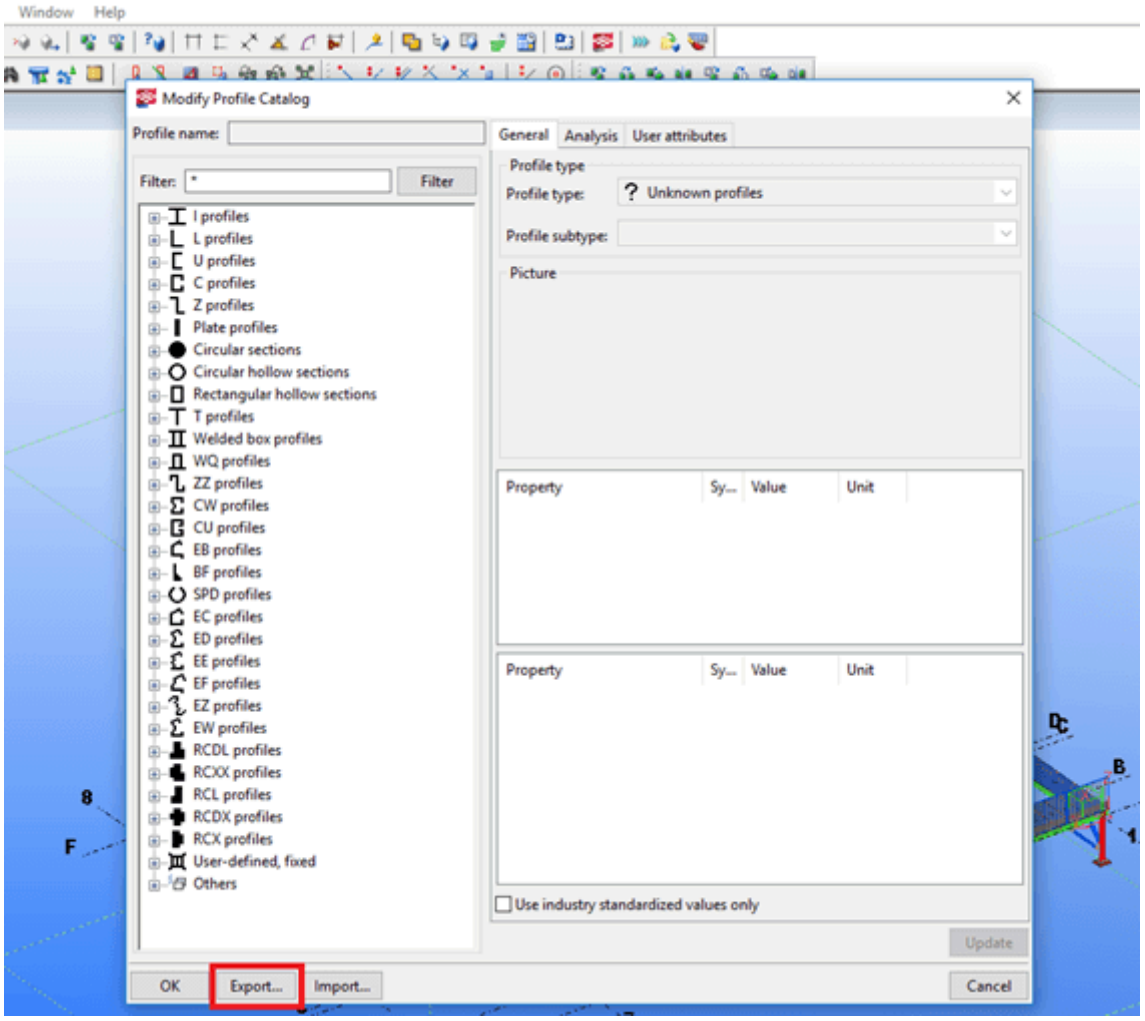
- TSteel 3D did not find the correct DLL to make the connection
- There was a previous connection break, to resolve this close and reopen both Tekla and TSteel 3D.
- The Tekla version does not allow connection via API. We have reports that some university versions were unable to connect.

Additionally, in the directory of the model you are using, TSteel 3D creates a file called “TeklaConnect.log”. This is a text file that contains step by step connection attempts and where they failed.

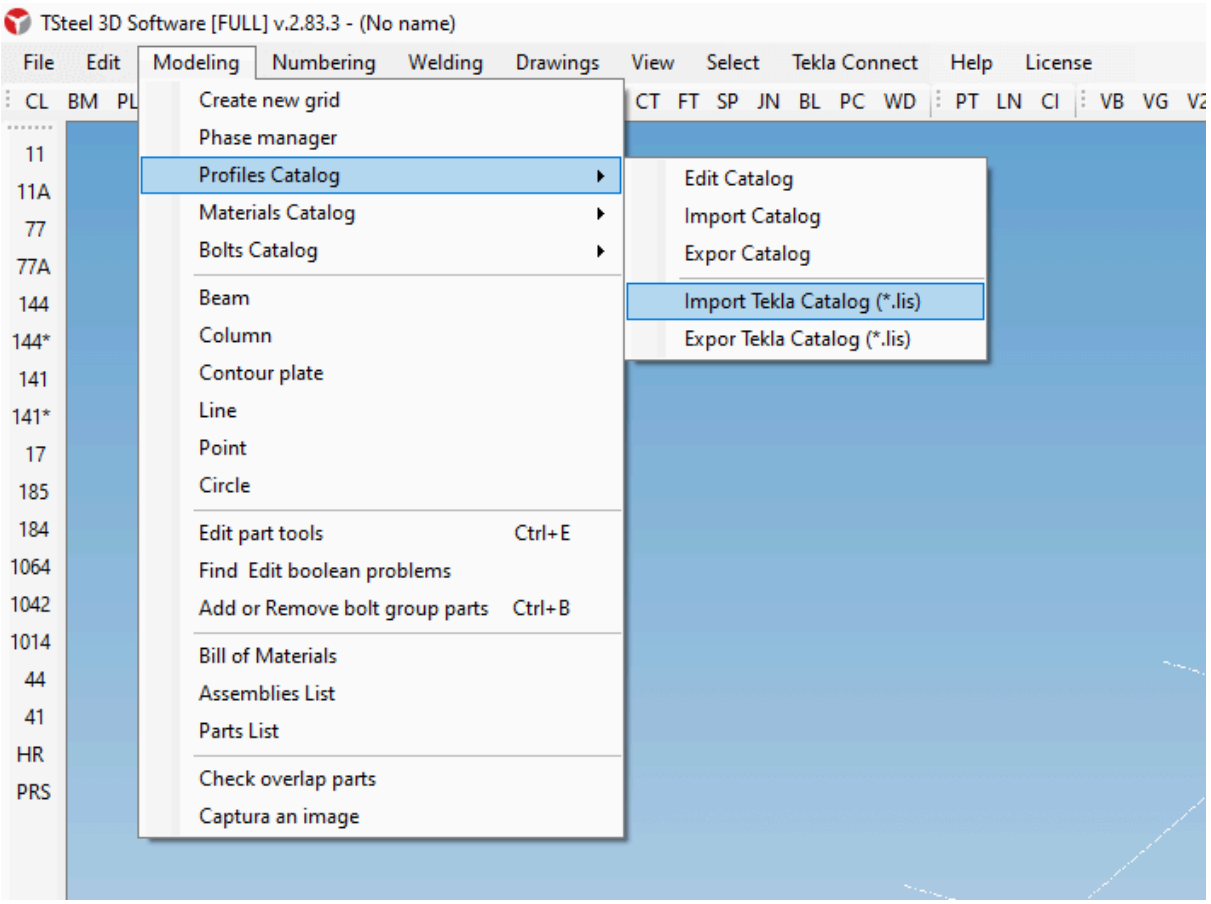
[Watch a video on how to resolve connection issues with Tekla]

How to import Tekla profile catalog

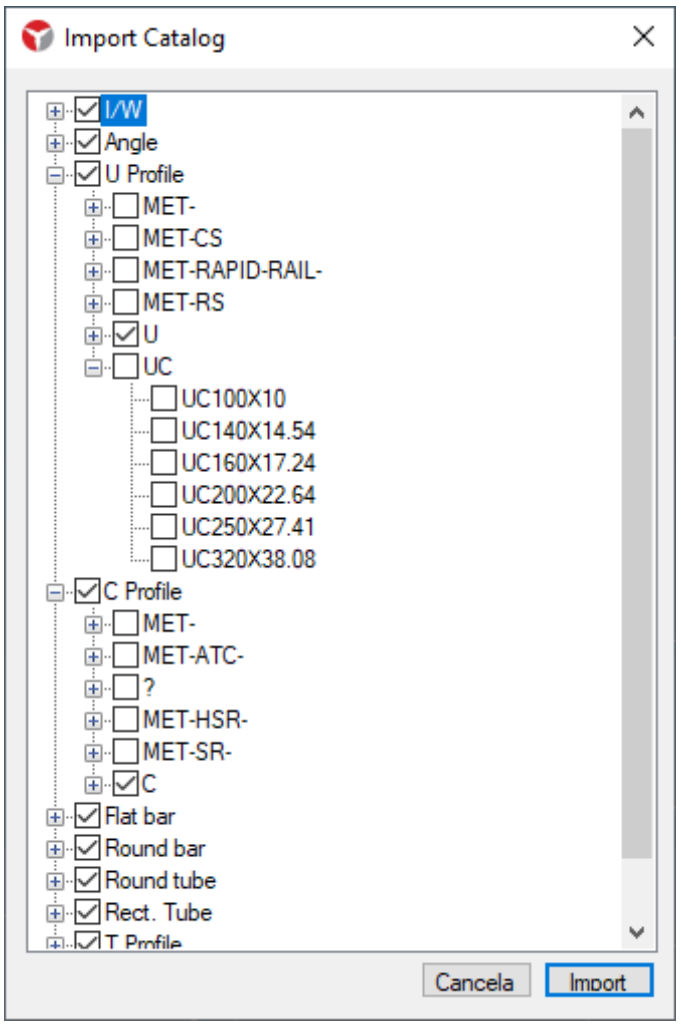
Open your profile catalog on Tekla



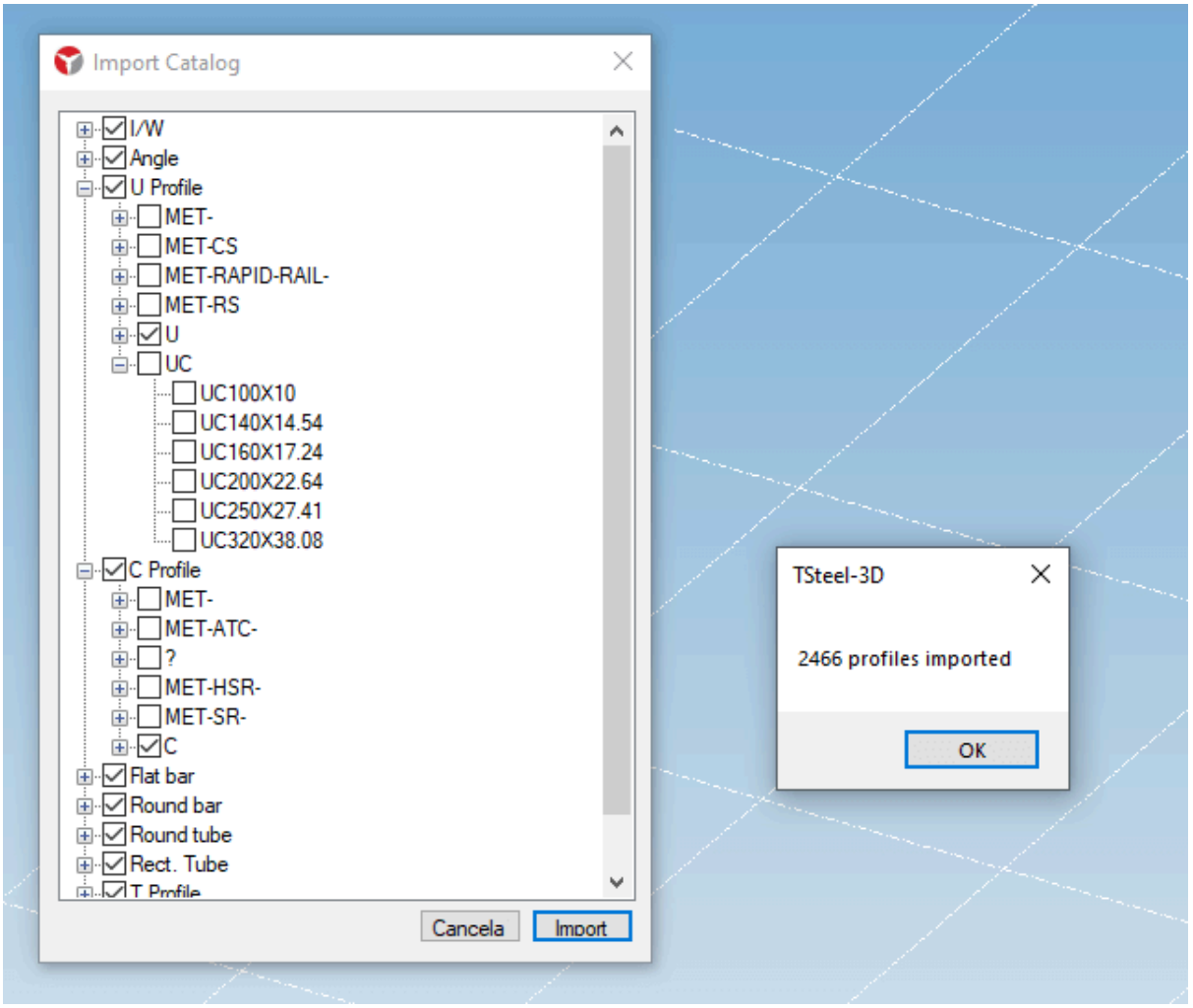
Click “Export” and create a file. By default, Tekla creates a file with the extension “.lis”
Once the export is complete, open TSteel 3D and select “Import Tekla Catalog *.LIS”



Not all Tekla profile types are supported by TSteel 3D. You will receive a message informing you if any type of profile cannot be imported.
TSteel 3D will then present a screen where you can choose which profiles you want to import into your catalog:



After choosing which profiles you need and clicking “Import”, you receive a message with the number of imported profiles:



Ready!, just enter theprofile catalog and check your import.

SDNF

Some calculation programs export the model in this SDNF (Steel Detailing Neutral File) format, usually files with the .SDN extension

The advantage of importing SDNF files is that the imported parts are native to TSteel and not just references (like IFC and DWG files).