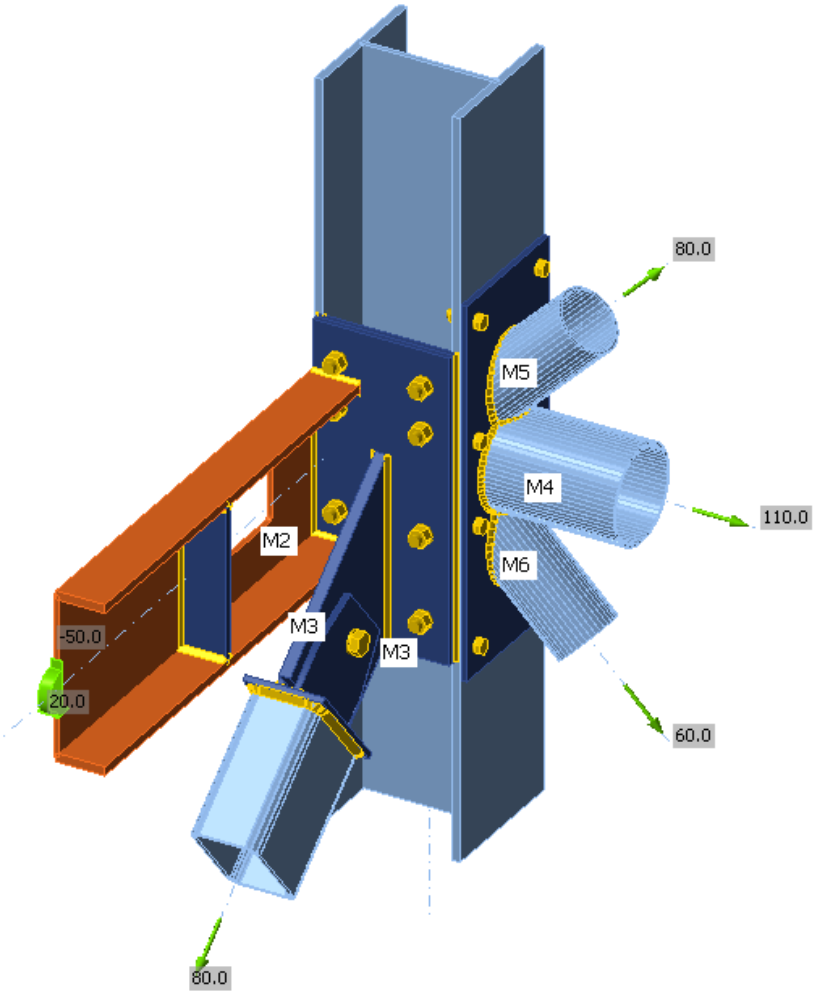
Quick Start Guide

IDEA StatiCa Connection



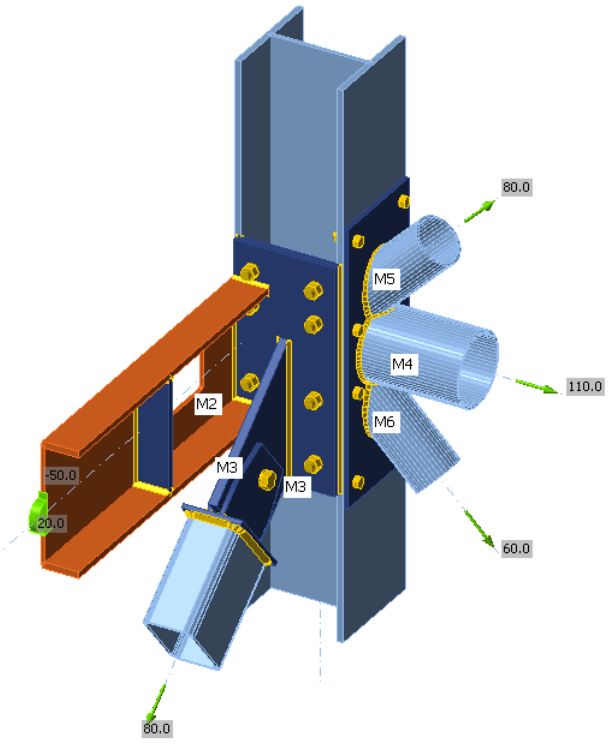
IDEA StatiCa®

Calculate yesterday's estimates

Introduction

We have created this Quick Start Guide to enable the first-time user to become familiar with IDEA StatiCa Connection as quickly as possible.

This is the connection we are going to build:



You may have seen this connection before.

The reason why we are building this connection is because it contains many features that only IDEA StatiCa can analyse efficiently. It also contains many standard operations in simpler forms of geometry.

As we create this connection, we will introduce to you to the key aspects of the IDEA StatiCa solution.

We will, however, assume that the software has been installed, licenced and runs correctly.

This guide is split into five parts:

1 Starting IDEA StatiCa Connection
2 Starting the connection design
3 Load effects
4 Operations
5 Analysis and verification

Part 1 – Starting IDEA StatiCa Connection

From your desktop or Start menu start IDEA StatiCa. This is the main portal to the applications within the IDEA portfolio.

The application we are concerned with is Connection. This can be accessed off the home page or the Steel page. If you are interested in any other application, please get in touch with your local IDEA StatiCa partner.

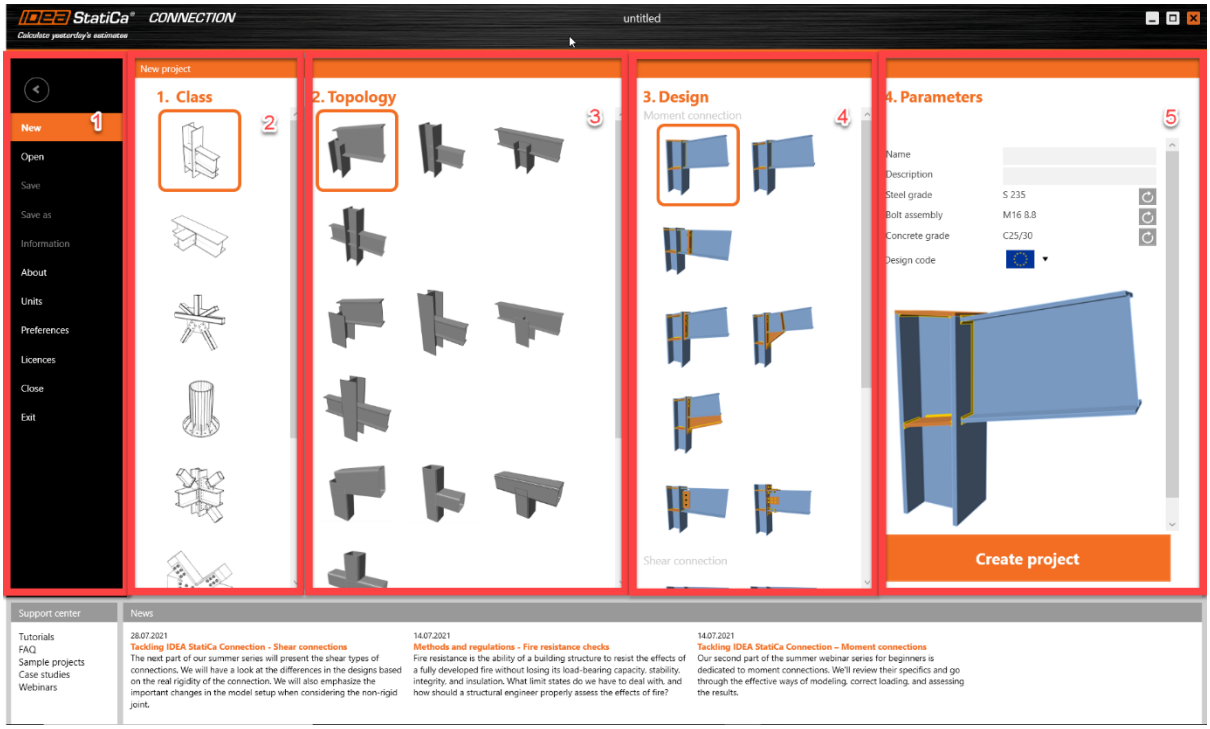
In the top right-hand corner, there are three additional icons:

From left to right they are licencing, initial setup and information. If you contact support, we may ask you to verify your installation with one or more of these buttons.

This screen also allows you to access your recent projects.

The right-hand side of this screen under 'News' contains information on events, blog posts etc.

After you start Connection, you will see the initial introduction or wizard screen which helps you setup an initial joint geometry. Please note that sometimes you may hear the word '*topology*'. This is just another name for geometry – it is the appearance of the joint you want to model.



The introduction screen has been designed to get you started on a connection as quickly as possible. It is organised as a series of panels.

1 Action to be taken (akin to the Windows File menu)
   a. New
   b. Open
   c. Etc.

2 Class of connection (2D/3D Simple/Not so simple)
   a. Beam to column
   b. Beam to beam
   c. Baseplate
   d. Etc.

3 Topology (Joint geometry)
   a. Top of column simple/not so simple
   b. Inclined connections
   c. Hollow section variants
   d. Etc.

4 Design
   a. Bolted
   b. Welded
   c. With/without stiffeners
   d. Etc.

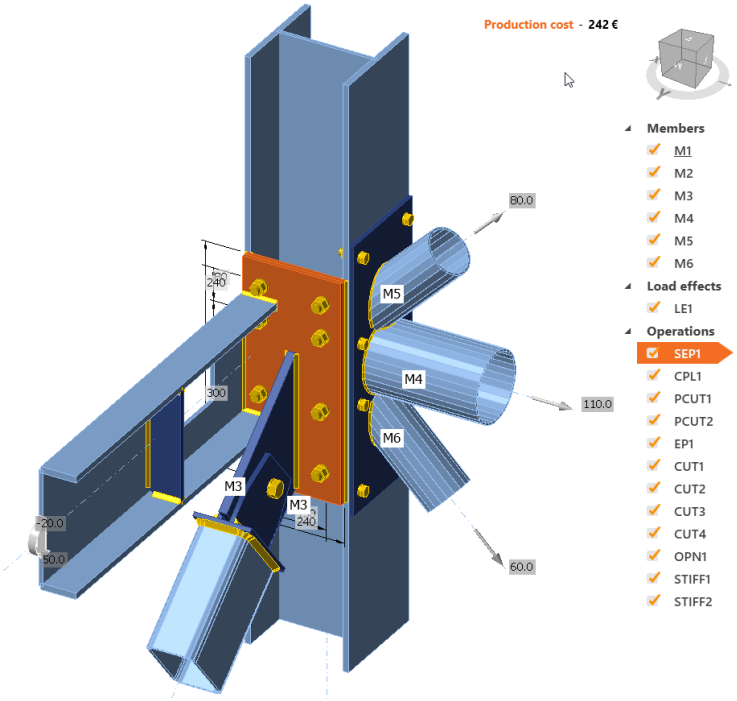
5 Parameters
   a. Name
   b. Description
   c. Code
   d. Etc.

The top row varies depending on the action you choose in panel indicated 1 above.

Beneath these panels are ones to access Support centre (tutorials, sample projects etc) and a News feed (like the one on the main portal).

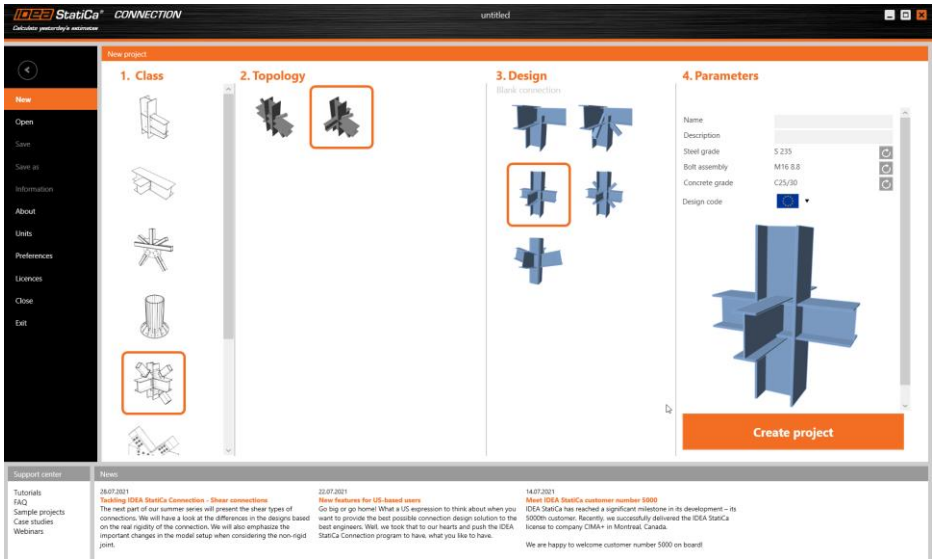
The figure above shows the layout if the action New is selected as the action. If Open was selected, for example, then the panel to the right would change to show the recent projects.

Part 2 – Starting the connection design



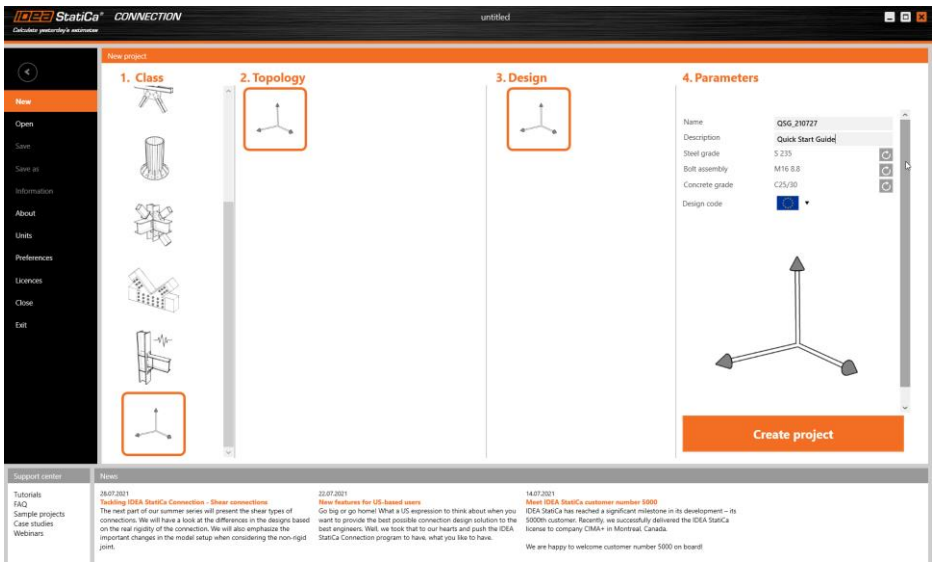
This is the connection we are going to design (in case you have forgotten already). It is comprised of members (6 No.), load effects (1 No.) and operations (12 No.).

There are several ways we could start this design. One would be to choose the relevant icons in panels 2, 3 and 4 to get near to what we have above. This could even be modified to use the braces in the Design makeup to the right of the one shown below.

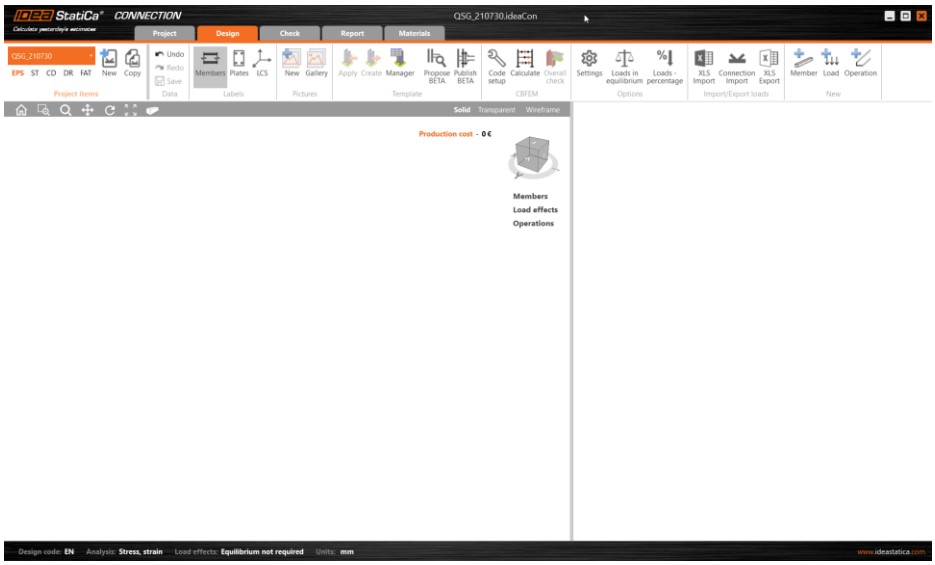


However, for this exercise we are going to go through building this connection from scratch. Look for the axis tripod icon in each of the panel and select this in each of the panels. In this example I have called mine QSG_210727 and given it a brief description. In the Parameters panel we can also set the default steel grade, bolt assembly and concrete grade. The important item to set is the design code. Our design will be based upon the European Code. It is quite useful to set the default grades to a

grade that will be used to most in the connection. For instance, if there were more hollow sections it might be useful to set this to S355. We will leave these grades as they are. Later, as we build up our connection, we will use the modification button  to change the current setting.



If you now press on Create project a blank project will be created. I find it quite useful to save the connection at this stage. In the toolbar above the workspace, you will find the Save icon. Choose a suitable name and save this connection design as it stands currently. Now we can start to build up the connection. Please remember to save your work at regular intervals!



The main screen is comprised of the toolbar across the top, the main workspace to the left and an options panel to the right (we will see more on this later).

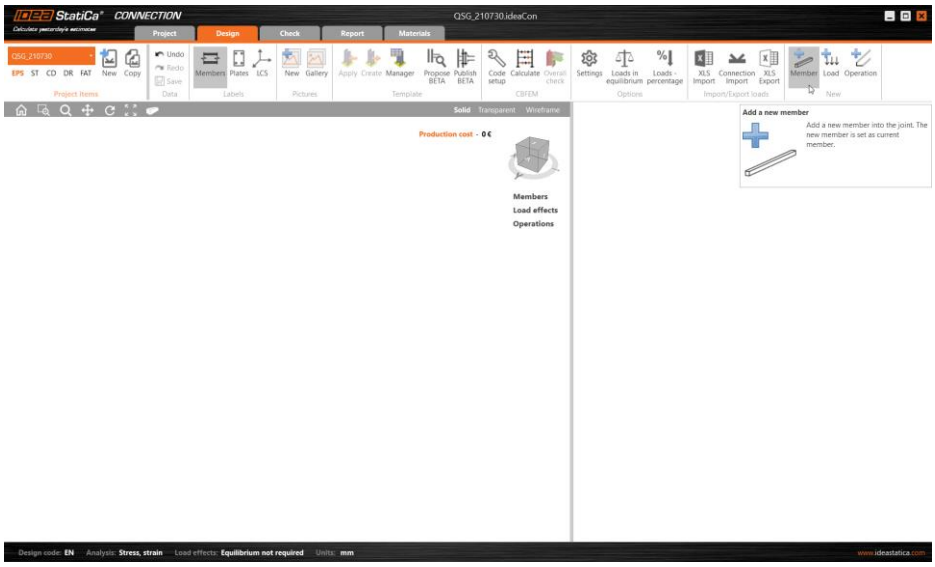
The first thing we will do is build up our members:

Member	Designation	Grade	Rotations
M1	254x254x73 UC	S275	0/90/0
M2	300x90x41 PFC	S275	-90/0/0
M3	200x120x8.0 RHS	S355	-90/-40/0
M4	168.3x5.0 CHS	S355	0/0/0

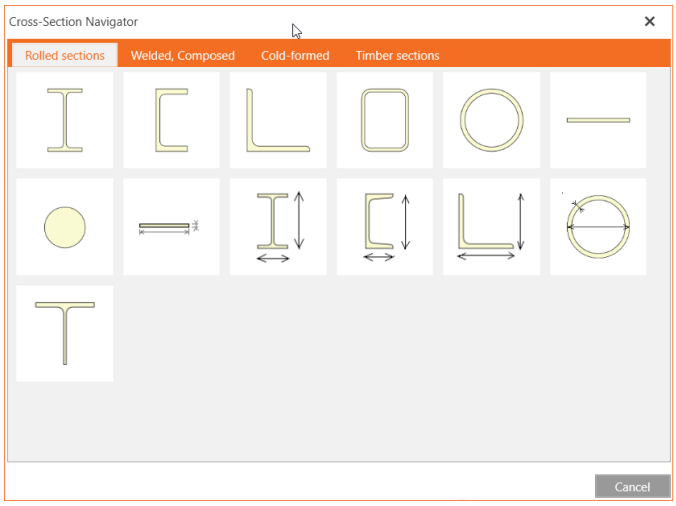
M5	114.3x5.0 CHS	S355	0/45/0
M6	114.3x5.0 CHS	S355	0/-45/0

I have included the member rotation above for you to refer to next. We will be using five different sections arranged according to the actual setting out requirements of the frame. This is important as when we bring our members into IDEA StatiCa it will resemble the frame as defined by your frame analysis solution with all the members aligned to one common node through their centrelines.

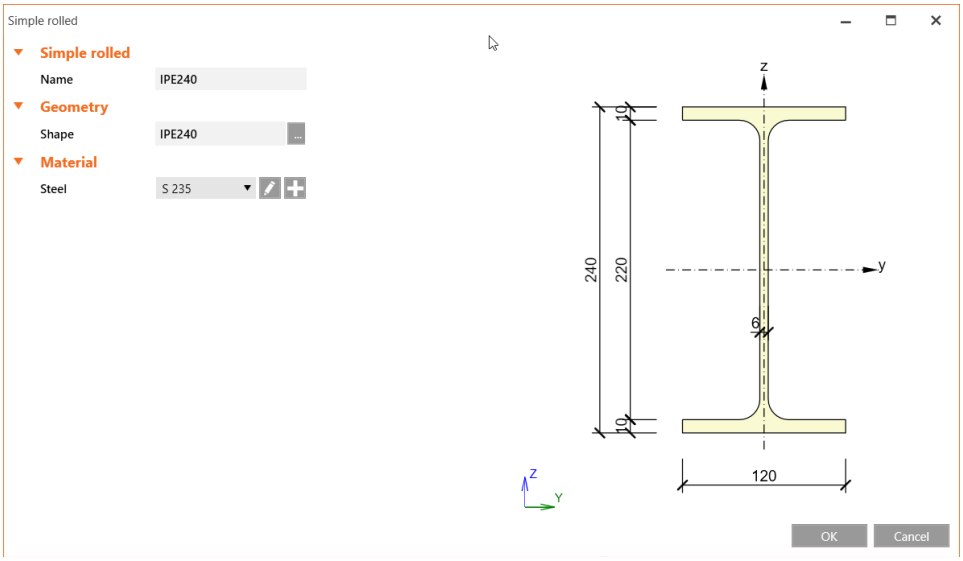
I find it best to be methodical and complete one thing at a time before moving on to the next. In IDEA StatiCa there are several options which allow us to visualise items on the screen. We will highlight these going forward.



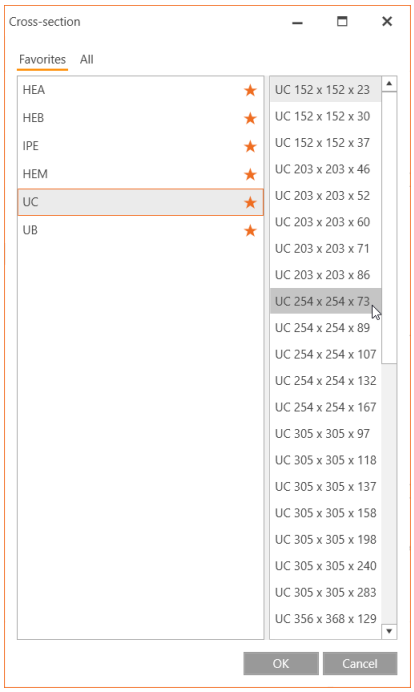
To add the column (M1) we go to Add a new member from the toolbar (or you can also right mouse click on Member in the Workspace and select New member). This brings up the following dialog which allows you to choose a suitable cross-section. As you can see there are many different types of cross-section but we are concerned with Rolled I section (top left).



When you select the rolled I section you can choose its Shape and Steel grade.

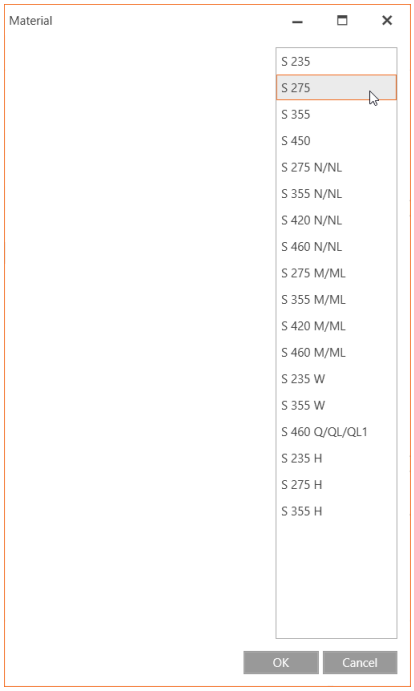


The button with three dots to the right of where it describes the section will take you to another page where you can choose from several libraries. You may think that there are not many libraries for you to choose from, but this screen is split into Favourites and All!

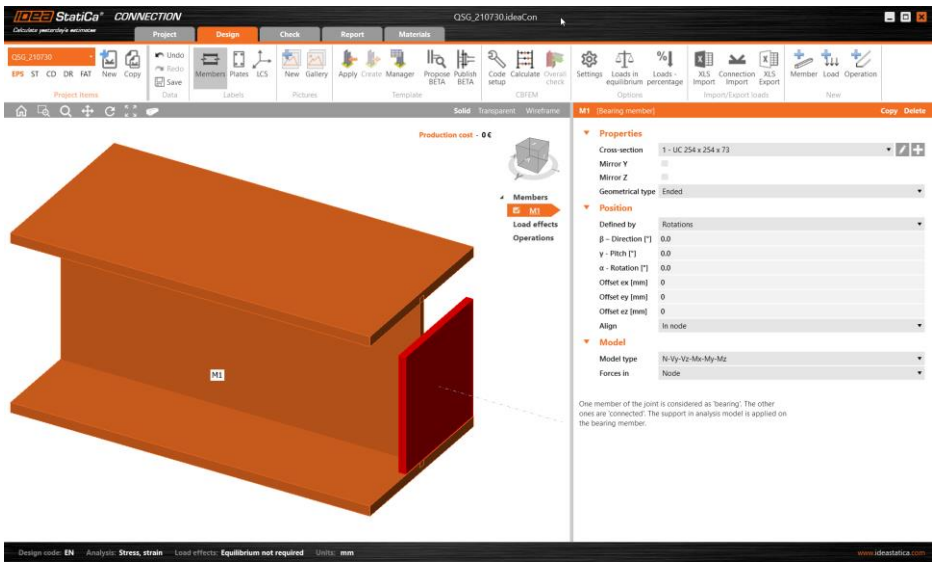


When you picked the correct section and then select OK this is inserted back into the previous input screen and now we can allocate the correct steel grade. You will see both a pencil and a plus symbol. The pencil allows you to edit what is currently indicated. However, it is not a good idea to edit a standard grade of steel unless you really need to!

Pressing the plus symbol brings up the material database.



Choose S275 (as per the table) the select OK and this is inserted into the previous input screen. Select OK again on this screen and you should see something similar to my screen below.



Congratulations you have created your first member in IDEA StatiCa Connection! Now the fun really begins! Notice that now we have lots of Properties to allow us to position and orientate this member correctly.

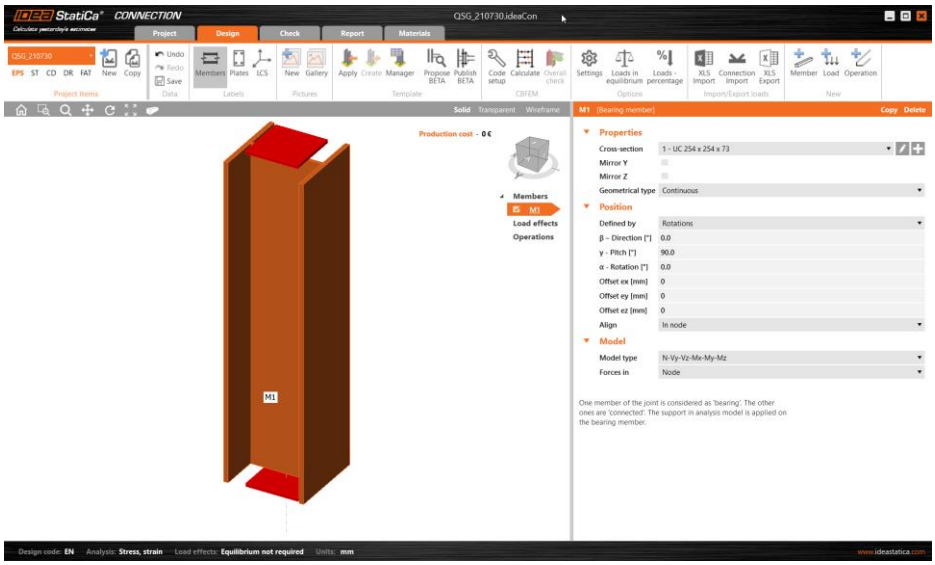
As this is the first member to be defined in the connection it is assumed that this is the bearing member i.e. the member that supports all the other members. If this is not the bearing member then you can right click on another and set that one to bearing.

Under Properties in this panel, you will see an entry called Geometrical type. It is currently set as Ended. An ended member is considered as fixed or supported at one end only (as shown by that red square on the member in the workspace). In our example the member is continuous as it is a column that that continues both below and above the joint. Changing this to Continuous both increases the length and adds another red square to the other end of the member.

This member is positioned using rotations. The rotations were given in the table. Applying those rotations gives us the vertical column. We are aligning this member to a Node. We will see late other options.

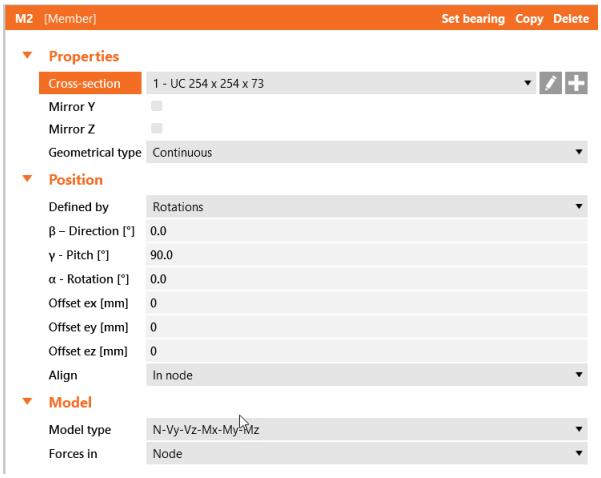
Under Model we see Model type and Forces in. There are many more tutorials which discuss these options in greater detail and I would refer you to those. In this example we can consider the column to carry forces of each type in any direction. Later we will see that a brace, however, will not be allowed to take any moments. We have obtained our member load effect results via a suitable frame analysis and we will be using these so we can keep Node as the preferred option for Forces in.

This is what your screen should look like after inputting all of the necessary options for this initial member.



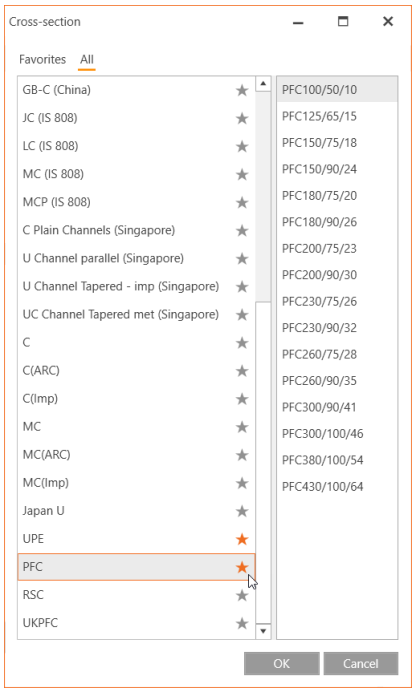
Moving on to the second member we know it is a PFC but this time it is aligned face to face not at the node so we must adjust it accordingly.

When you add the second member it takes on the form and location of the previous member.

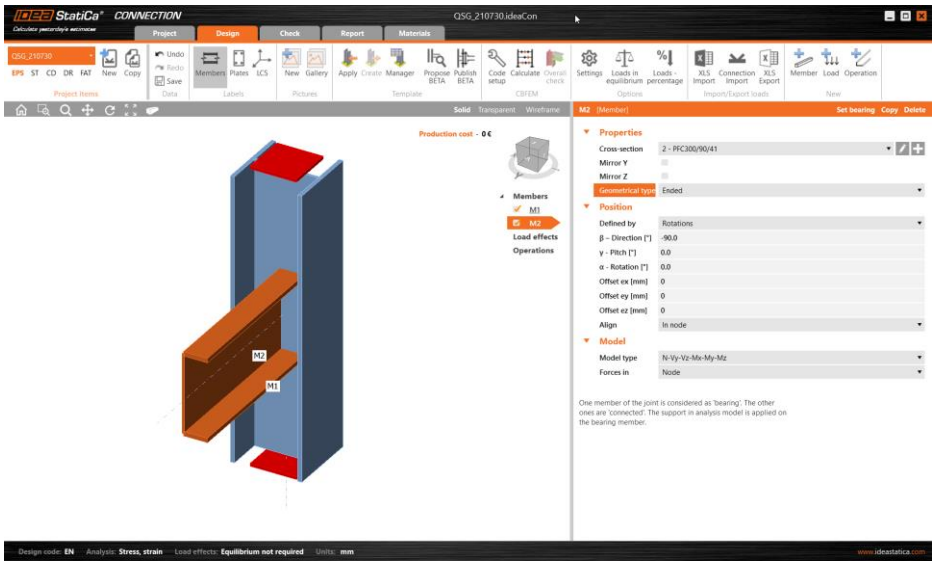


Notice that this time we have to change the cross-section using the plus symbol. If we edit this section to a PFC then both the column and the new beam will be that shape. This is because IDEA StatiCa uses a connection library of members that we add to, copy and edit.

Once you have pressed the plus symbol you must choose the type of cross section you require. This is a standard rolled channel section. You may not see the PFC library in the cross-section database because it is not a favourite. Just click on the star to have it appear from the start rather than going to the All tab.



Remember to select OK and change the steel grade to S275. Then you should have a PFC that is in the same space as the column. Remember to change the end type to Ended and the rotation angles as per the table to adjust its initial location.

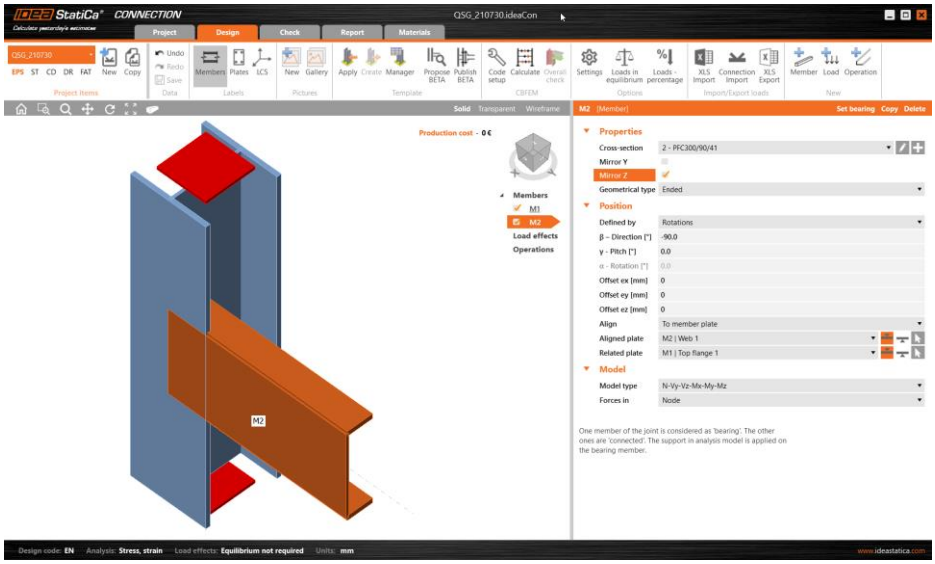


Member 2 now must be aligned to the outer face of the column. The view that we are currently in isn't very conducive so we need to change our viewpoint. We can either hold down the Ctrl key and the middle mouse button or use the view cube to 'spin' the view so that we are able to see both faces of the column and beam that we want to align or we can simply pick another corner of the view cube. This aligns the view to one of several pre-defined viewpoints. You can do the same with any edge or

face. If you get stuck then just press the Home icon on the grey banner above the workspace and this will reset the view.

We are going to change Align to Member plate.

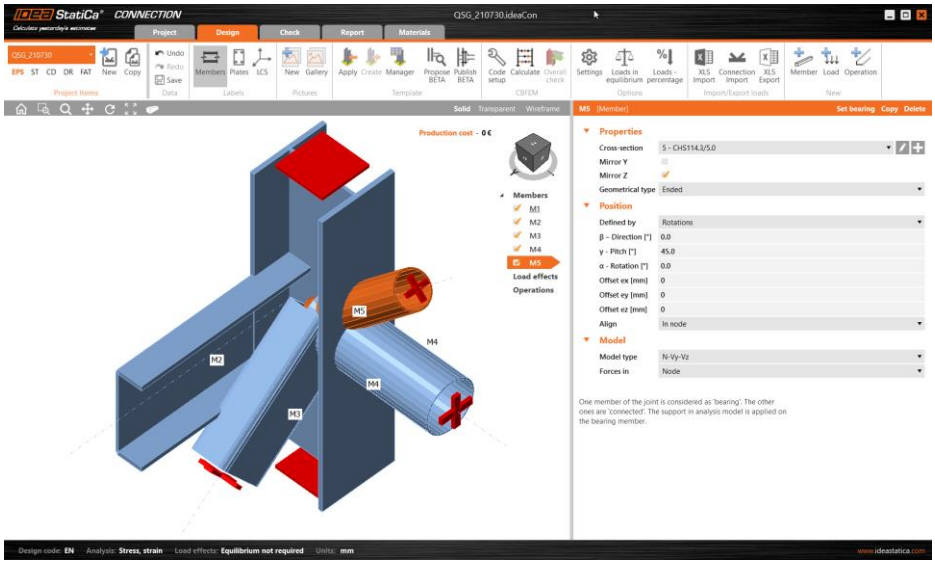
The aligned plate is the outer face of the web of the PFC. The related face is the outer face of the flange of the column. But which one? You can use the arrow button to select the flange you want.



We now have our second member.

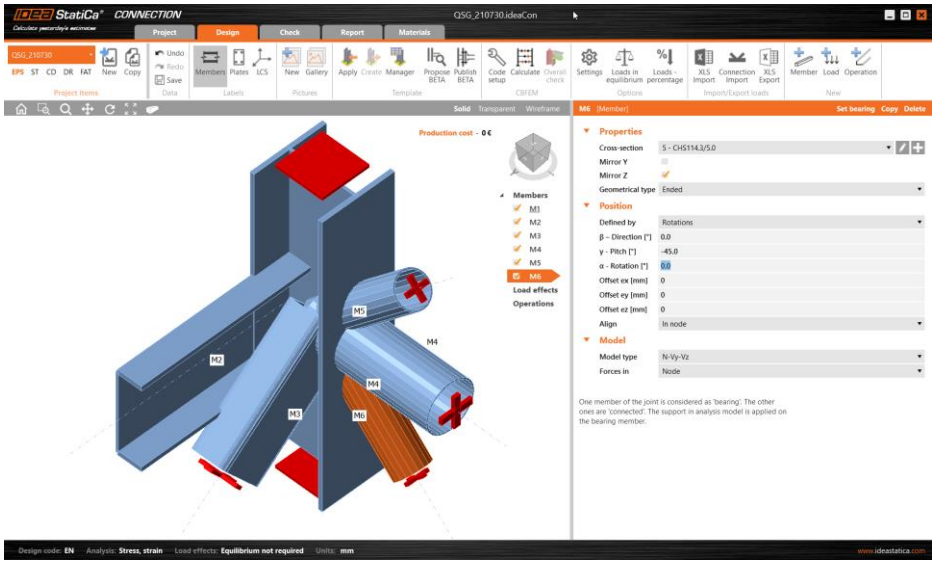
Using what we have learned already we can add M3, M4 and M5. Consult the table to get the right cross-section, steel grade and orientation. Remember also to change the Model type to N-Vy-Vz as these members will not be taking any moment.

At the end of placing five members your connection should resemble the one below.



To place the final member we are going to copy M5 and adjust its orientation as most of what we need is already defined.

Select M5 in the Member list in the workspace or actually select the upper brace that is M5. Then right mouse click on the highlighted member in the list and press Copy or press Copy in the orange banner of the properties panel. This will create a copy of M5 and name it M6. Then you can amend the inclination angle and we have all six members on the connection node.



In the next part we will add the load effects to the members.

Part 3 – Load effects

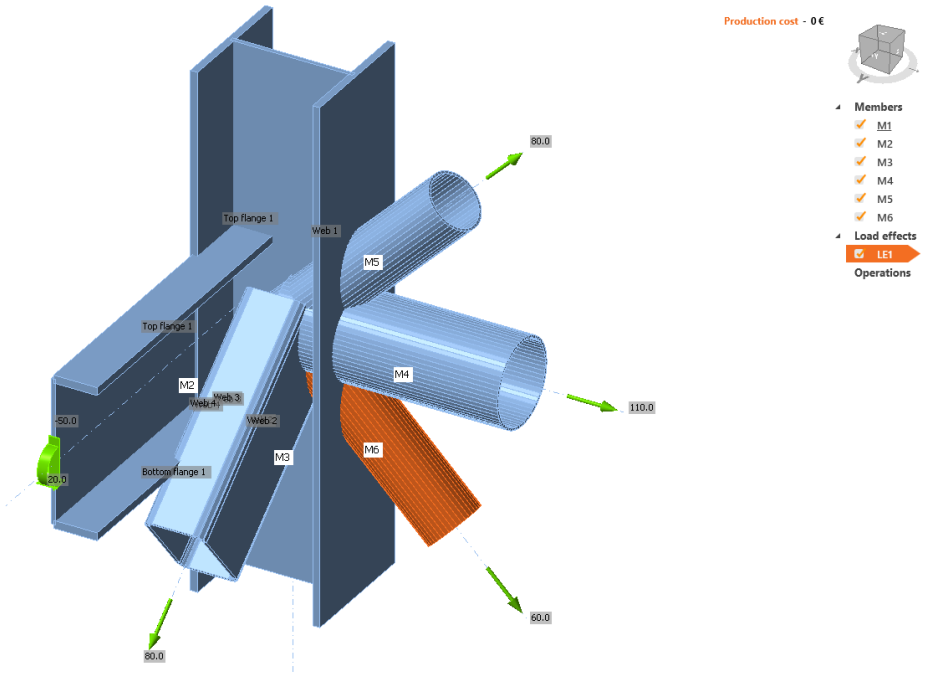
What are load effects? Essentially they are the forces that result on each member after a global analysis of the overall system or frame. Some refer to these forces as member reactions. After a global analysis all of these forces are assumed to act concurrently at the node. There are methods to adjust these forces so that they operate in a more realistic manner but that is beyond the scope of this guide.

We also have two ways we can utilise the loads applied. We can either take the attached members in isolation or we can bring into play the forces that are also acting on the bearing member by selecting Loads in equilibrium from the top toolbar.

For now we will only take into account the forces on each of the attached members we have using the table below.

Member	N (kN)	Vy (kN)	Vz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)
M2/End	0	0	-50	0	20	0
M3/End	80	0	0			
M4/End	110	0	0			
M5/End	80	0	0			
M6/End	60	0	0			

You will note that some inputs are automatically disabled as the member type for that member does not allow moments. The directions of the acting forces are also shown in the workspace and should be the same as below.



Part 4 – Operations

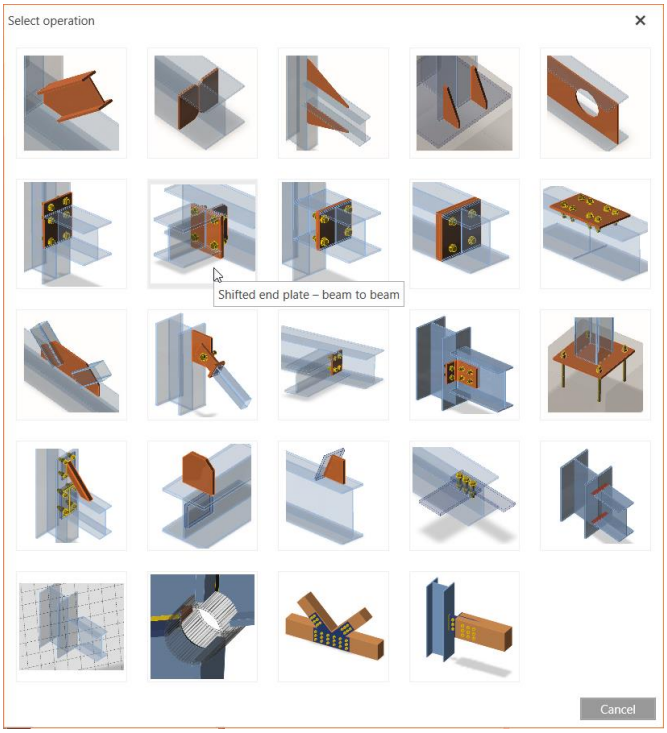
This part is quite complex and uses many different types of operations so we will break it down into smaller pieces.

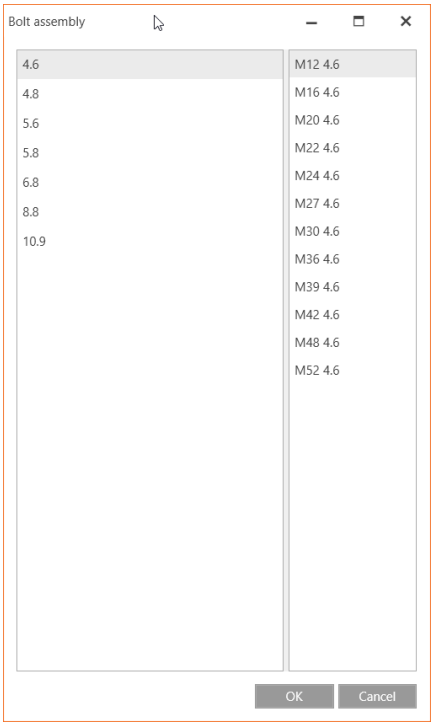
The operations that are created are executed by the software in the order in which they are created. It might be that one operation is in conflict with another, in which case you may see warnings to the left of the main workspace. Conversely if you see strange or unexpected results in the workspace this might also be down to the order in which an operation is processed. This is where the tick boxes come into their own as you can turn on and off members, load effects and operations using these.

4a – Shifted endplate M2 to M1

This is sometimes called a flange plate or 'dog' plate. There may be other local terms for such connections. If you know of anymore, please let us know!

Select New operation from the toolbar or right mouse click on Operations in the workspace and select New operation. Choose the Shifted endplate template from the Select operation screen.

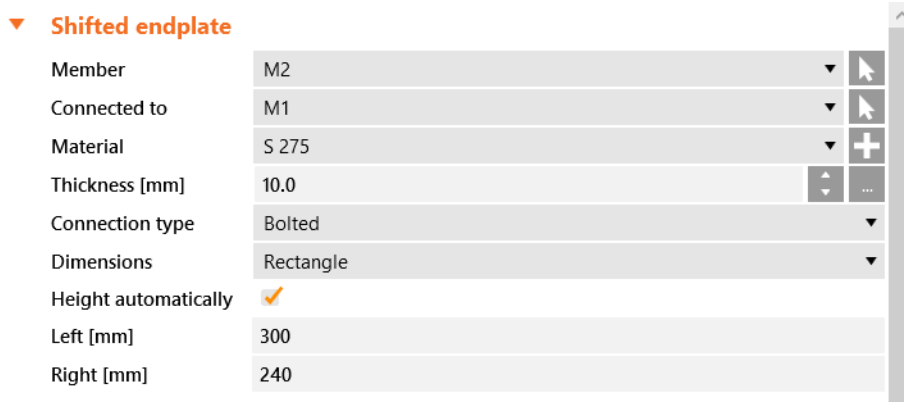




You are required to nominate a grade and size of bolt for this connection first. We shall try M20/8.8 bolts. Select OK to continue.

The connection immediately gets modelled to the best suited members based upon the current properties of the joint. If the connected members are incorrect then just amend Member to M2 and Connected to M1. The material ideally should be the same as the members but it can be left blank and IDEA StatiCa will take the default strength.

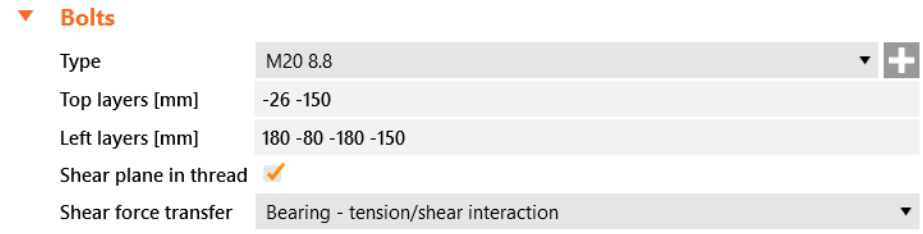
The Thickness, Connection type, Dimensions etc. are given below. Notice how the workspace reacts to the inputs you give here in real-time. There are many more options as to how to set operations out relative to themselves or members. Please see additional tutorials for more information.



Change the Beam endplate thickness to 0mm and its Dimensions to Identical. The two plates are now the same thickness and size.

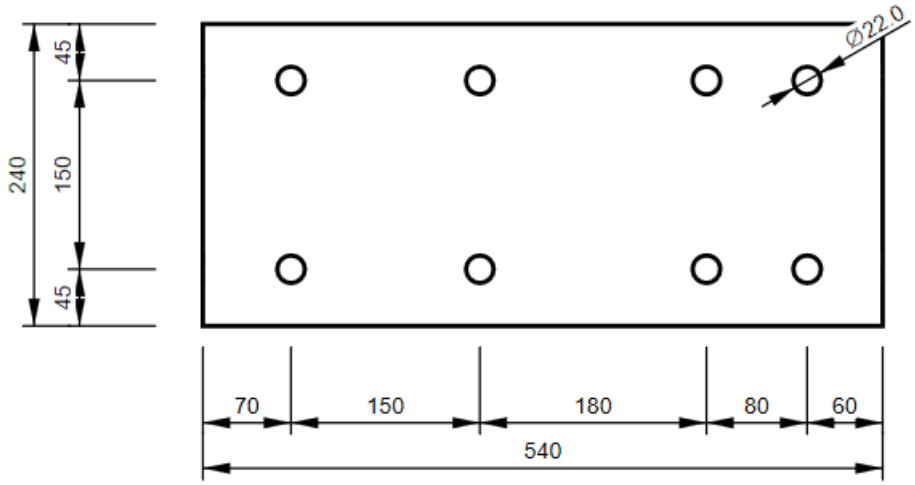
Remove the Stiffener by setting its type to None.

Enter the values within the Bolts section to those below.



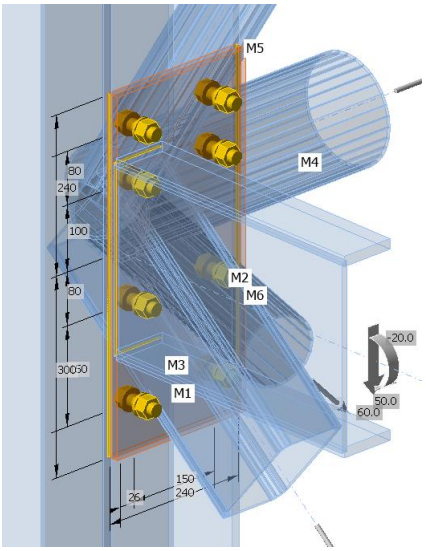
The syntax for the Top and Left layers follows some simple IDEA StatiCa conventions to arrive at a logical bolt arrangement.

If you want to see the overall dimensions for a plate or stiffener within a connection, you can use the Editor option and switch to Drawing to show the dimensions.



How did we get to the bolt spacings above from the dimensions entered in the bolts section?

First of all, we will turn on the Transparent view in the workspace. If we also highlight the operation, then the dimensions associated with the plate will display.



Possibly the first thing you will see is that the plate is orientated to a different angle. You can see where the distances for left and right are coming in and these are relative to the centre of the PFC. Left was set as 300mm and is the lower dimension whereas right was set as 240mm and is the upper dimension. This helps orientate the plate and hence the bolts. The depth of the plate is being controlled by the section it is being welded to and will adapt to a change in section which is useful to know.

Turning to the top layers of bolts these are called up as -26[space]-150. -26 refers to the distance and direction for the first row (or column) from the PFC and -150 is the distance and direction to the next row (or column).

The left layers are defined as 180[space]-80[space]-180[space]-150. 180 refers to the distance and direction to the first column (or row) from the PFC. -80 is the distance and direction to the next column (or row) and so on.

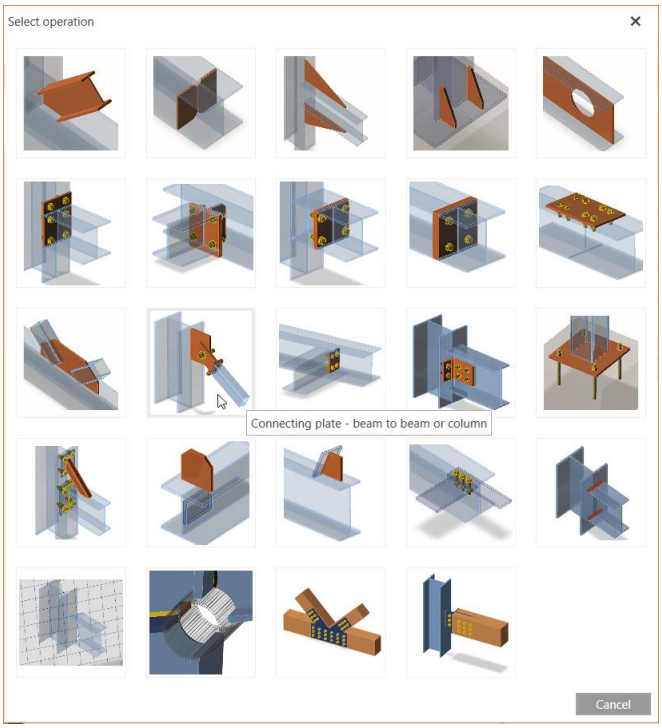
There are many ways to position bolt groups in IDEA StatiCa. There are many ways to position the same group of bolts using different combinations of numbers and signs. Please refer to the tutorials we have on our support centre for more information.

Congratulations you have completed your first operation!

4b – Connecting plate M3 to M1

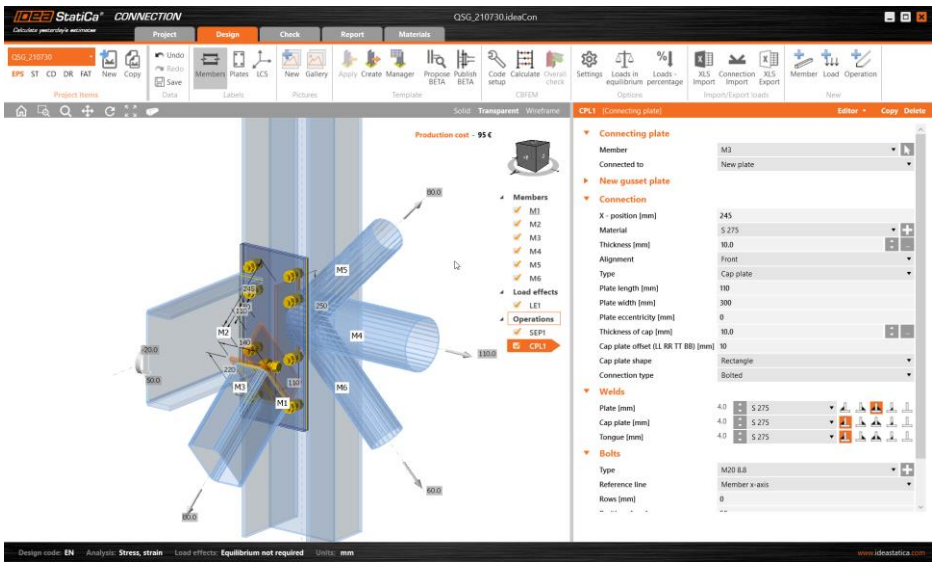
This operation creates a connection that is typically applied to a brace and connects to a column and/or beam.

Create a new operation. The operation will most probably be applied to M2 but don't worry, just change the member to M3 and it will start to look much better!



This is also connected to a new plate.

The new gusset plate is at first associated with member M1. To see this more effectively you can turn on transparency mode for the workspace. This will show you exactly what the connection looks like as you enter the relevant details.



The new gusset plate is not related to anything so select the X. This has an immediate effect on the plate that was connecting to member M1 in that it now aligns to the brace axis. Make sure that X is selected in the also related to field as we want this plate to be considered in isolation.

The material can be left as S275. Change the thickness to 20mm, the width to 250mm and depth to 600mm. The plate is now quite different! Don't worry if the length of this plate appears too long – we will be doing more operations on this to tidy things up!

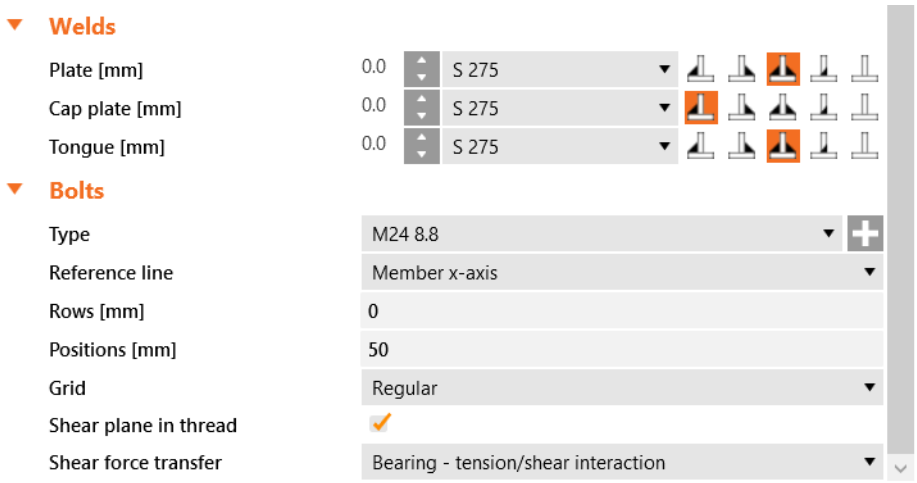
Moving on to the connecting plates. By moving the X position we are shifting the point of connection down the axis of M3. Set this to 480mm. We can keep the thickness at 10mm. However, if we want IDEA StatiCa to choose a thickness for us we could set this to 0mm.

We shall keep the alignment as front and the type can be asset as cap plate 2x. This creates a sandwich of plates. Feel free to experiment with other options.

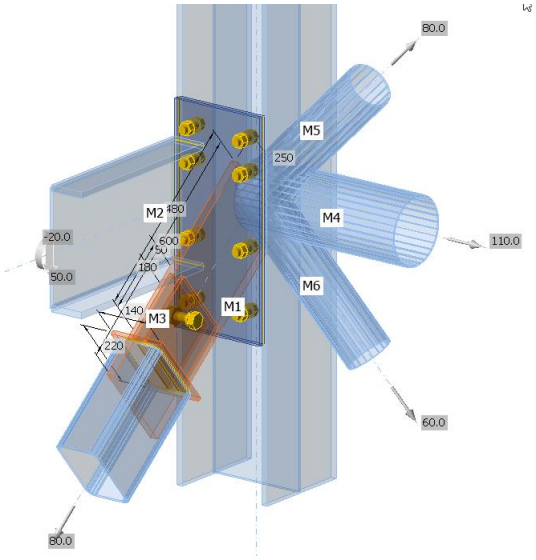
Set the plate length to 180mm and width to 200mm. Keep the eccentricity to 0mm, cap thickness as 10mm and cap offset as 10mm.

The shape and connection type is rectangular and bolted.

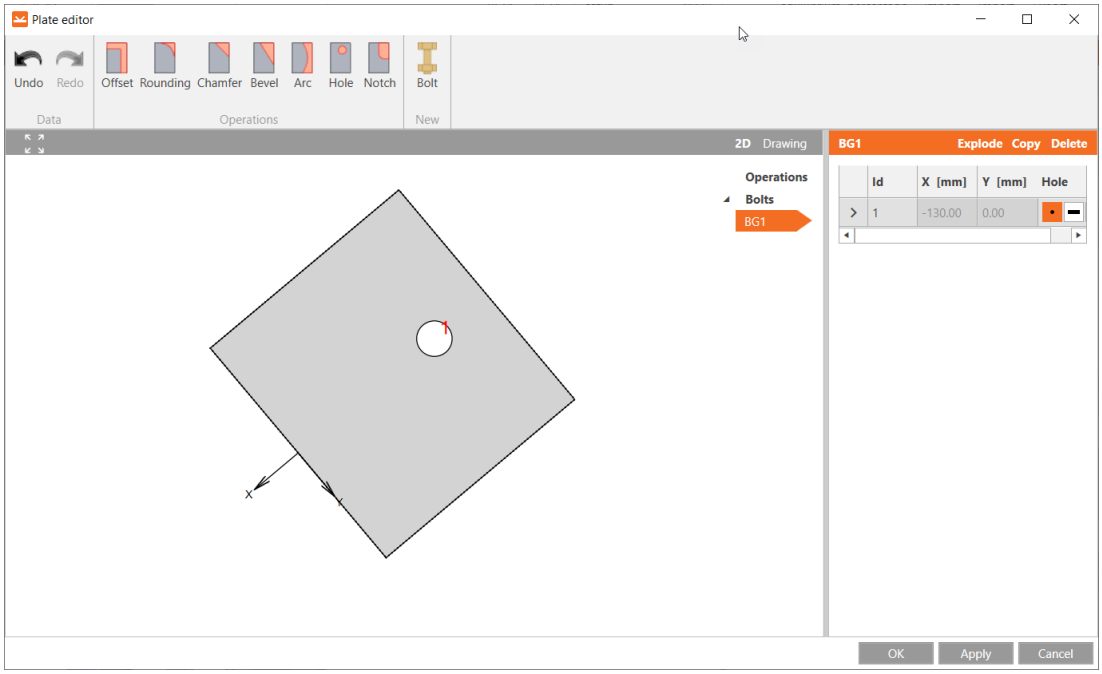
Adjust the welds and bolts to the extract below.



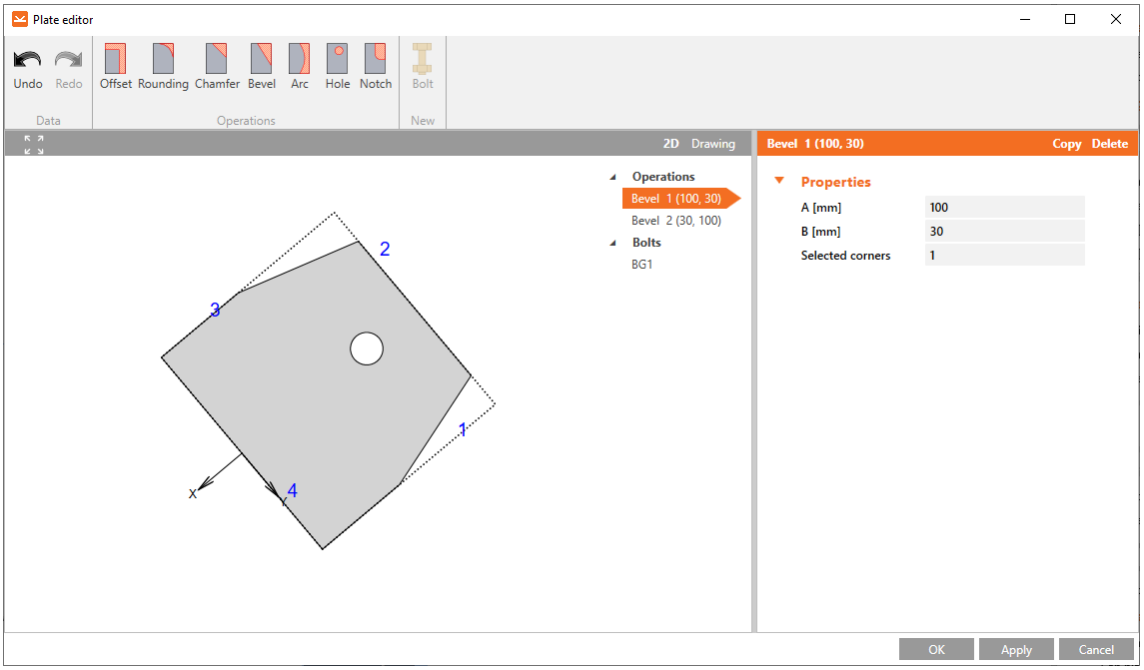
Note that we have changed the diameter of the bolt to M24. Remember that to add additional entries we use the plus button and select a different diameter. Th joint should appear like the one below.



The tongue of the connection needs adjusting to make it more visually appealing. Built-in to the connection is a plate editor. Within the orange band where the properties of the operation are described you will find the editor with a drop-down arrow. If you expand this you will see several entries. The one we are going to edit is the tongue.

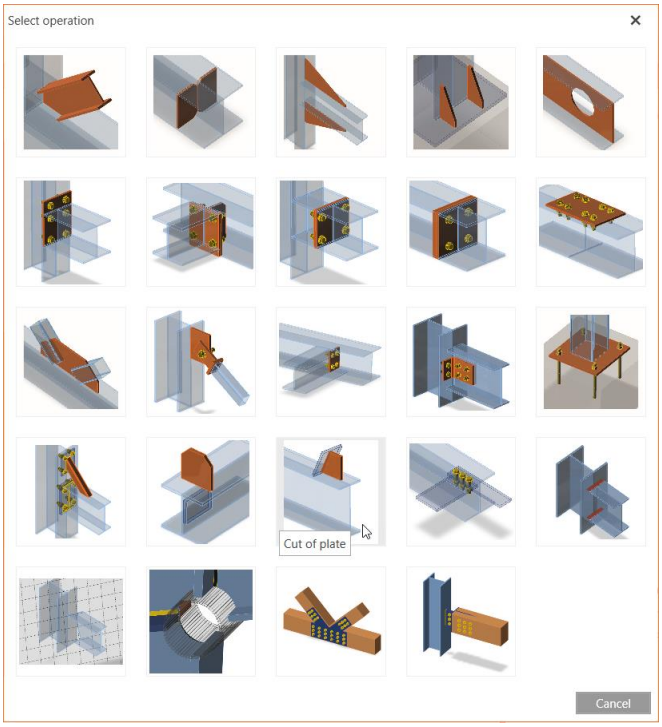


Currently we see a rectangular plate. We would like to take off a 30x100 bevel from each of the corners nearest the hole. When you select bevel the plate editor shows an example bevel on corner 1. The editor has annotated the corner numbers (the nearest number to the corner). The two corners we want to amend are 1 and 2. As you change the values the appearance of the plate changes in the editor. Pressing Apply will commit these changes to the plate and pressing OK to the model.



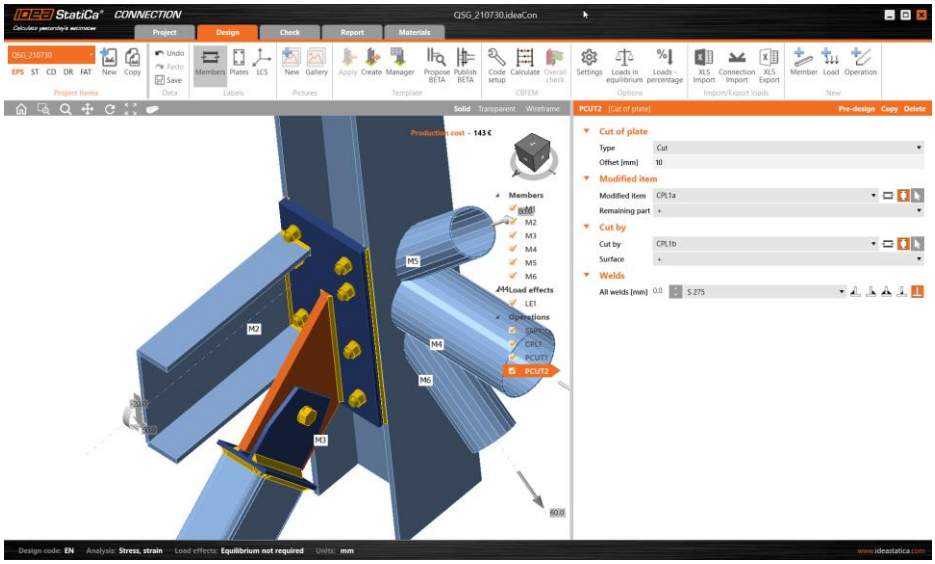
To complete this part of the connection we need to cut the gusset plate to the existing flange plate and the cap plate of the brace. The flange plate will be with zero clearance and welded whereas the cap plate cut will be with an offset of 10mm and not welded.

Select cut of plate as the new operation.



We are going to modify the gusset plate. Use the plate option and arrow to select the correct plate. It is going to be cut by the existing flange plate. Again, set the plate option and arrow to select the correct plate. You can experiment with the + and – options for remaining part and surfaces.

Can you do the same for the other end where it needs to be cut back from the cap plate? As you are modifying the same item there is an argument to use the copy option of the previous operation and change what the plate is cut by and its offset.

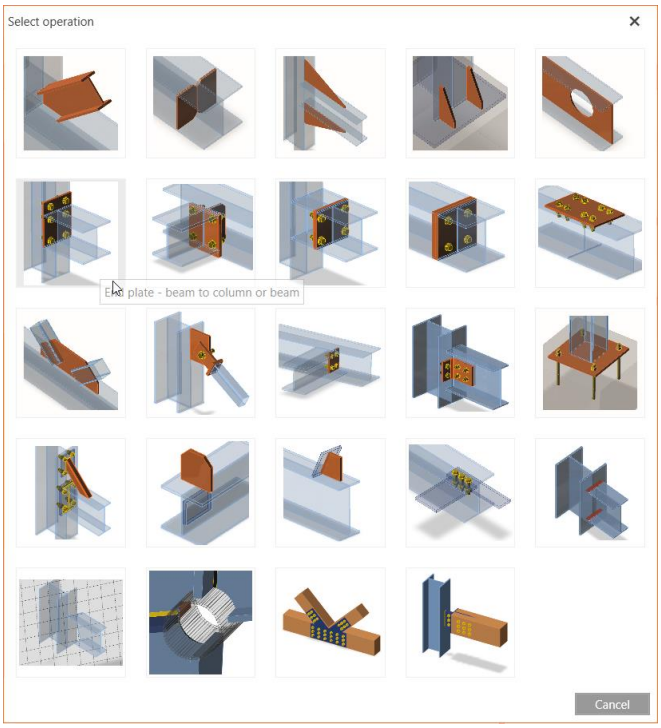


The above is what the finished connection should look like after both plate cutting operations.

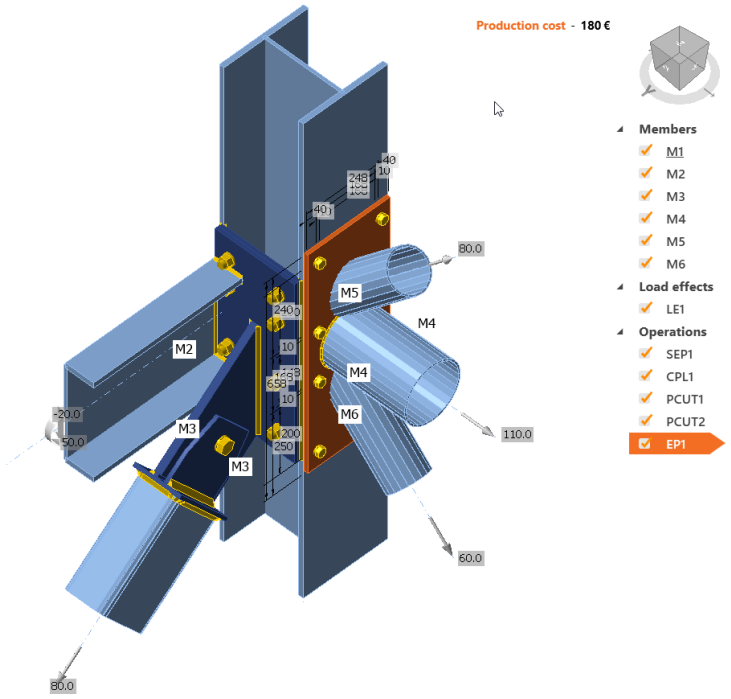
Part 4c – End plate connecting members M4, M5 and M6 to M1

This connection is quite complex as it involves one end plate with three CHS members cut back to it and coped against each other.

The initial connection is formed by using a standard endplate.



We are going to use M4 as Member 1 and connect it to M1. Use the values below to help you create the initial connection.



EP1 [End plate] | Pre-design | Editor | Copy | Delete

End plate

Member 1	M4
Member 2	Not specified
Connected to	M1
Material	S 275
Thickness [mm]	10.0
Connection type	Bolted
Dimensions	To profile
Top [mm]	240
Left [mm]	40
Bottom [mm]	250
Right [mm]	40
Backing plate	☐
Notch	☐

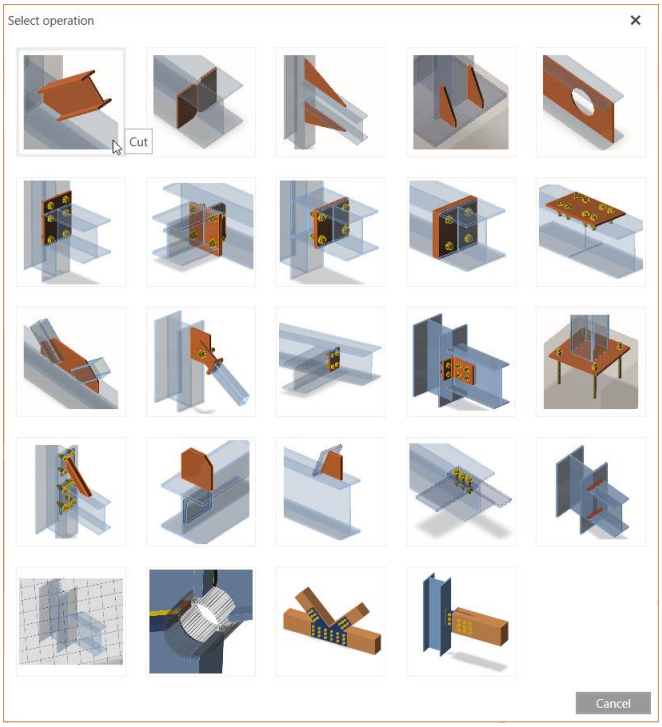
Bolts

Type	M16 8.8
Top layers [mm]	200 -210
Left layers [mm]	10
Bottom layers [mm]	200 -210
Right layers [mm]	10
Shear plane in thread	☑
Shear force transfer	Bearing - tension/shear interaction

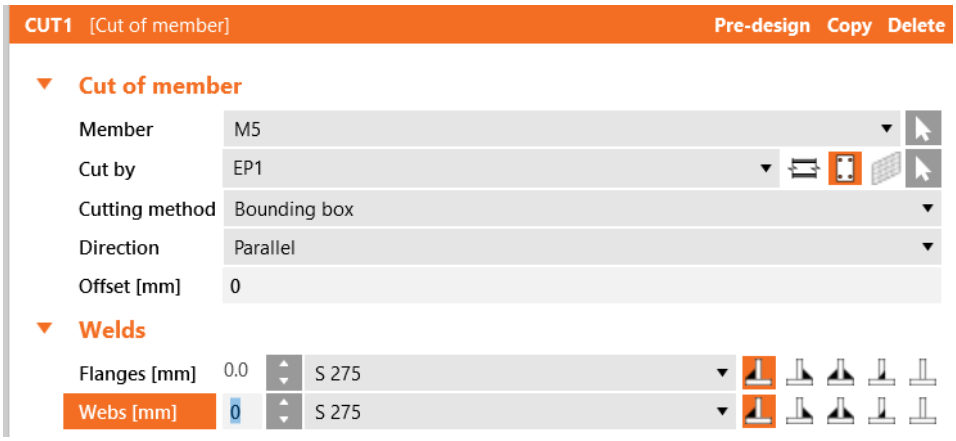
Welds

Flanges [mm]	0.0	S 275
Webs [mm]	0.0	S 275

We now need to cut and cope members M5 and M6 using the cut operation four times.



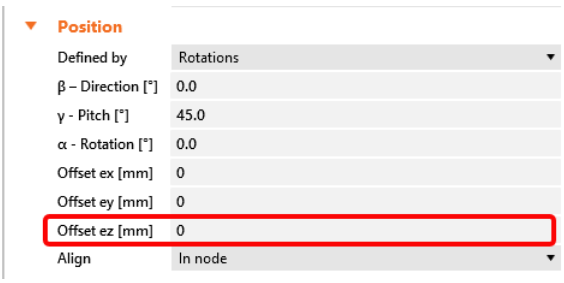
We are cutting each member back to the end plate and welding it using a default weld. The entries should look like this.



You can also copy this operation to member M6.

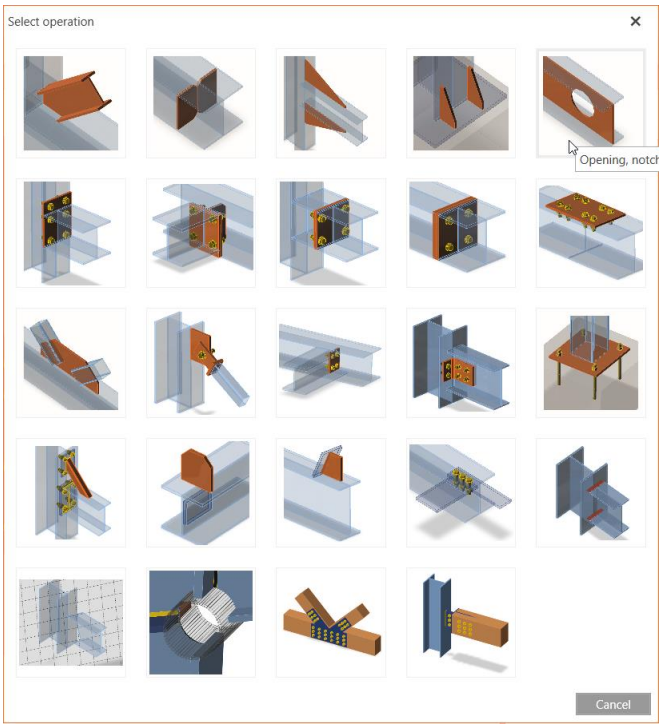
To create the coped profile to M4 we need to cut members M5 and M6 to M4. I would suggest the same principles apply: do one and then copy it. Remember to use the surface as the cutting method.

Alternatively, can you think of a way to avoid this complicated coping? Hint – you can control the separation of the Circular Hollow Sections by their z component.

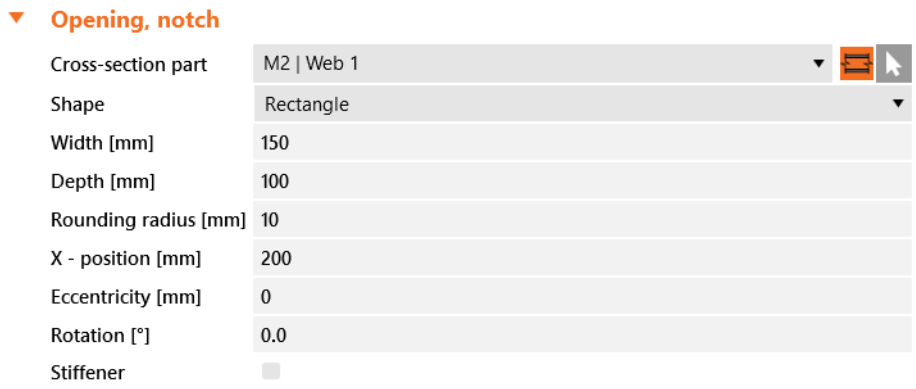


Part 4d – the opening and stiffener in M2

We will be using the opening operation on M2.

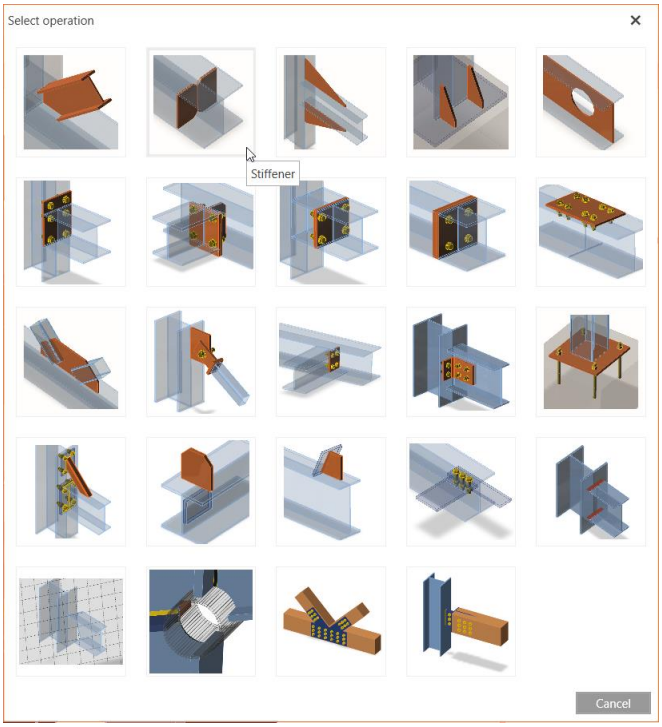


Here are the settings to create the opening.

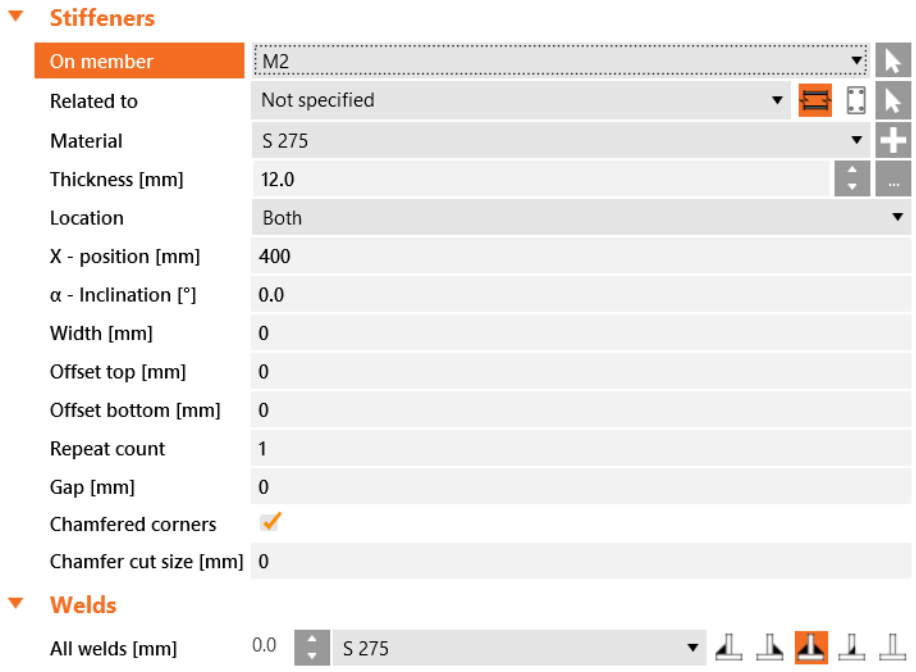


What we need to do next is create the stiffener that is required to reinforce the opening.

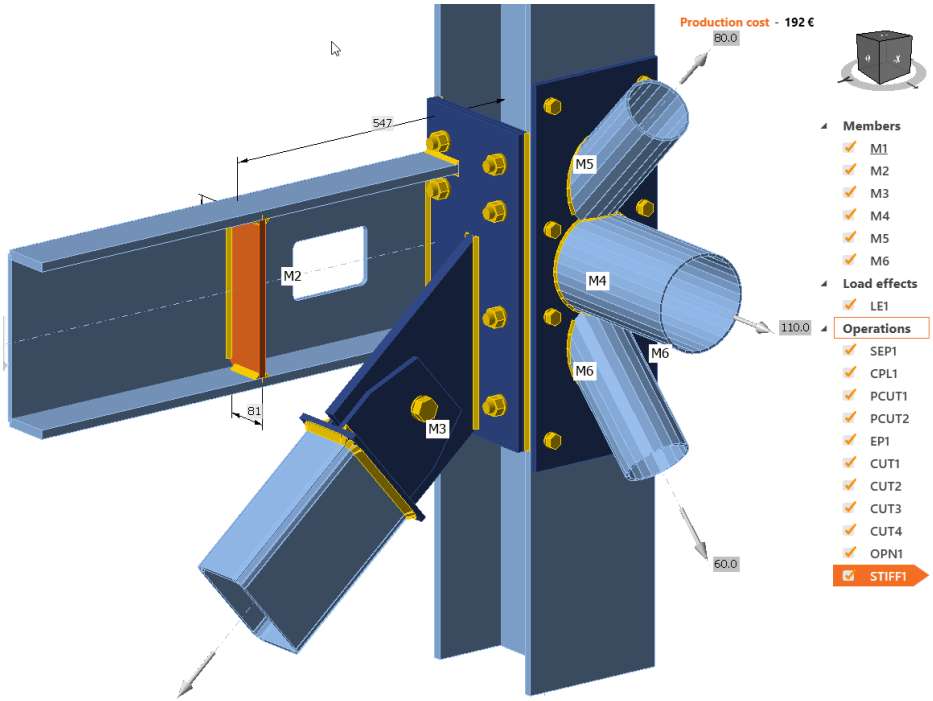
By now, you should know what comes next: we must add a new operation. The operation we are looking for is stiffener.



These are the settings required to create a stiffener on the PFC.



The connection with the completed PFC opening and stiffener now looks like this.

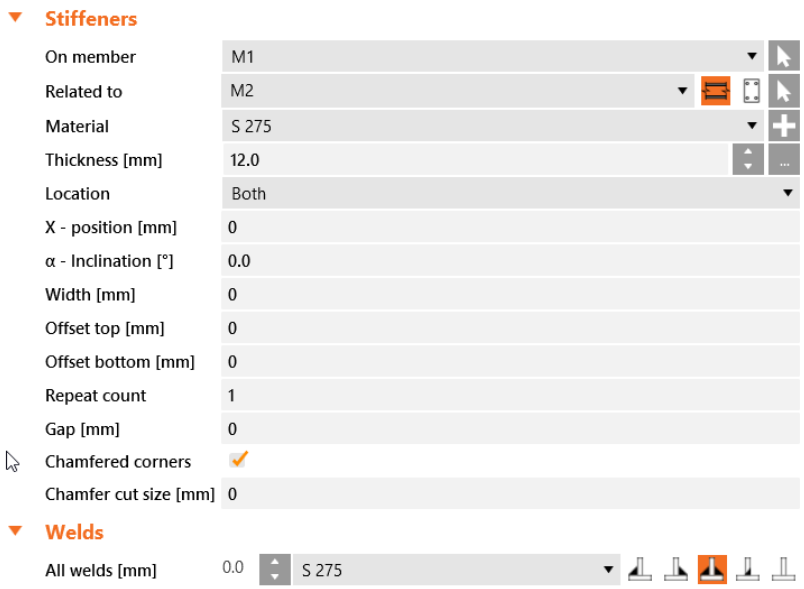


Part 4e – Column web stiffeners

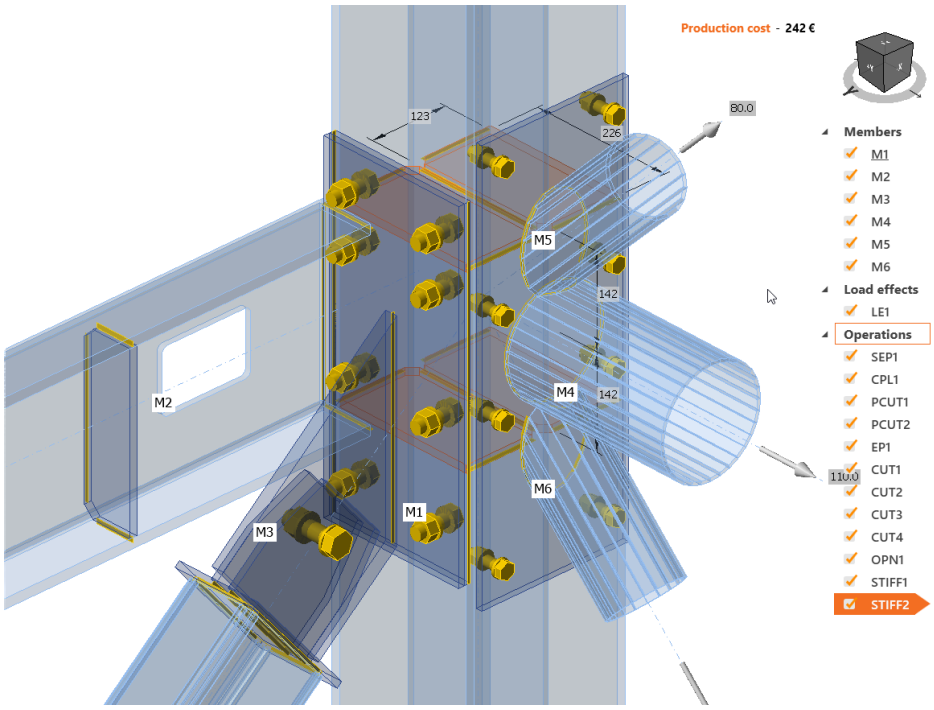
The final part of this connection is the provision of web stiffeners aligned with the flanges of the PFC. You may remember that there was a provision for stiffeners with the shifted endplate operation. This option only created two stiffeners in one location. This operation will create four at the required locations. Please note that this example is provided as an exercise and there might be operations that cannot be completed in fabrication. Always remember that whatever you design should be fabricated. This applies to the correct weld type for the intended location and stiffeners (in this example) that might be difficult to place!

Again, we are looking to create stiffeners. Again, use add operation to do this.

Here are the values we are using.



Here you can see what the stiffeners should look like. Notice that this view uses the transparency option to allow us to visualise the stiffeners.

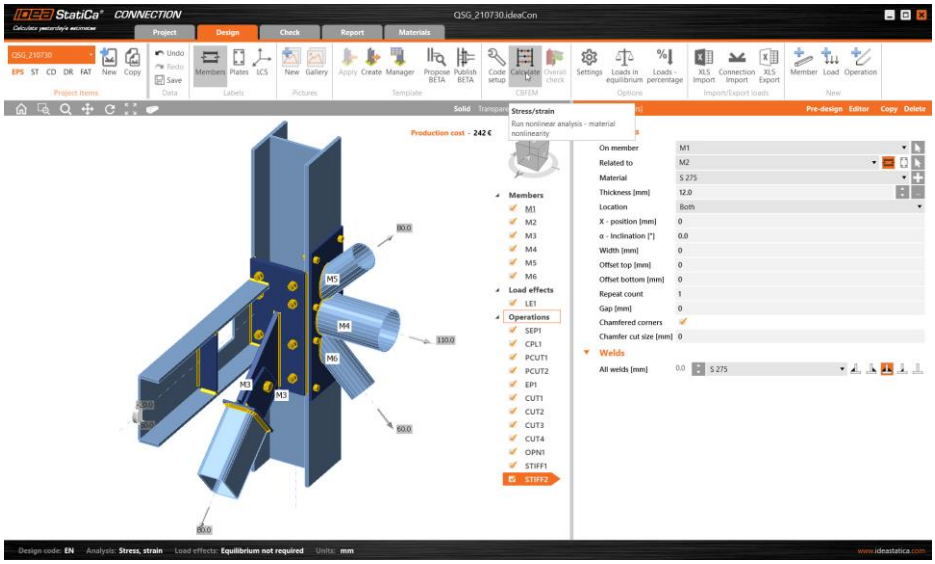


The connection is now complete and the next step is to perform an analysis. Notice that some operations are not in the same order as you would fabricate this item. Hopefully, you can see why this is the case and have gleaned an initial understanding of how a connection goes together.

Part 5 – Analysis and verification

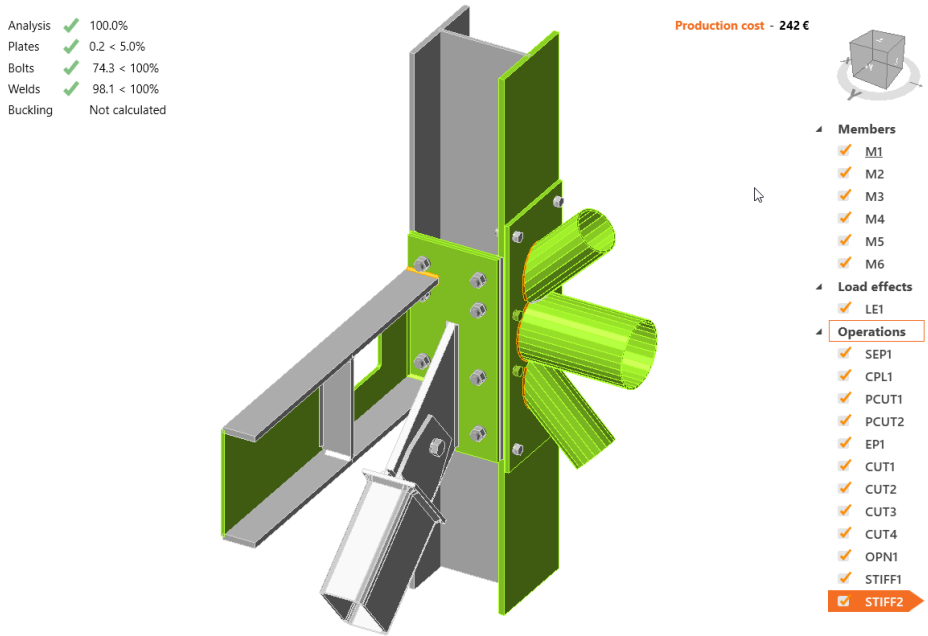
The theory behind IDEA StatiCa Connection is well documented and there are several articles on our website should you wish to delve further. If, however, you are comfortable with this then we can proceed and see how good this joint is.

On the toolbar there is the Calculate icon.



This will perform an analysis based on the current Code setup settings. For this exercise this will be sufficient as our aim is slightly different currently.

These are the initial results.



What does this mean? The calculate option from the Design tab of IDEA StatiCa Connection presents the results in a traffic light format. Grey is under-utilised. Green is ok. Amber is up to 100% and red is over 100%. Needless to say, red is where you have problems! There is also a useful summary in the top right-hand corner of the workspace. In this example we have no red indicators (which is good),

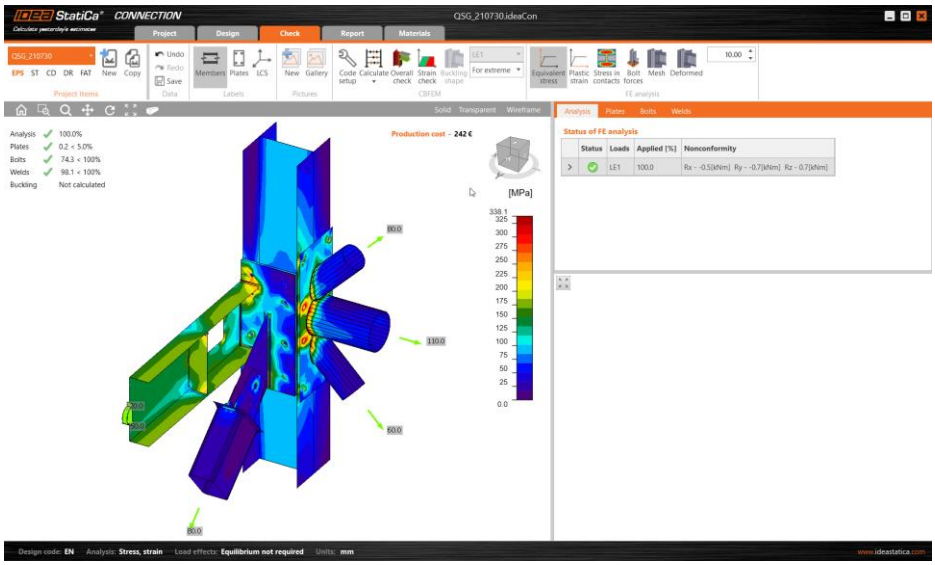
some amber warnings on the welds, lots of green and more grey. There is a school of thought which also treats grey as a warning but a positive one. This shows where a component is over sized or over specified.

If you wanted to change any item in this connection you are free to do so. More on this later. A useful comparison between designs is not only the traffic light results but also the estimated cost of the connection. This is currently standing at €242. These are not meant to be used as definitive costs but as a tool to benchmark one connection against another.

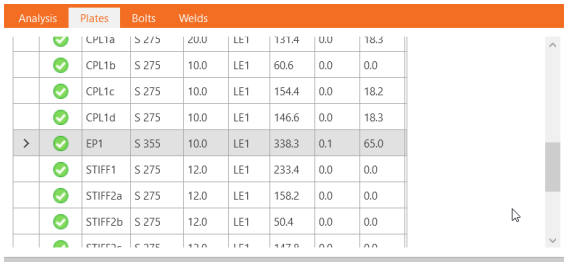
Now that we know that this connection works (and works well) what other results are available to us?

If we move to the Check tab there are more options available.

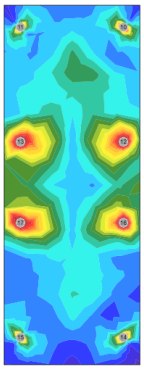
By pressing the Equivalent stress icon we can see the von-mises stress distribution on the connection.



We can also investigate items in isolation by changing the focus of the right panel in the orange bar to say Plates. Here we can look at each plate and the stresses etc in turn.

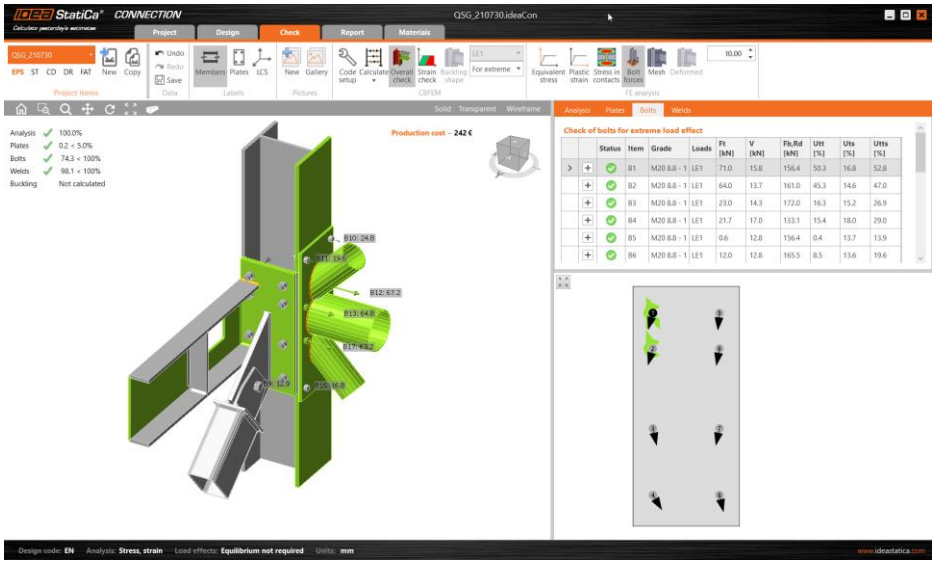


	CPL1b	S 275	10.0	LE1	60.6	0.0	0.0
	CPL1c	S 275	10.0	LE1	154.4	0.0	18.2
	CPL1d	S 275	10.0	LE1	146.6	0.0	18.3
>	EP1	S 355	10.0	LE1	338.3	0.1	65.0
	STIFF1	S 275	12.0	LE1	233.4	0.0	0.0
	STIFF2a	S 275	12.0	LE1	158.2	0.0	0.0
	STIFF2b	S 275	12.0	LE1	50.4	0.0	0.0



If we turn our attention now to the bolts… According to the summary these are operating at just over 75% and several are under-utilized. There is scope for adjustment to either reduce the size or number of bolts to increase their efficiency (or decrease it depending on your perspective).

Turning the Overall check back on and selecting Bolt forces in the main toolbar and Bolts in the orange banner we get a good picture on which bolt is working well and to what level.



Here we can see the bolt forces in the workspace and the calculations related to each bolt in the right-hand panel. The resultant forces are shown pictorially as well.

By expanding the line using the + button the actual code-based checks are displayed.

Check of bolts for extreme load effect

		Status	Item	Grade	Loads	Ft [kN]	V [kN]	Fb,Rd [kN]	Utt [%]	Uts [%]	Utts [%]
	−	✓	B1	M20 8.8 - 1	LE1	69.8	16.7	156.4	49.5	17.8	53.1

Tension resistance check (EN 1993-1-8 tab 3.4)

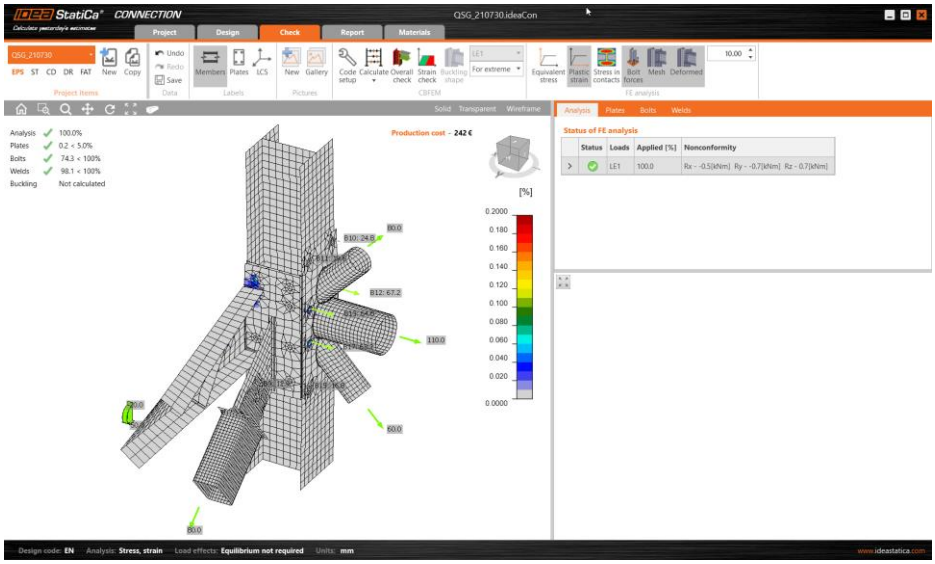
$$F_{t,Rd} = \frac{k_2 f_{ub} A_s}{\gamma_{M2}} = 141.1 \text{ kN} \geq F_t = 69.8 \text{ kN}$$

where:

$k_2 = 0.90$ – Factor

$f_{ub} = 800.0$ MPa – Ultimate tensile strength of the bolt

We can even examine the expected deformations in the joint under plastic strain conditions.

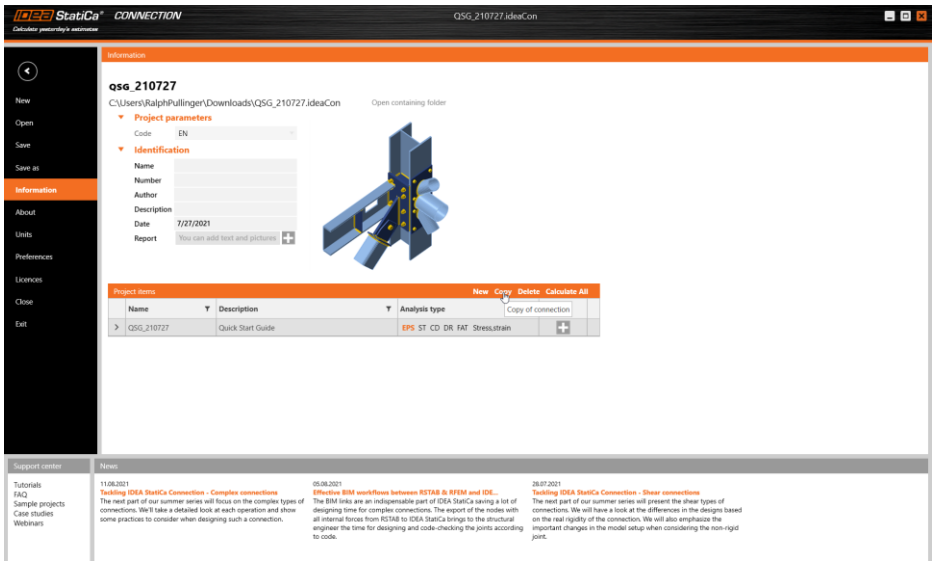


We can use this option to get feedback to see if the joint is behaving generally as expected. The additional checks might direct us to examine the strains around holes in plates for instance or we might want to control the stresses in the webs of members. All is possible to arrive at a sensible overall joint design. There are many tutorials on our web site that will take you through many different types of analysis and verifications.

With the Report tab we can generate documents that can be submitted to a checking engineer or kept for records. The Materials tab gives us a list of what is being used in the current projects.

Before we close and leave you to experiment it would be best if we describe to you how you can keep this connection as one that works and show you how to create a copy on which to experiment.

On the Project tab you are able to Copy the highlighted connection to another connection within the same file. This allows to keep different options or keep several related connections together in one project related file.



This quick start guide has been produced to enable you to give you a great start in your IDEA StatiCa Connection journey. We hope you have found it useful. Should you have any comments please contact us via the customer portal.

Further Reading

Like many solutions to get proficient you need to practise. In addition to the access you have to the Customer Portal where you can ask questions and raise cases etc there is also a list of handy tutorials and features that will be useful.

Bolted plate to plate

Footing with diagonal brace

Stiffness analysis

Buckling analysis

Seismic design

Ultimate joint capacity

General 3D joint

3D tubular joint

IDEA StatiCa®

Calculate yesterday's estimates