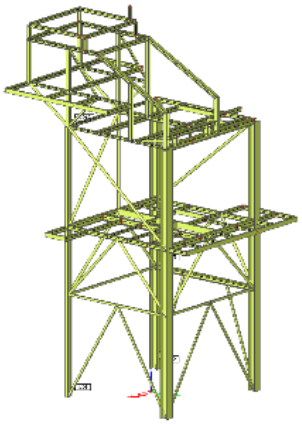
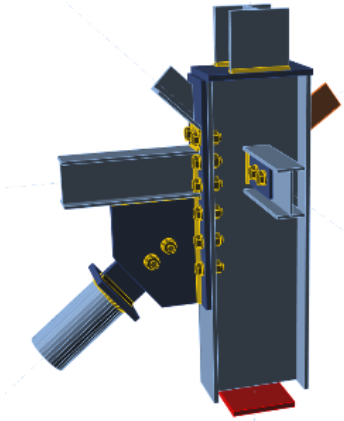
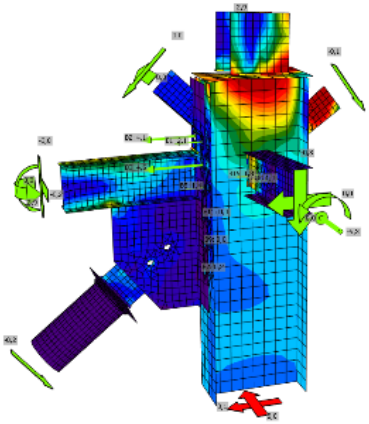
IDEA StatiCa Tutorial – SCIA Engineer link

Welcome to IDEA StatiCa tutorial. By following it step-by-step, you will learn how to design and code-check a structural steel joint using IDEA StatiCa Connection.

In this tutorial we will demonstrate how to use the link between SCIA Engineer and IDEA StatiCa Connection.

Activate the link Use the link Update the project Known limitations

1 How to activate the link

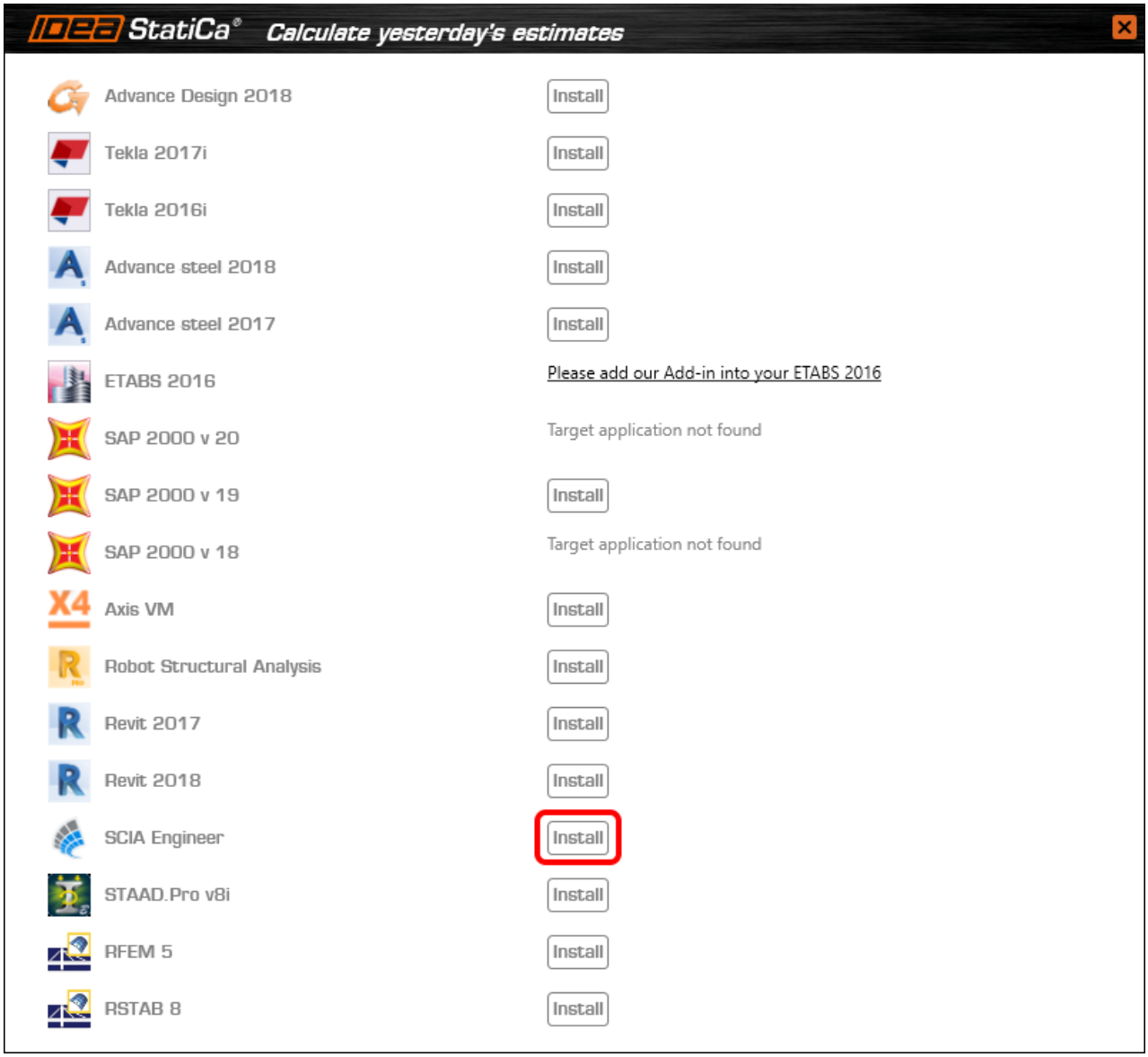
Install the latest version of IDEA StatiCa, get it in the **Downloads**.

Make sure you are using a supported version of SCIA Engineer – updates are published in the **BIM section.**

IDEA StatiCa automatically integrates the BIM link into your CAD/CAE software during its installation. You can check the status and activate more BIM links for later installed software in the BIM link installer.

Open IDEA StatiCa and navigate to the panel **BIM** and open the **BIM link installer**. A notification "*Run as administrator"* may appear, please confirm with the **Yes** button.

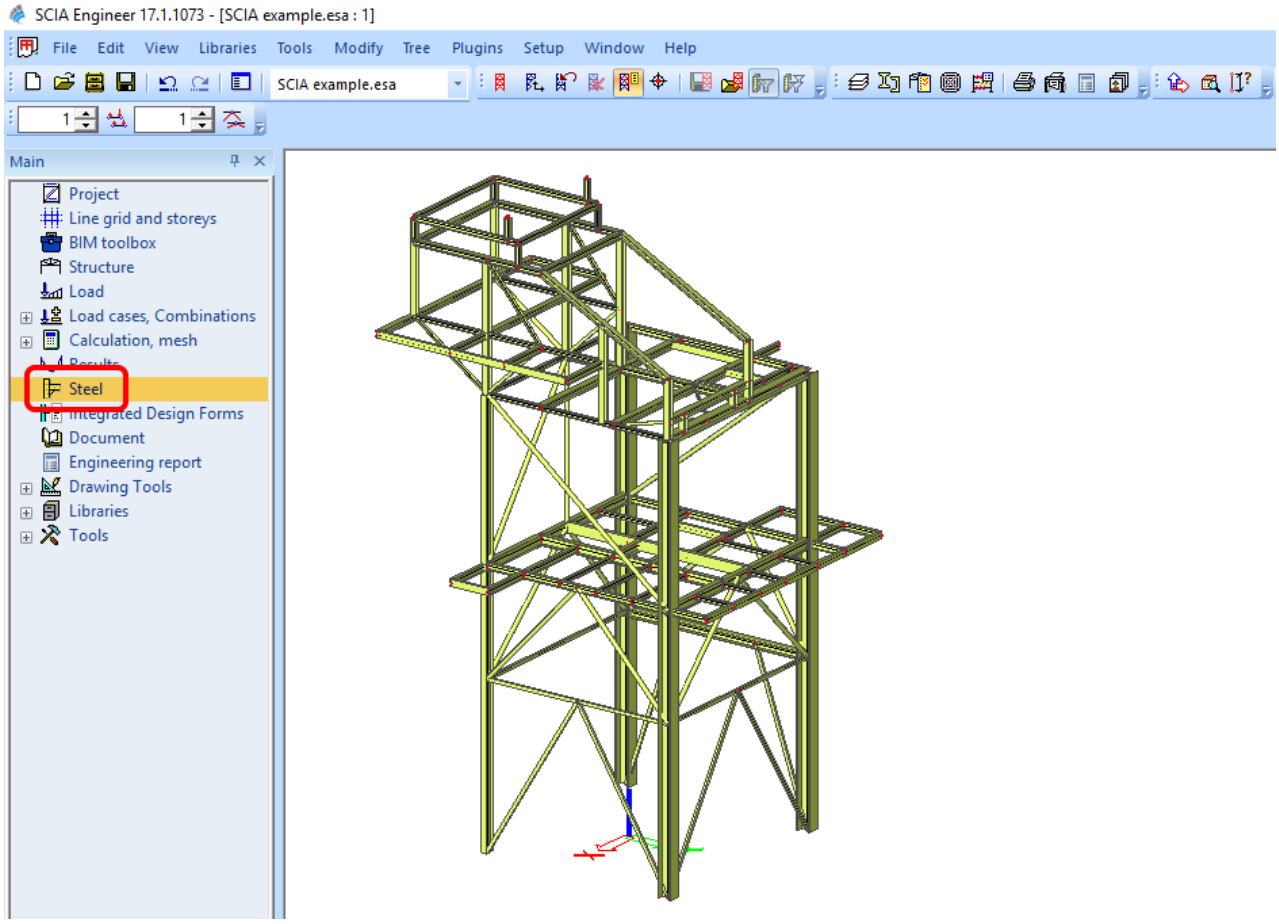
Select the software to integrate the IDEA StatiCa BIM link, click the **Install** button and check the Installed status.



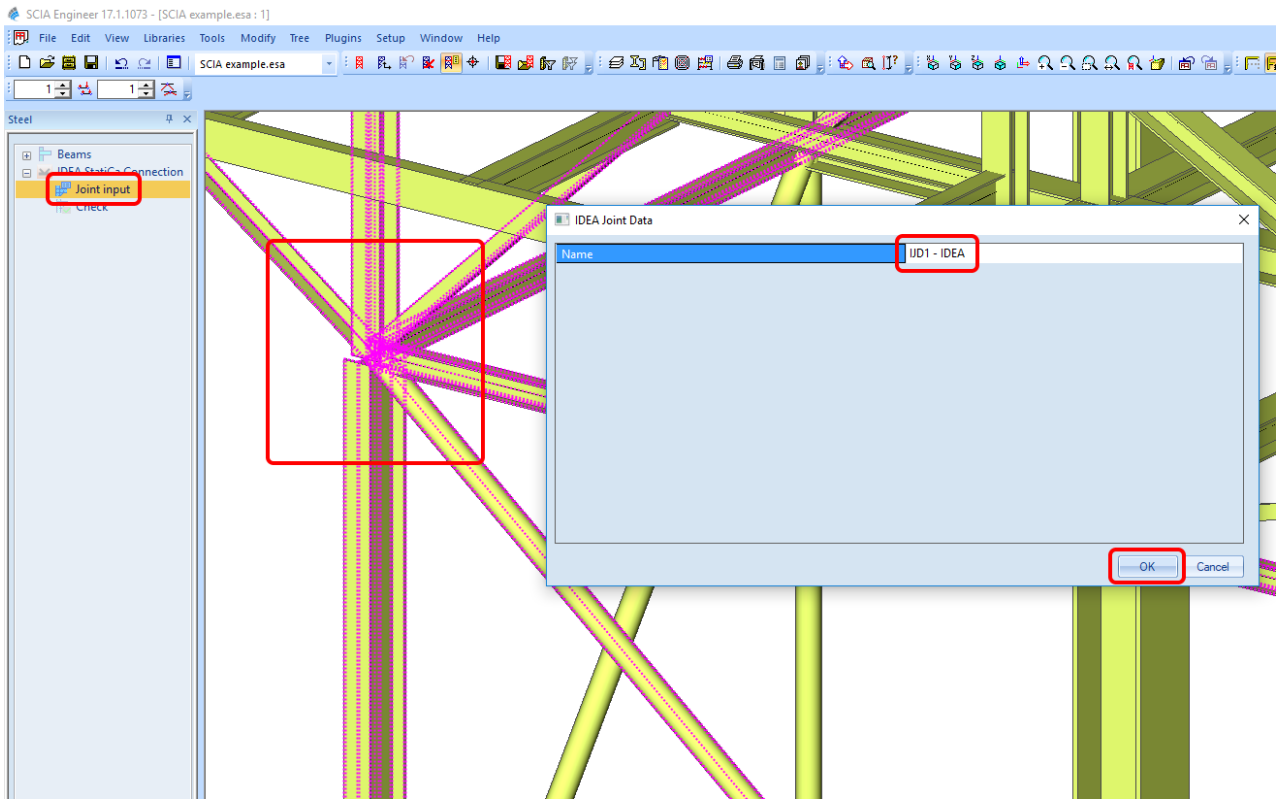
2 How to use the link

First, we download the source file SCIA example.esa, open it in SCIA Engineer and run the linear analysis to get the internal forces over the structure.

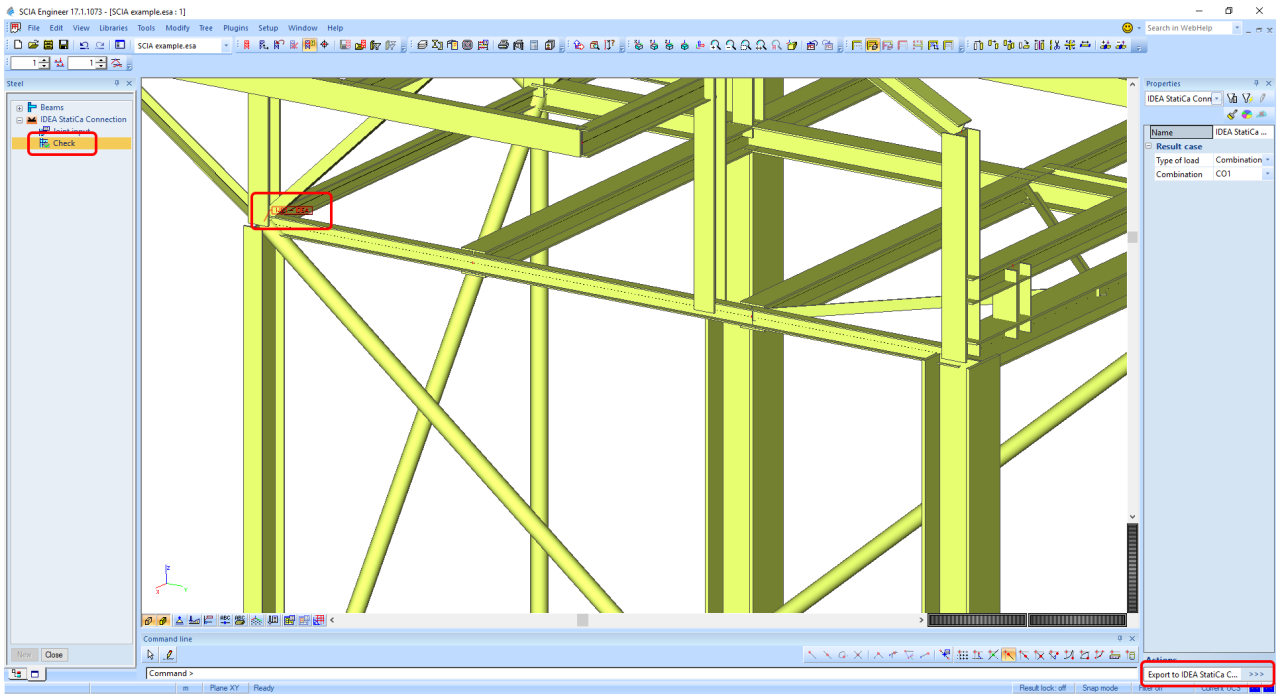
In the **Main menu** we go to **Steel** tab.



Here we go to the menu **IDEA StatiCa Connection**. First, we add a joint to this SCIA Engineer project list of joints by selecting the members of a joint, then by clicking on **Joint input** and typing the name of it (name of the IDEA StatiCa Connection project file) and **OK.**



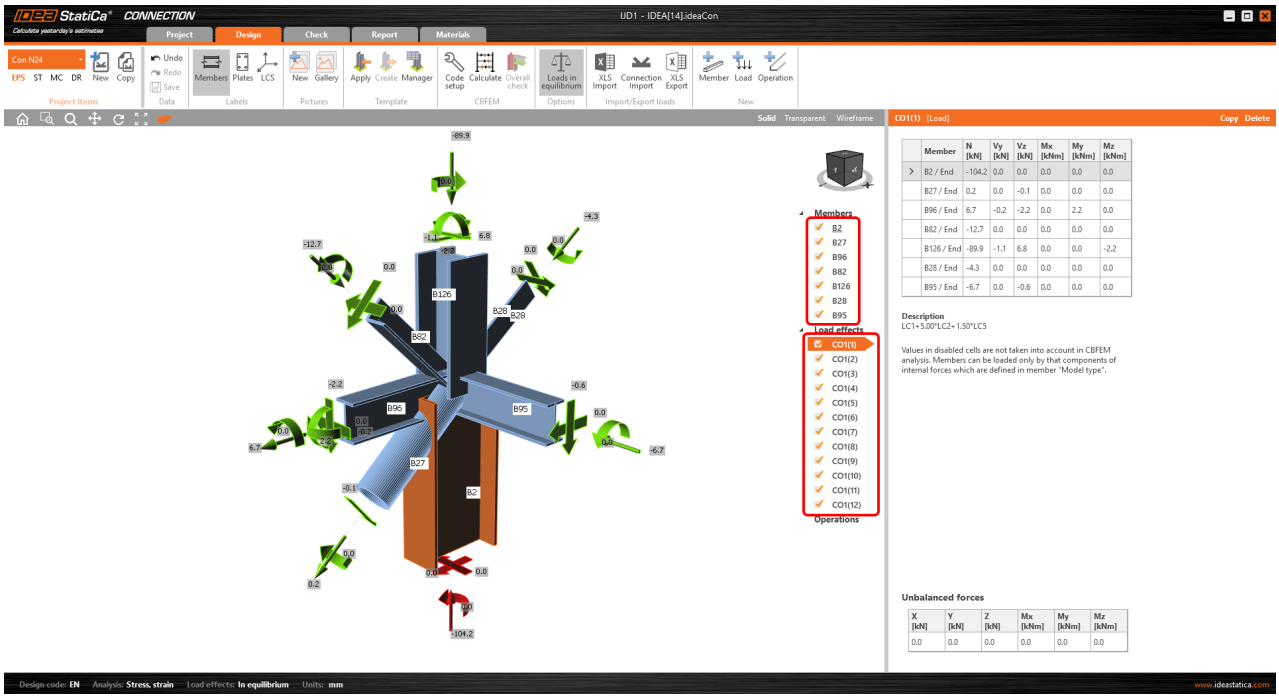
Then we open the command **Check**, run the command **Export to IDEA Statica Connection** in the Actions menu and select the joint sign in the workspace.



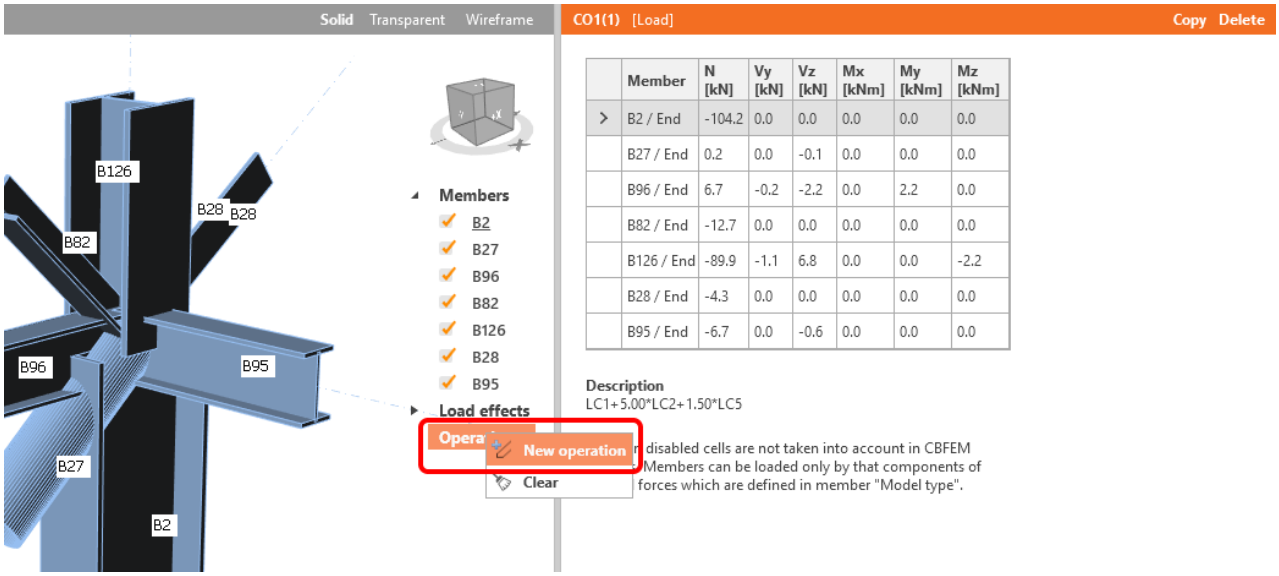
Application IDEA StatiCa Connection opens and we can design connections of the joint.

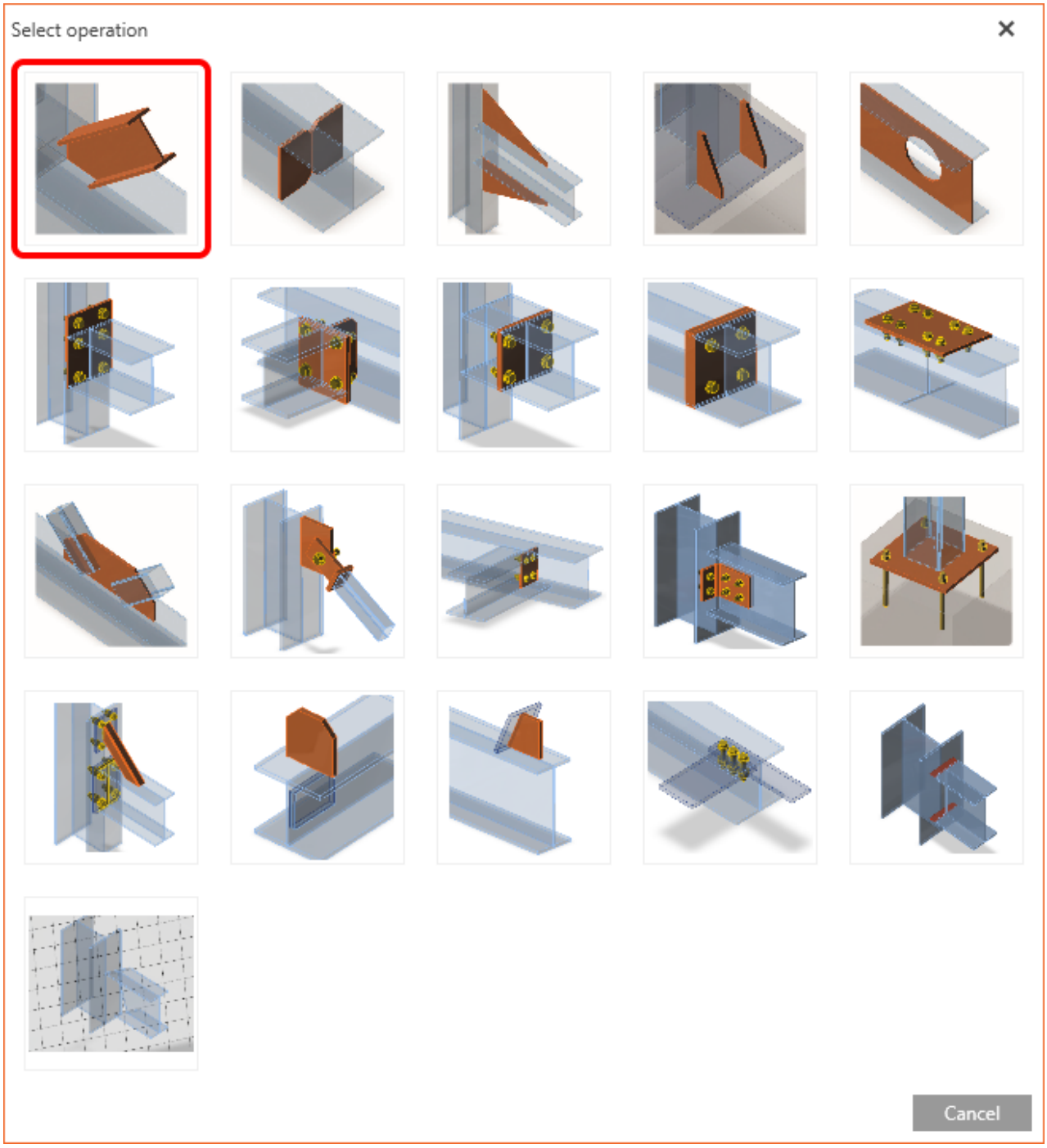
3 Design

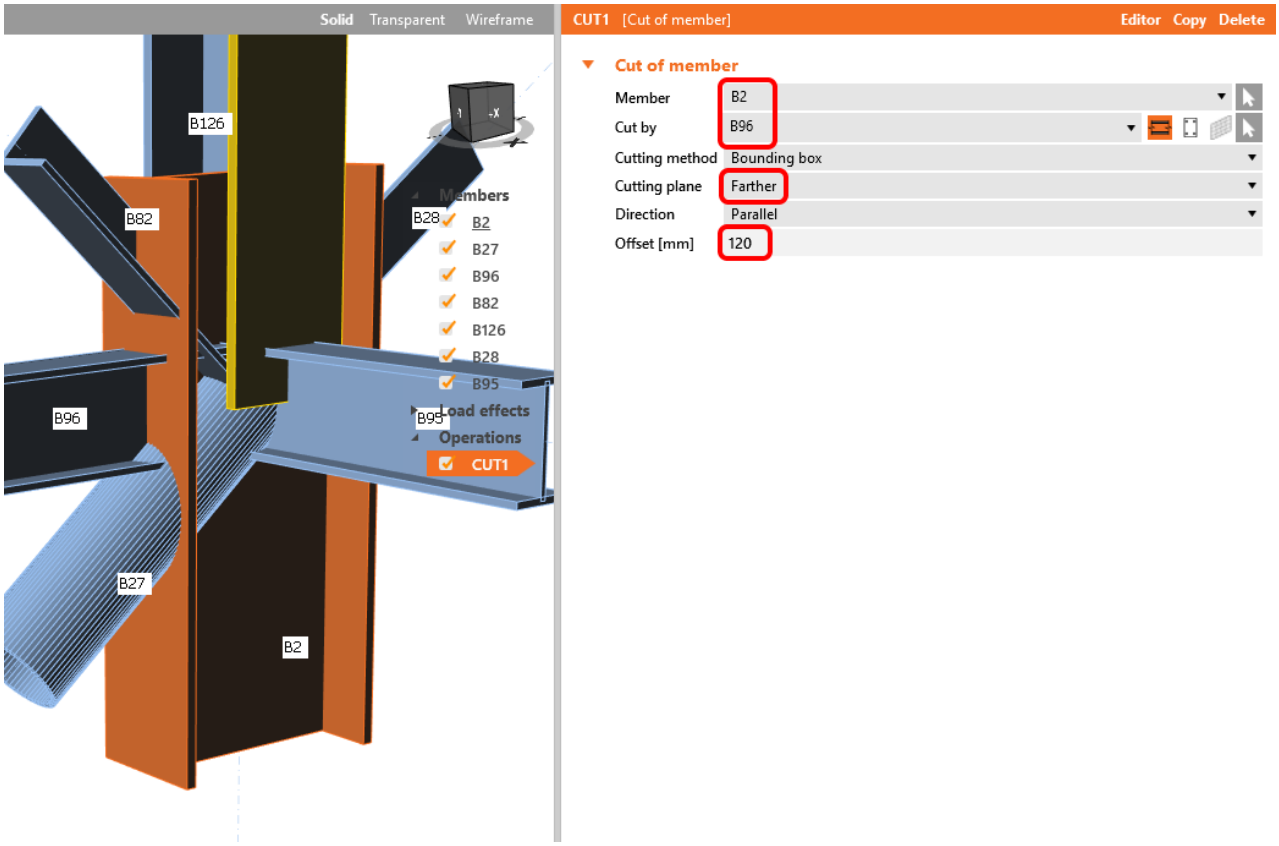
Joint geometry and load effects have been imported, we can start designing.



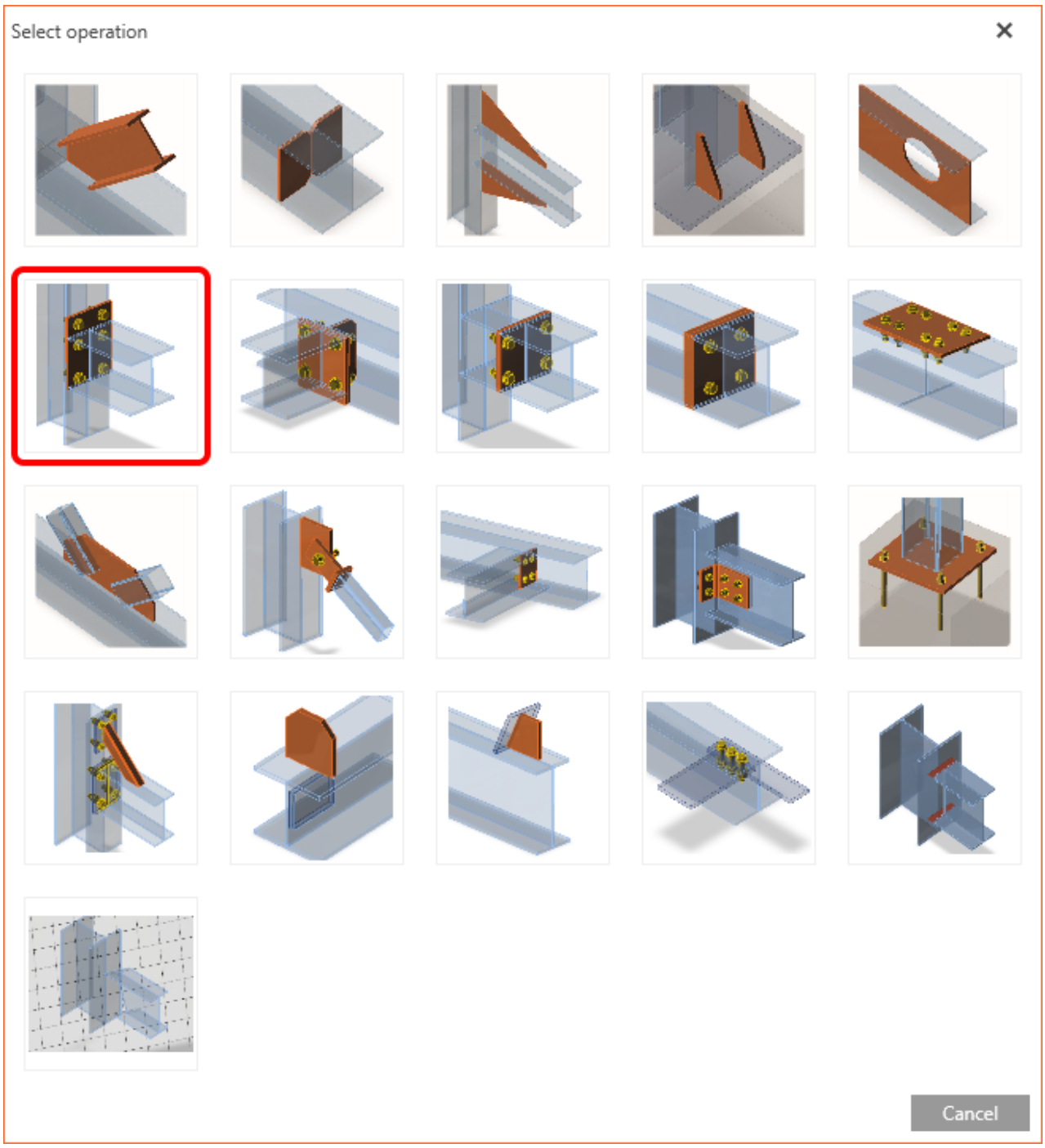
We will define a set of manufacturing operations to model connections between members. Let's hide the Load effects tree and right-click on **Operations** to add a new operation **Cut**.

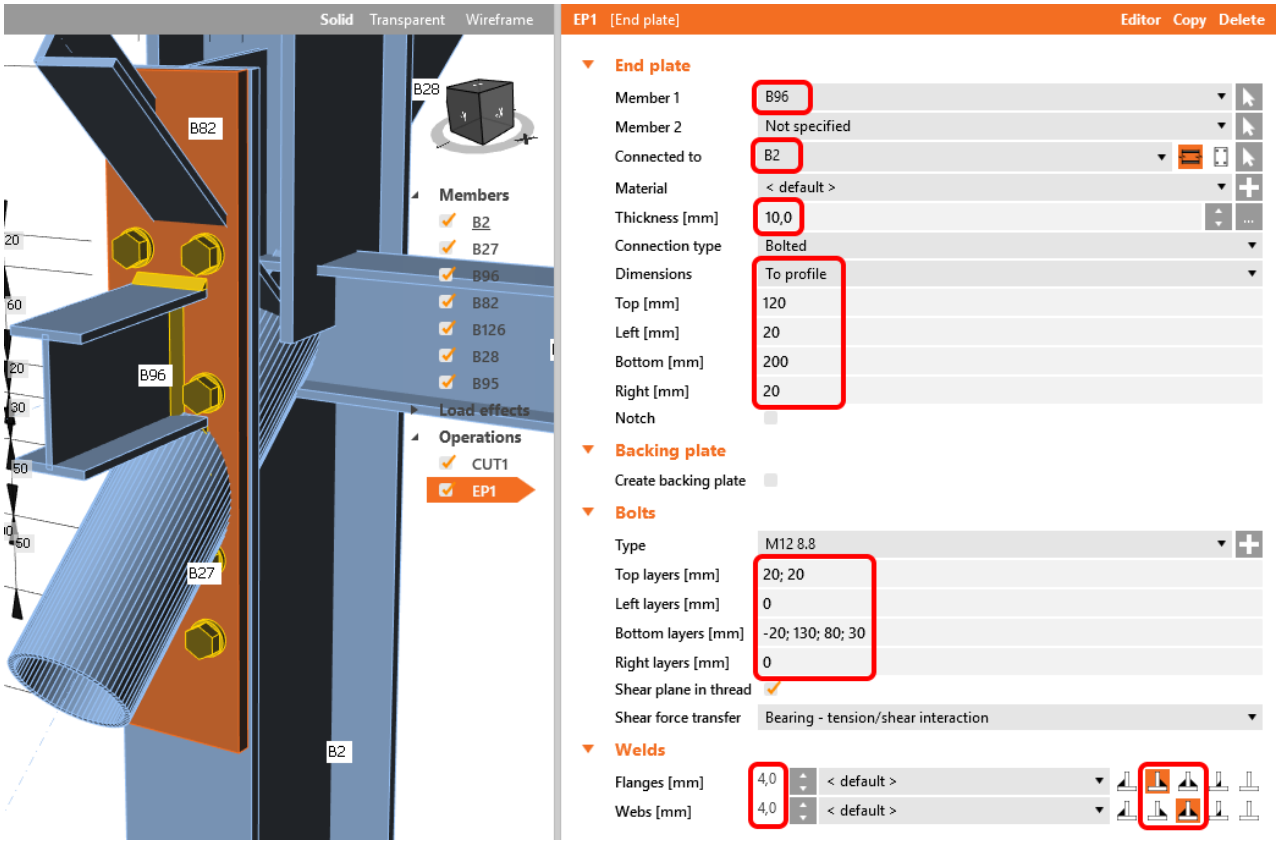



Select operation
Cancel

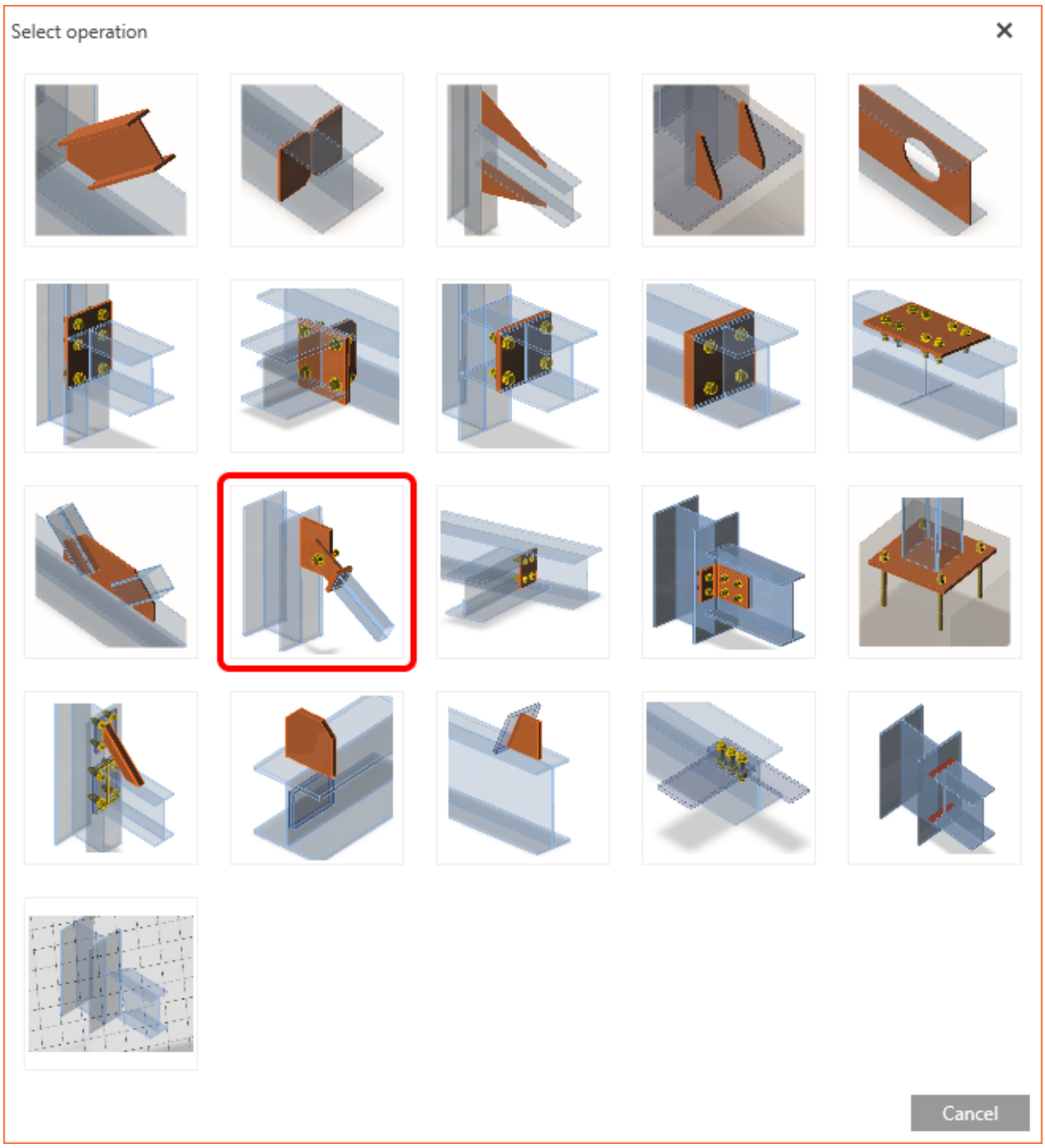


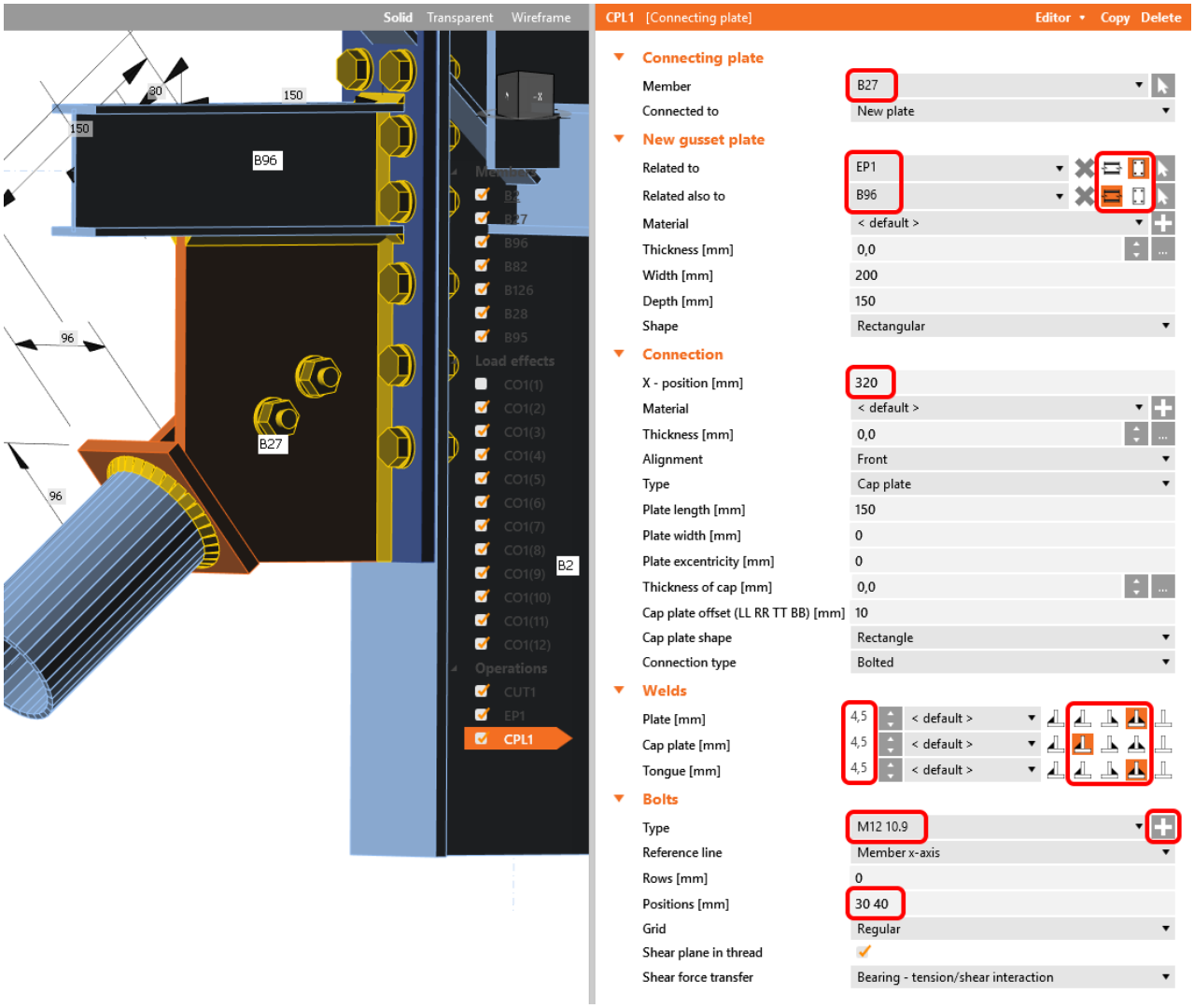
In the next step we add another manufacturing operation **End plate** and modify its parameters.


Select operation
Cancel

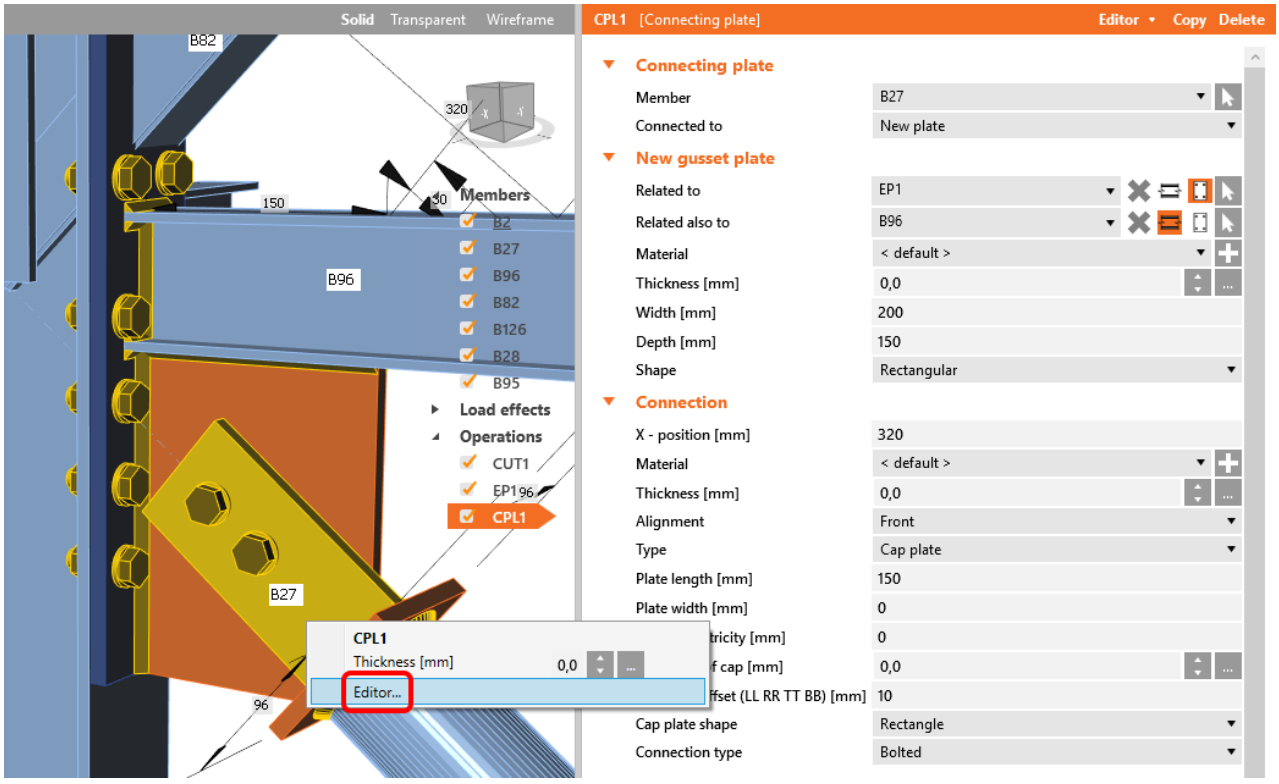


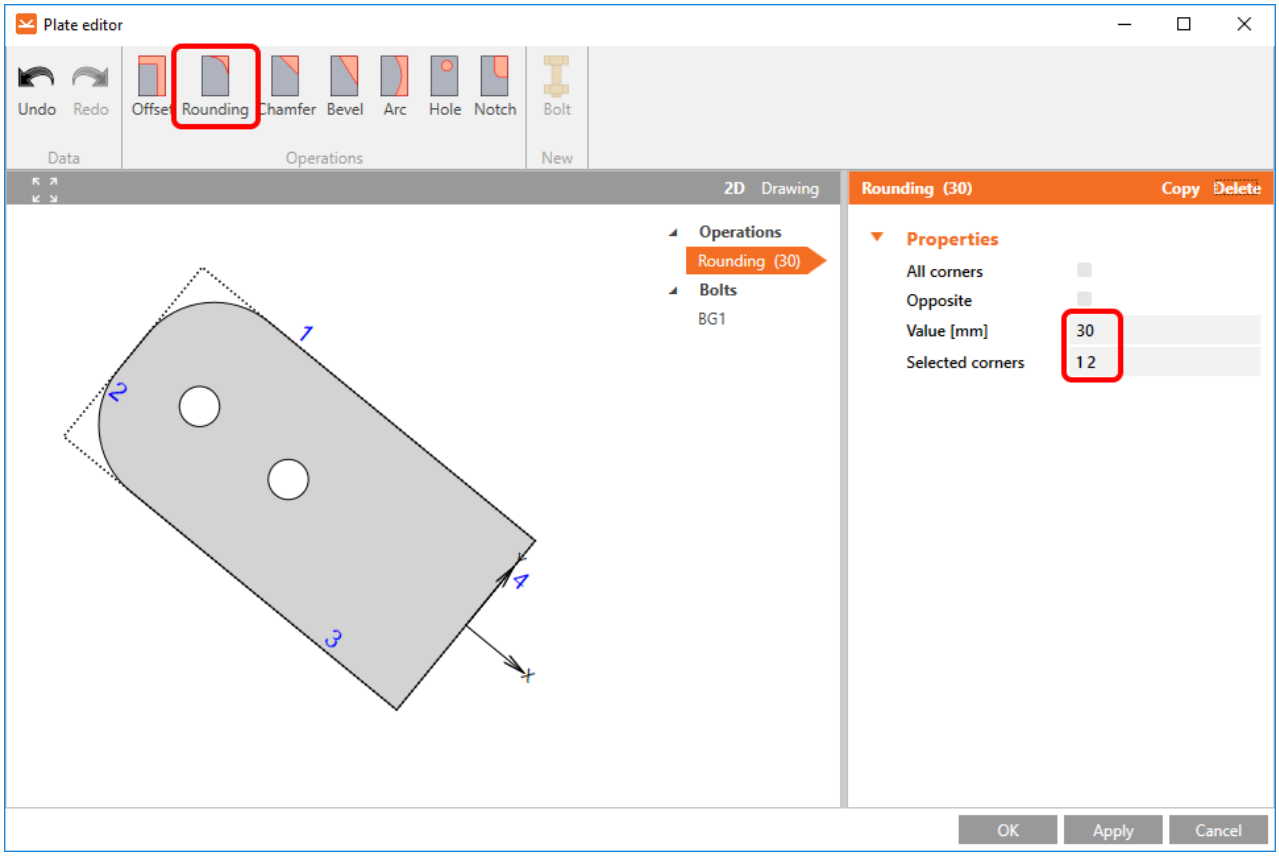
Then we add a **Connecting plate** operation and set its parameters for the appropriate member.


Select operation
Cancel

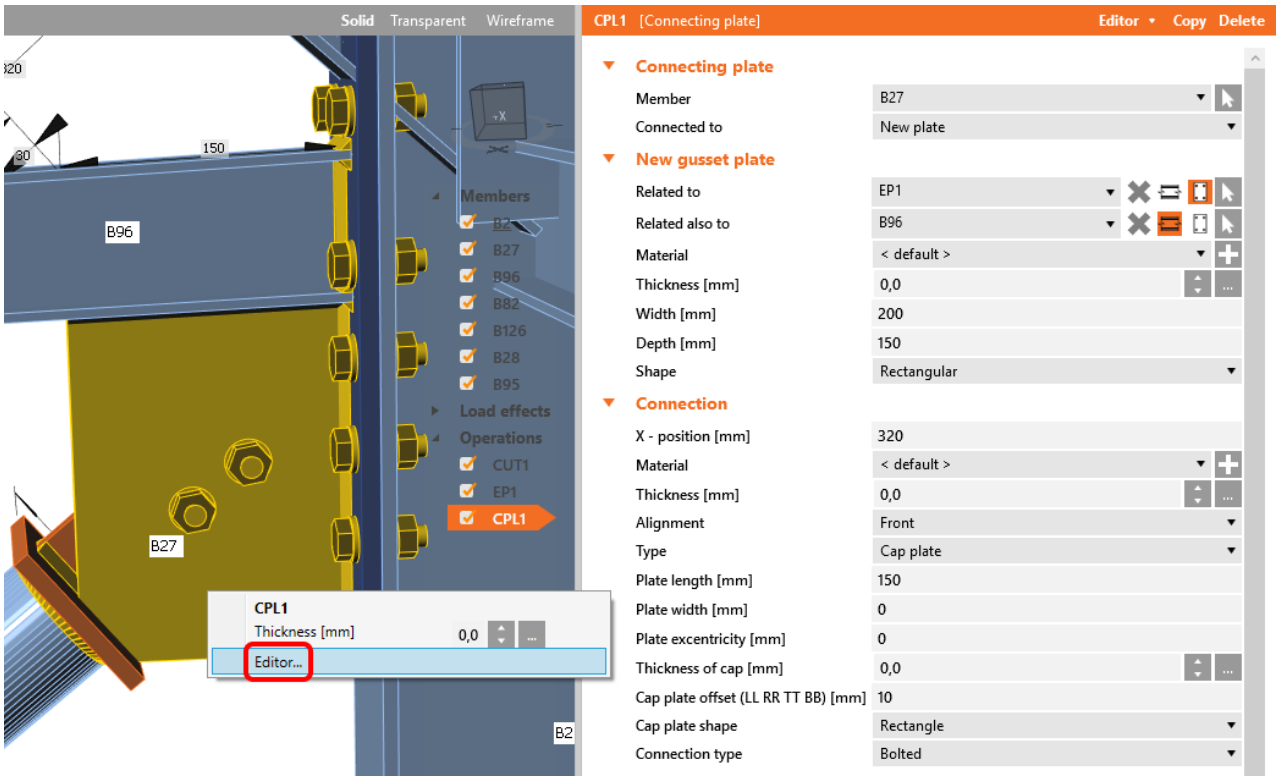


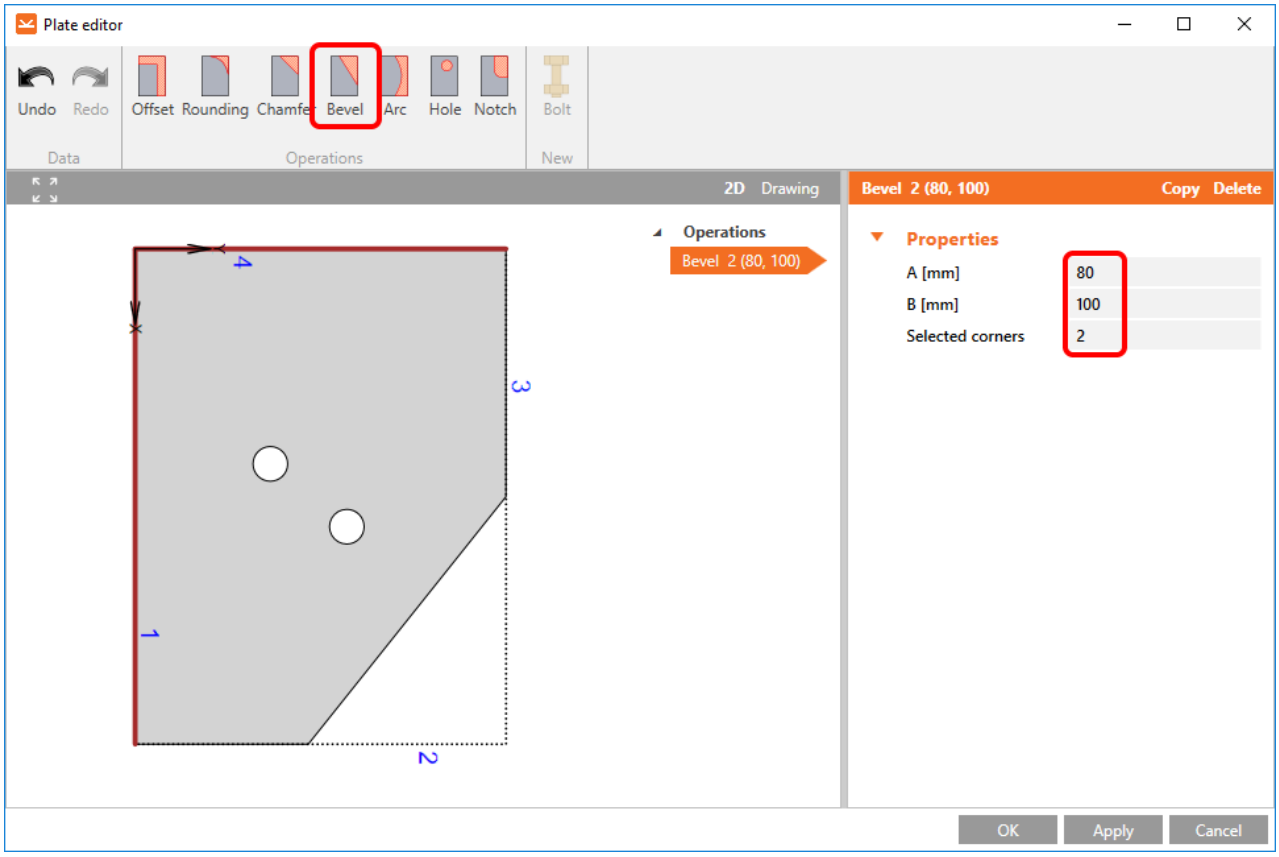
We right-click on the tongue plate, open the plate **Editor** and reshape the **Tongue** by adding an operation **Rounding.**



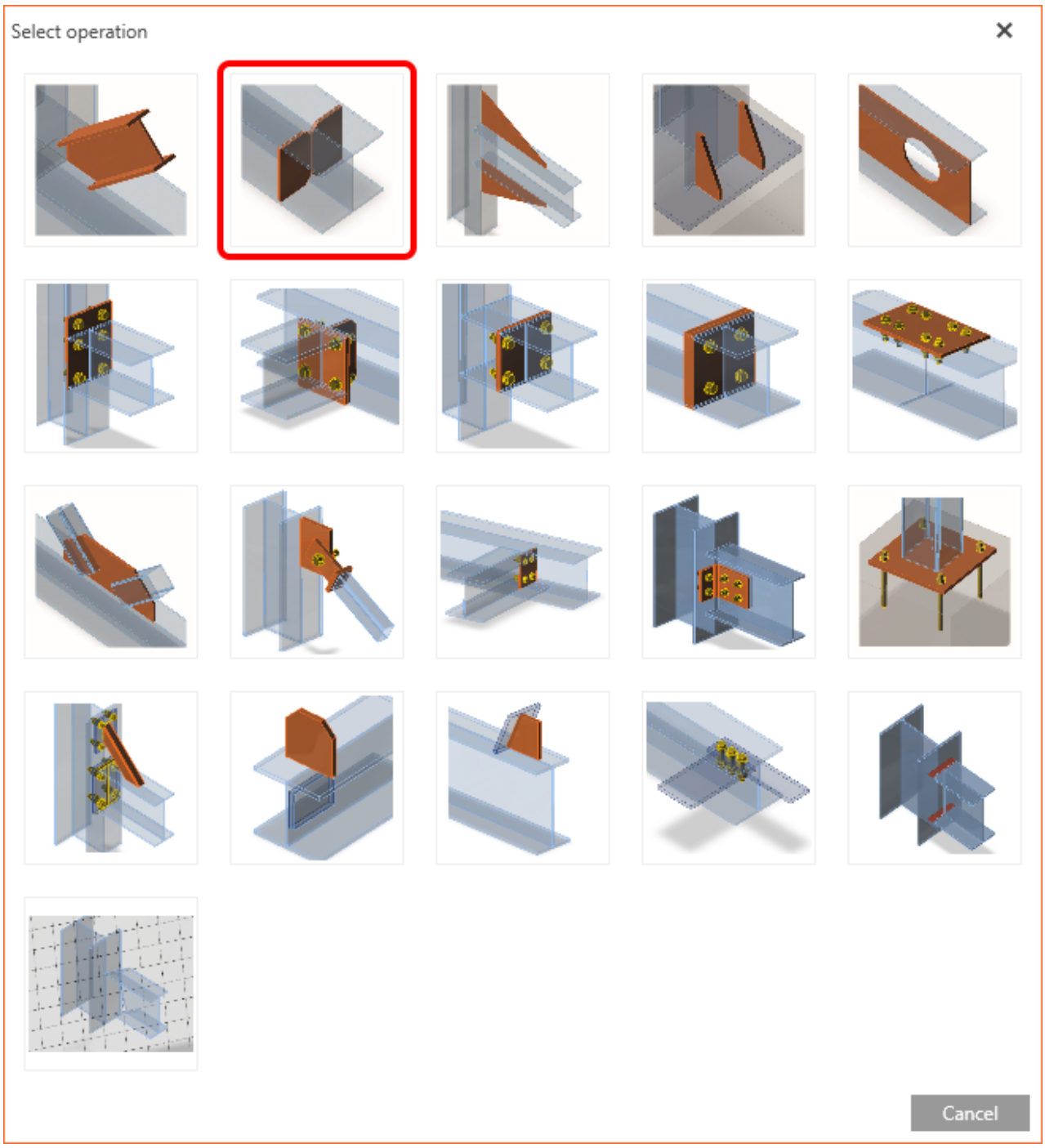


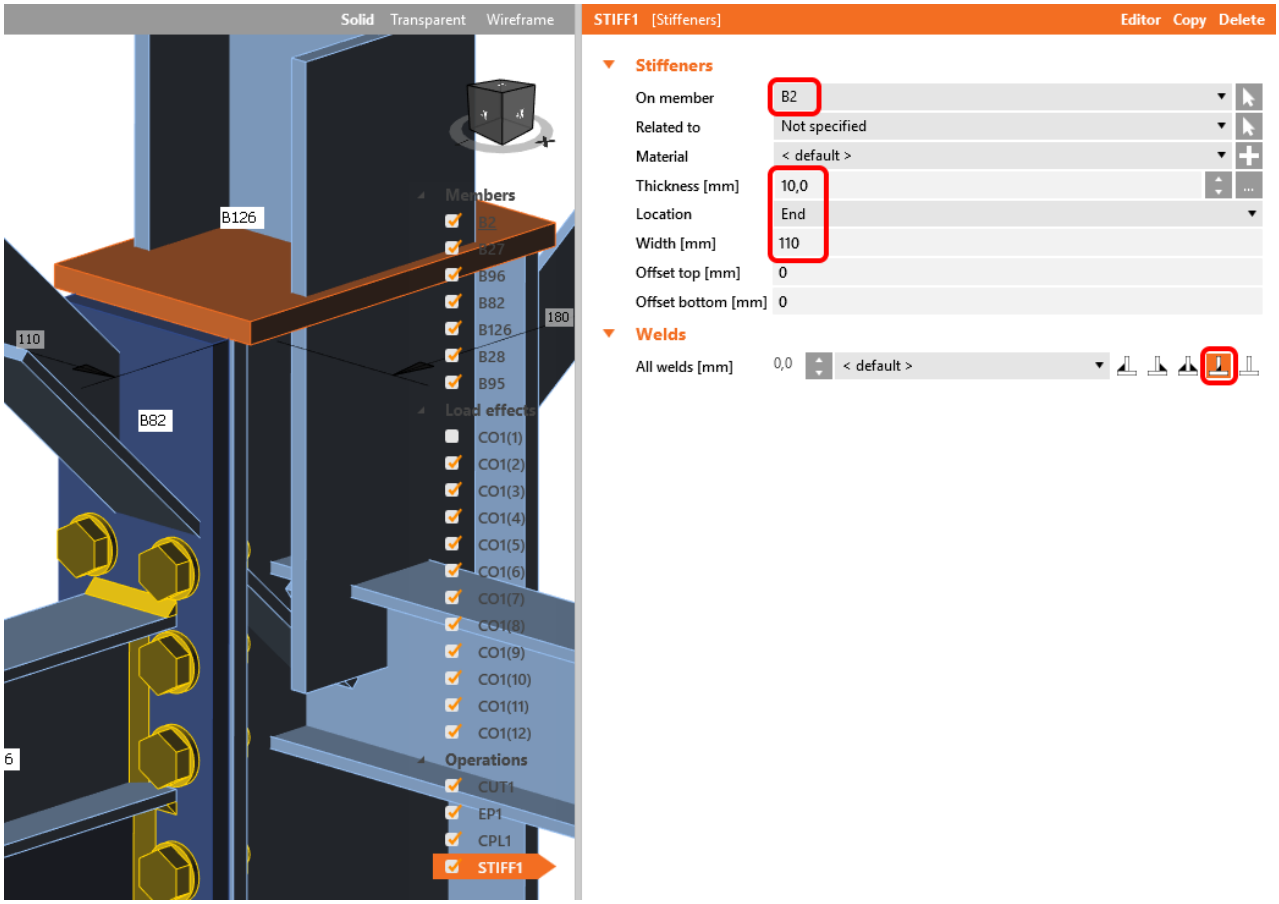
Next, we again right-click on the gusset plate and in the **Editor** reshape the **Gusset** plate by adding a **Bevel** operation.



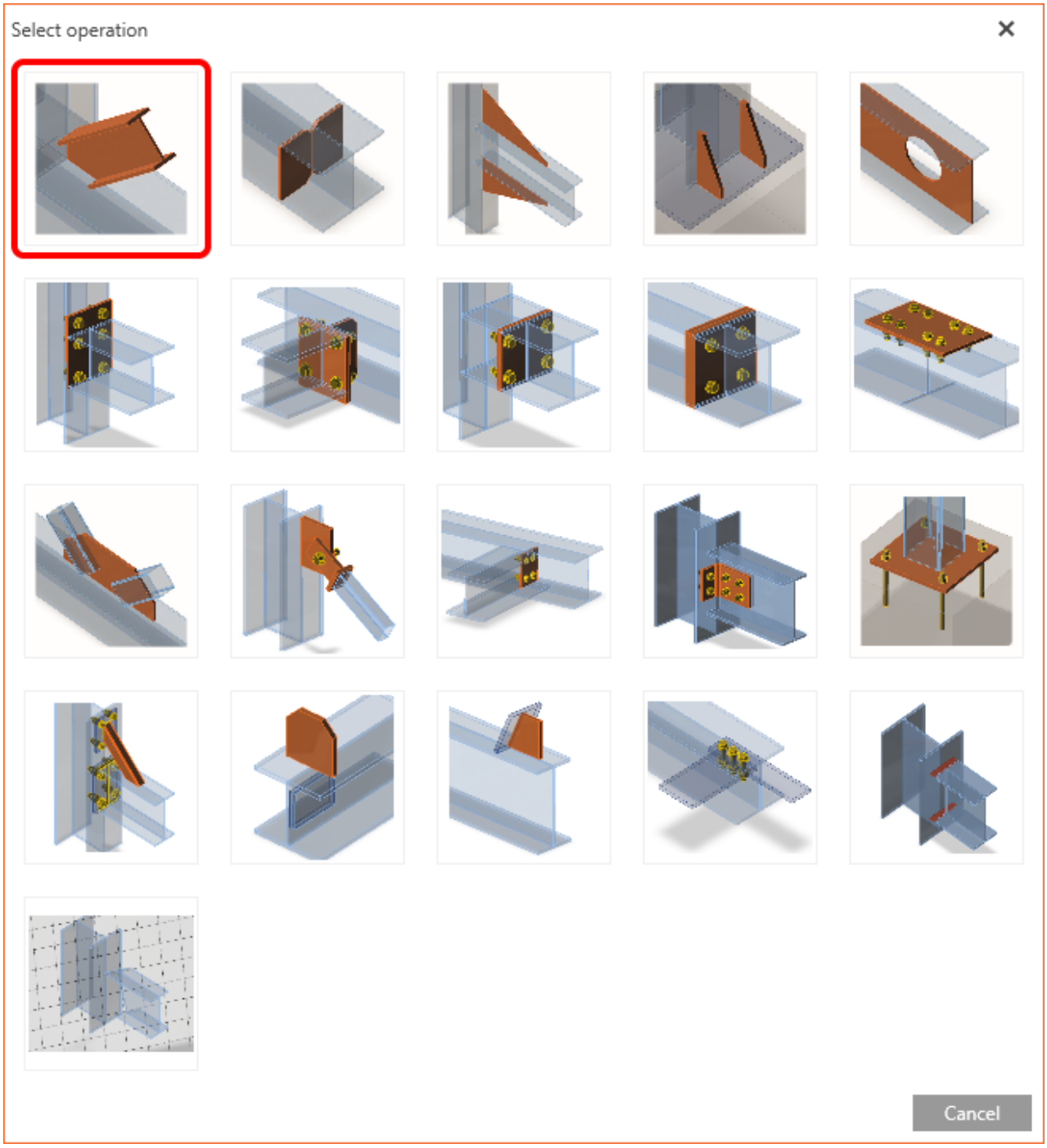


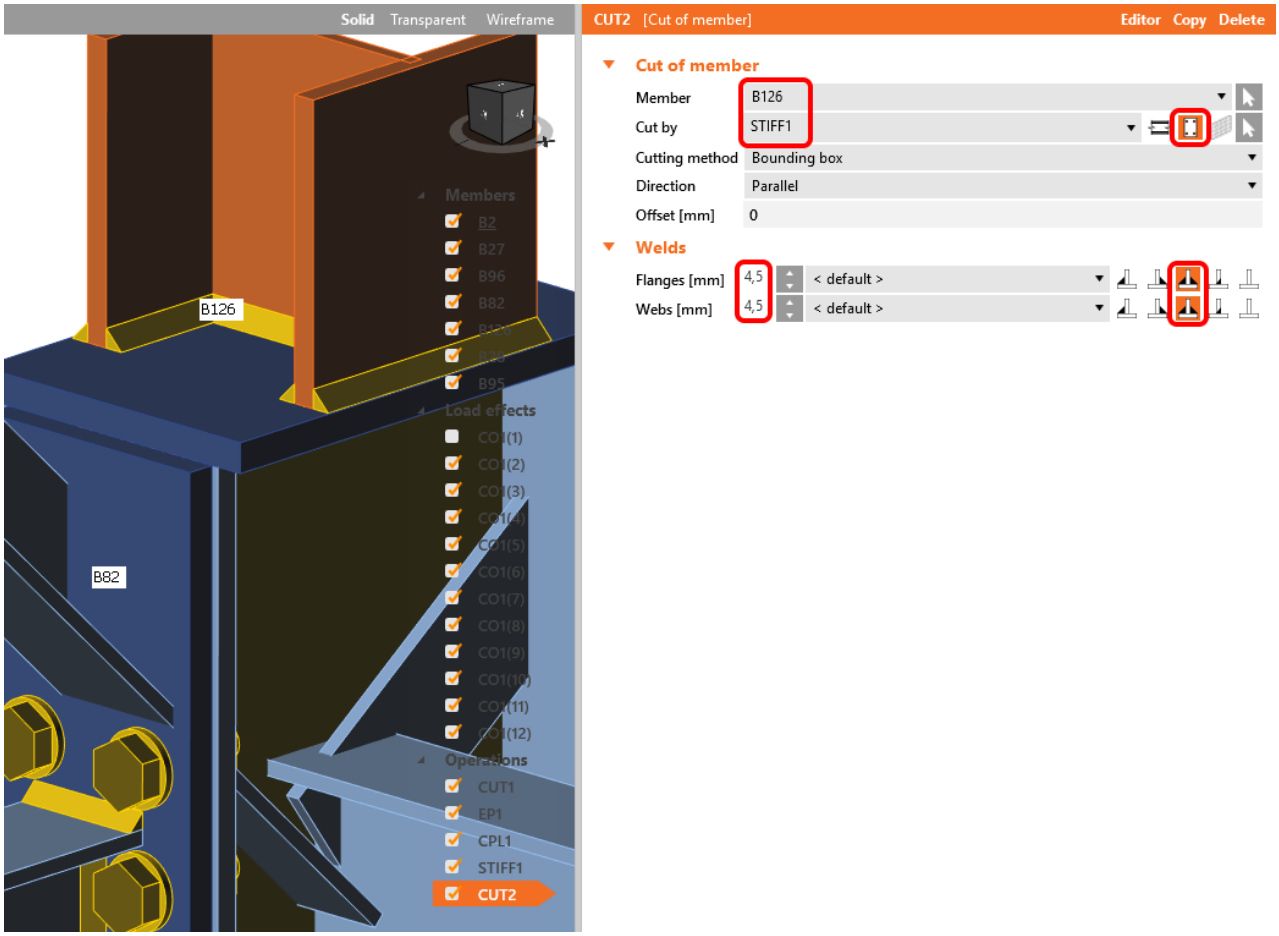
Now we add a **Stiffener** operation and modify its properties.


Select operation
Cancel

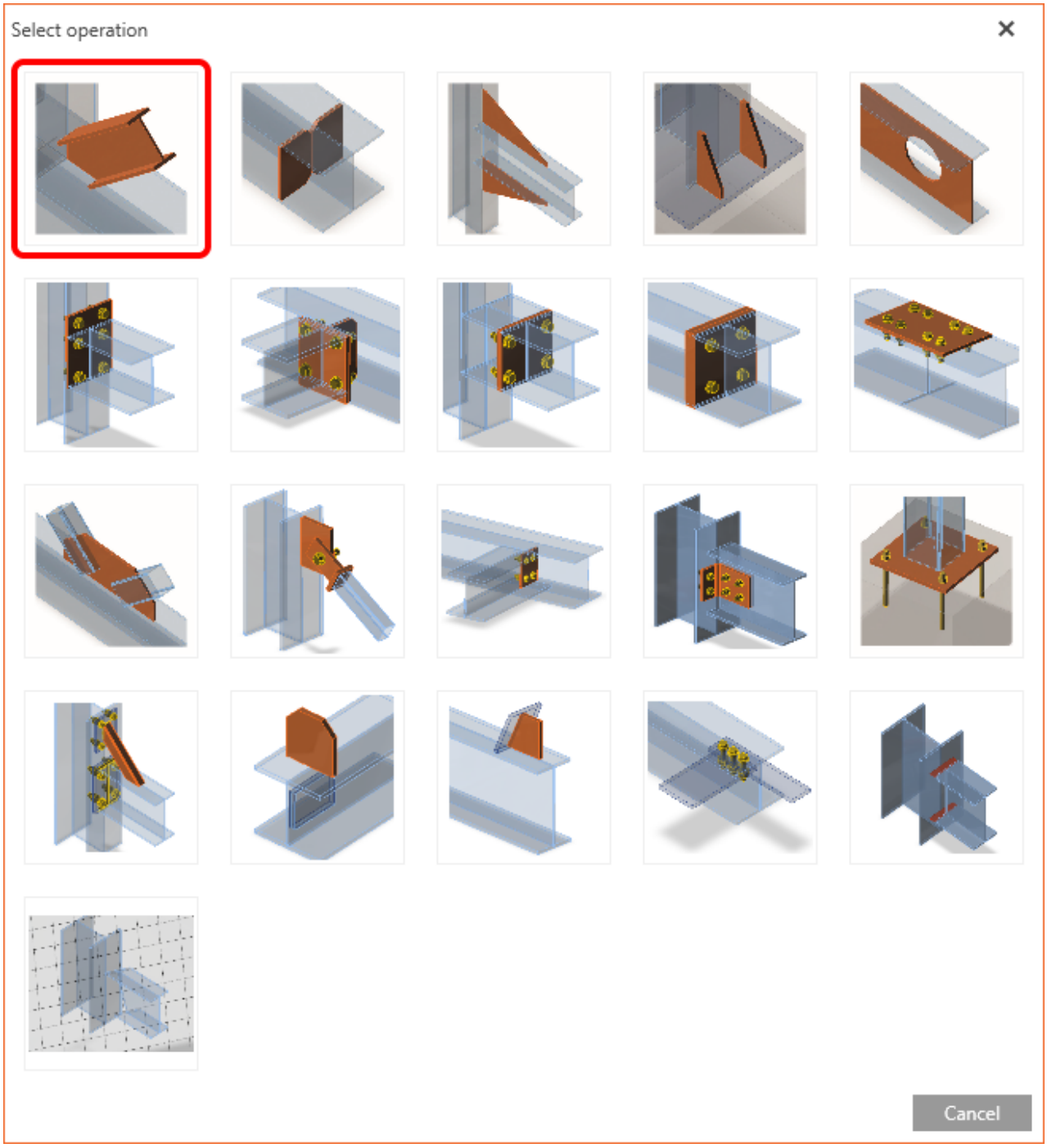


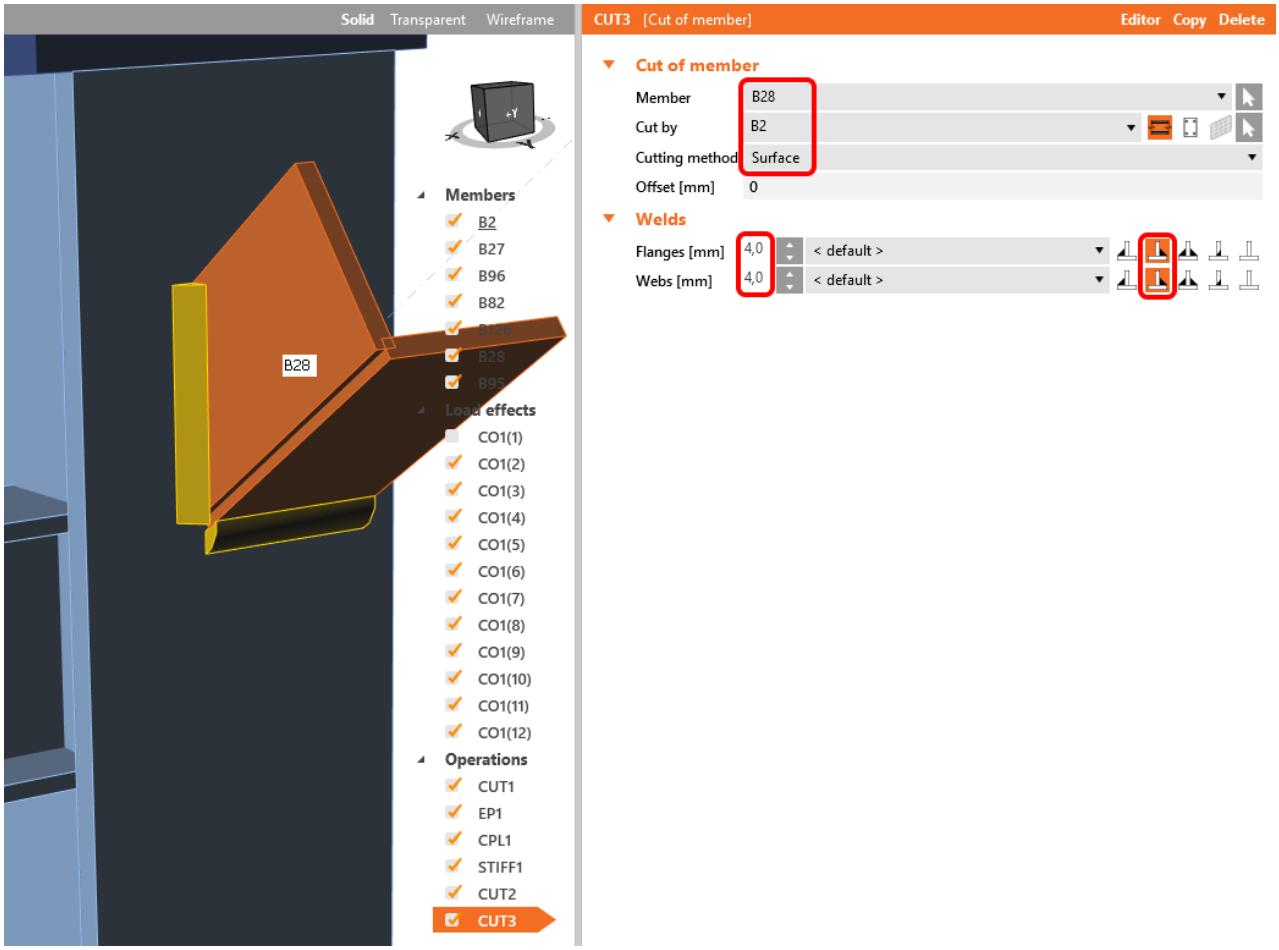
Then we add another **Cut** operation for the upper member.


Select operation
Cancel

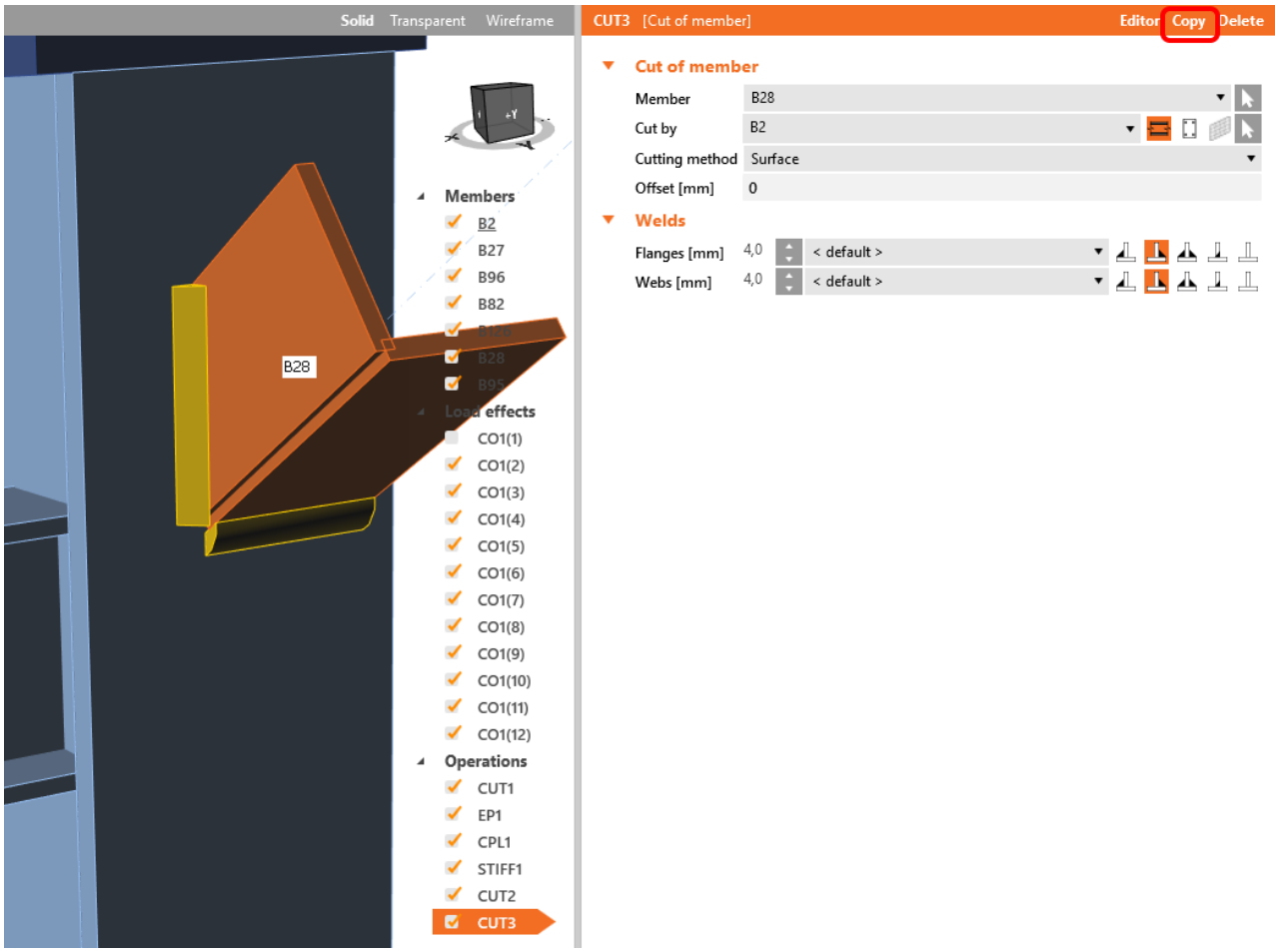


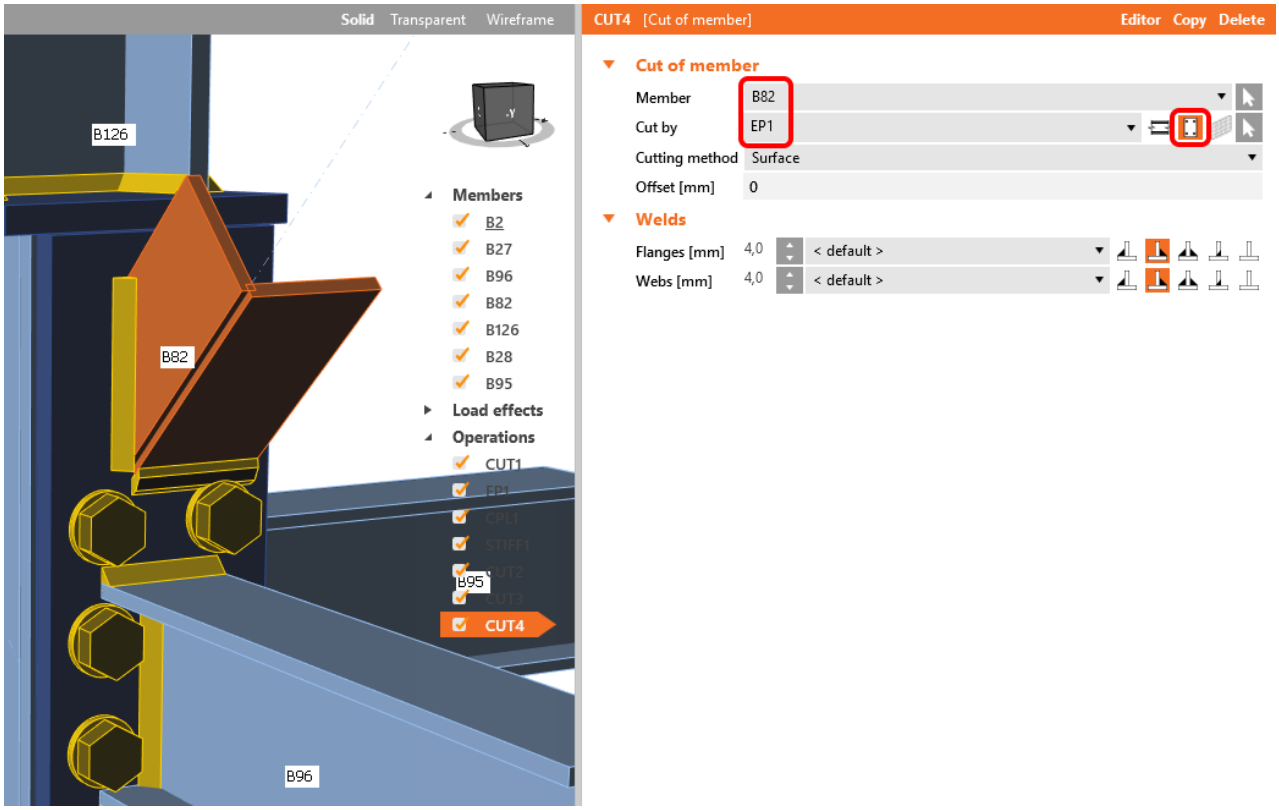
Again, we use the **Cut** operation for the diagonals.


Select operation
Cancel

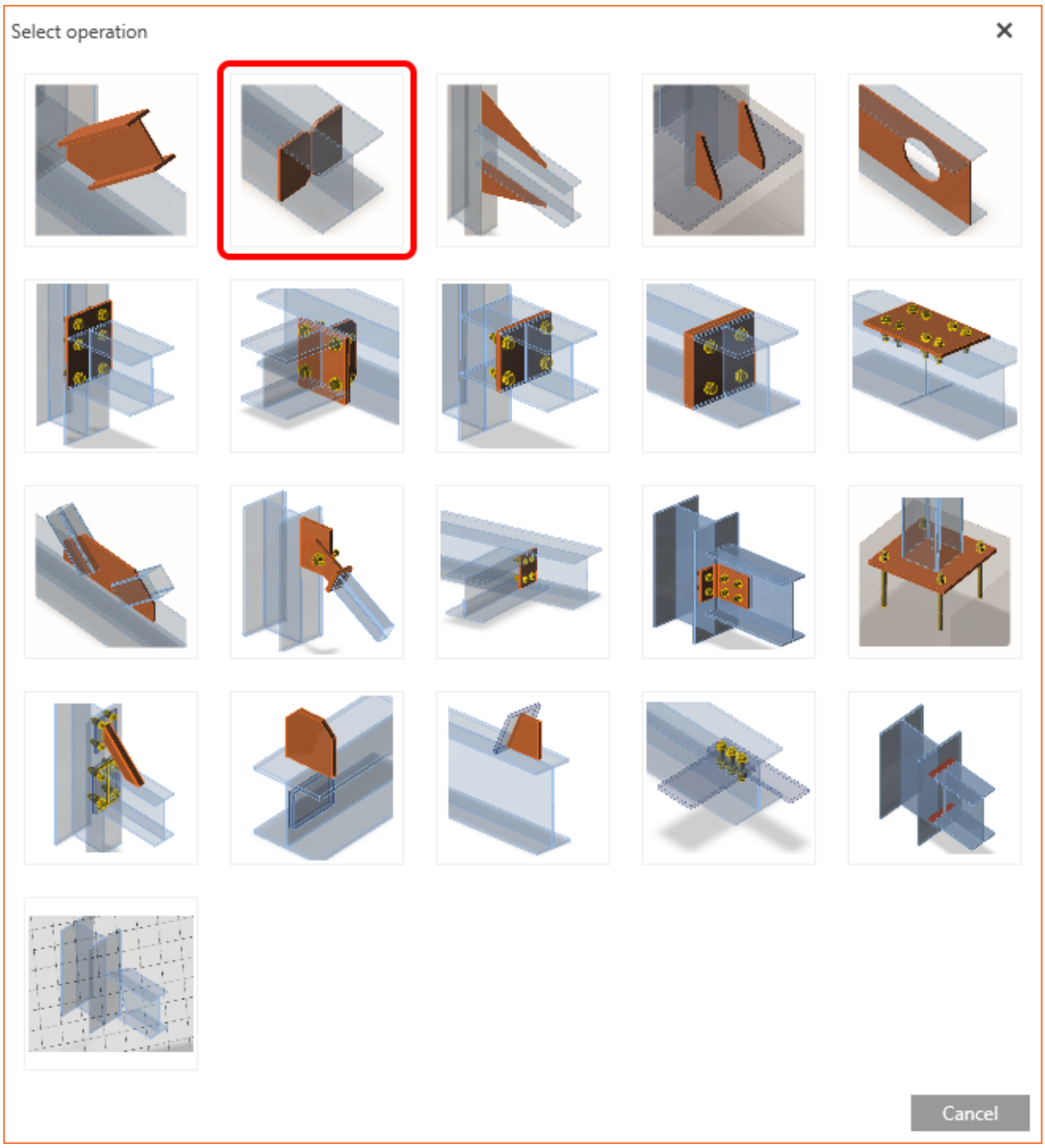


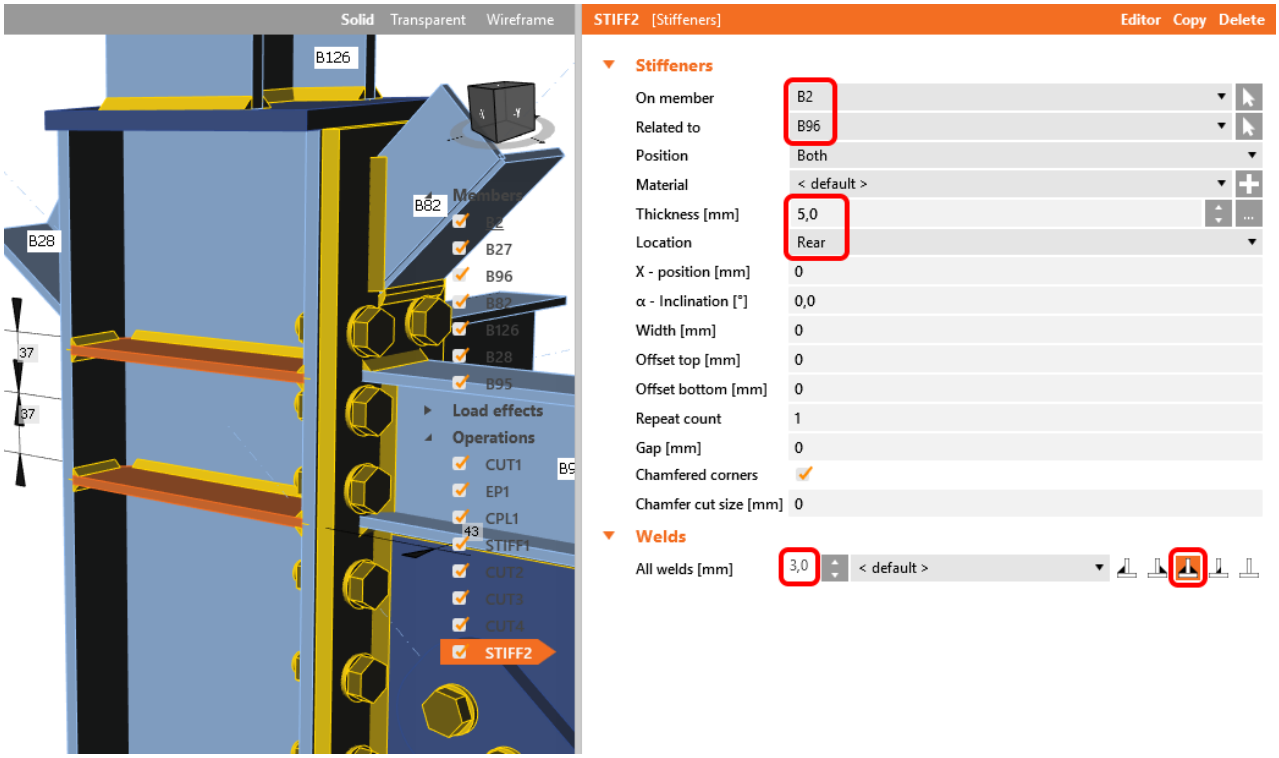
Now we **Copy** the last cut operation and set it for the other diagonal.



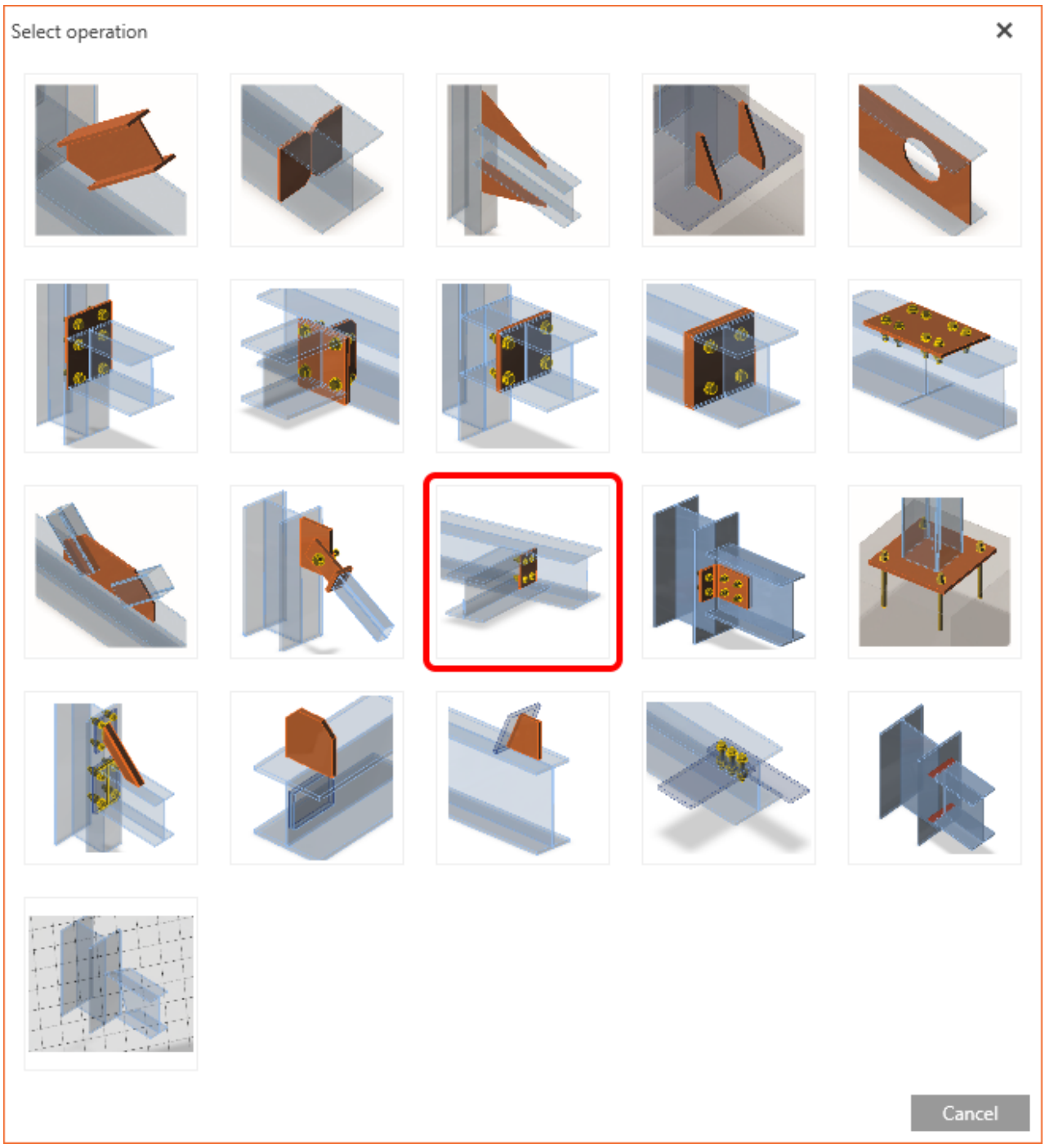


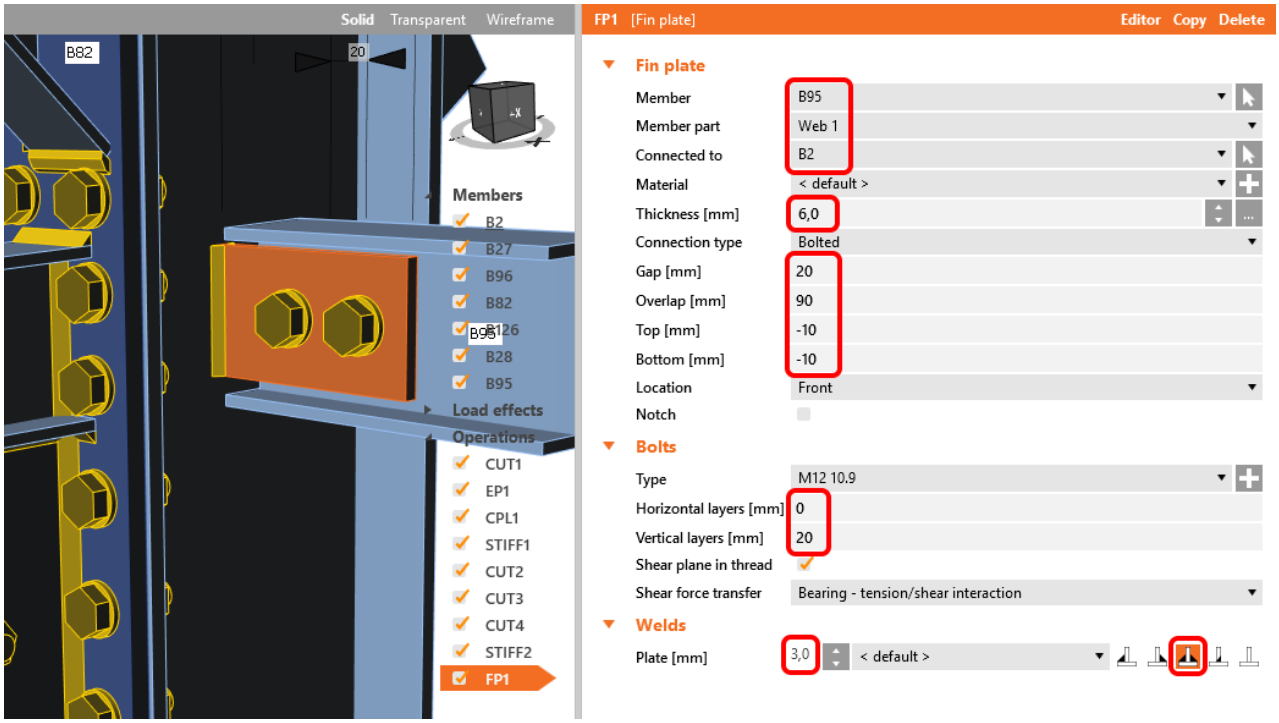
In the next step we add a **Stiffener** to the main column.


Select operation
Cancel



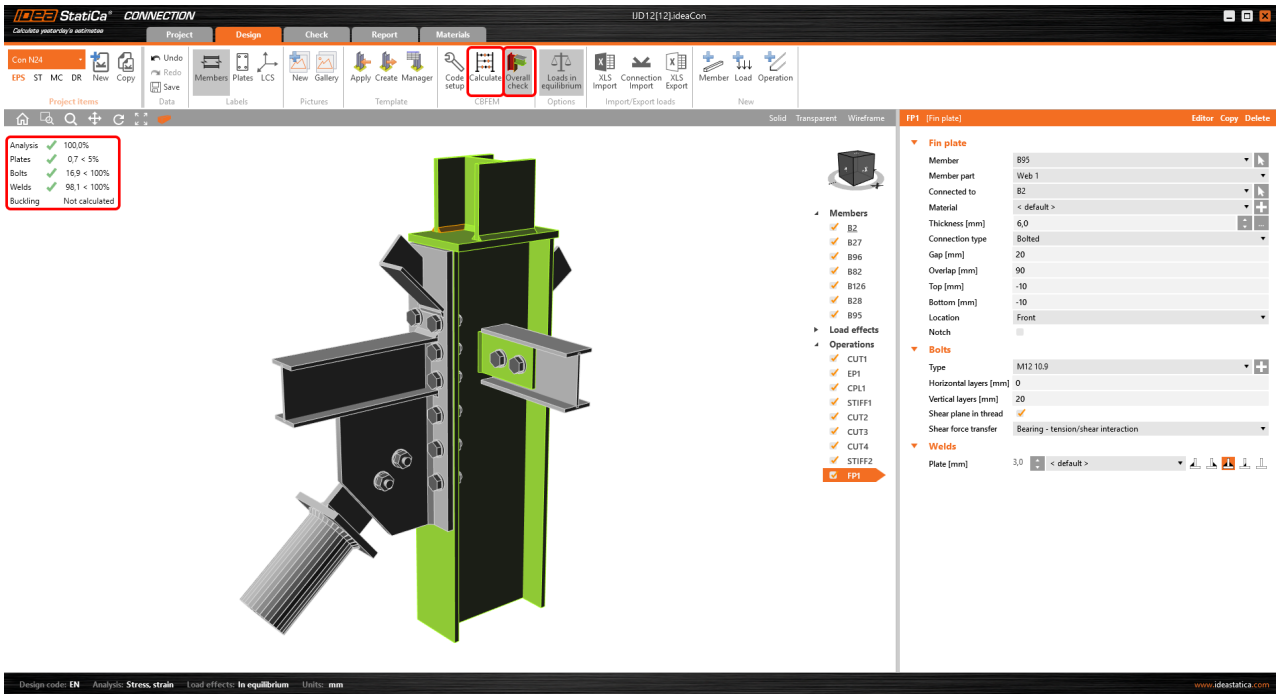
We finish the design by adding a **Fin plate** operation to connect the last member.


Select operation
Cancel

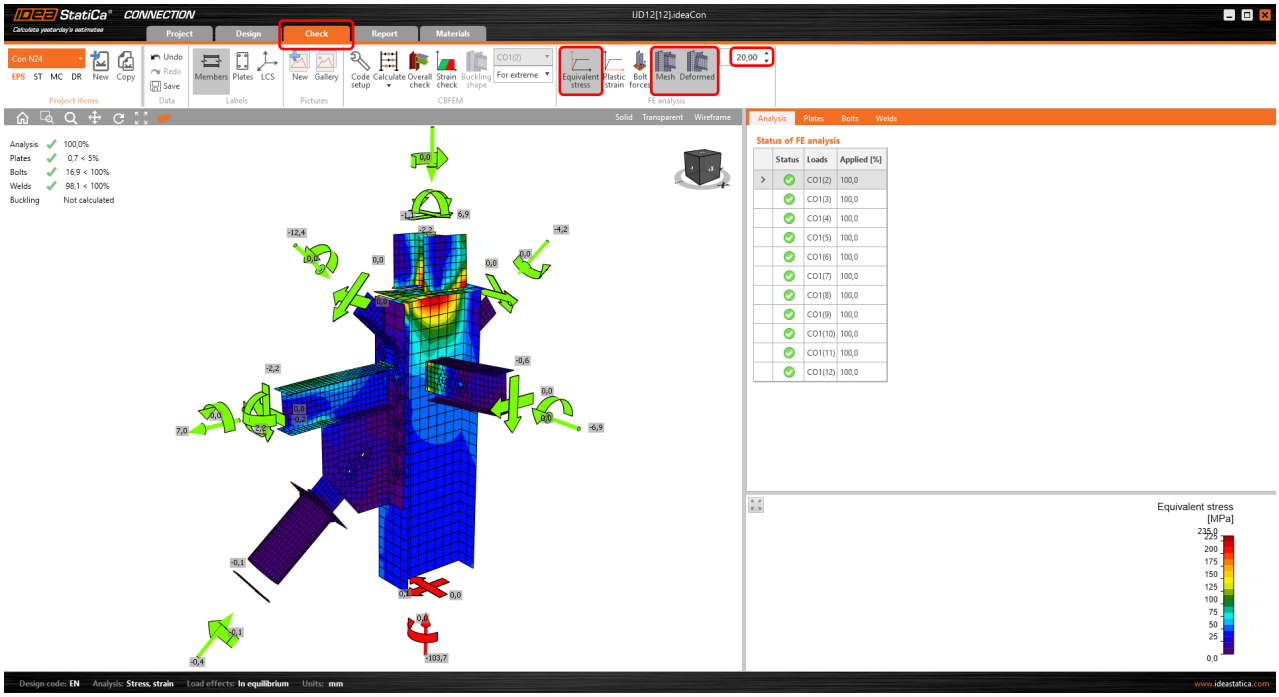


4 Check

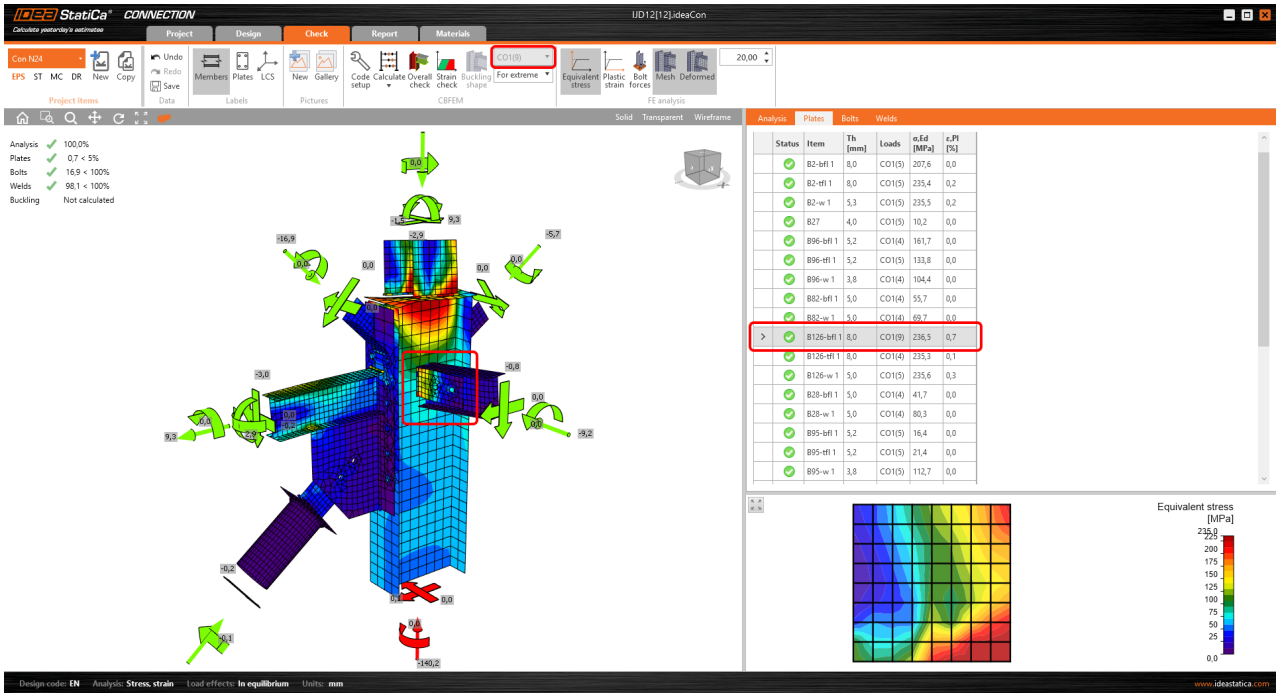
We start the analysis by clicking **Calculate** in the ribbon. Analysis model is automatically generated, calculation is performed for all the load cases and we can see the Overall check displayed together with basic values of check results.



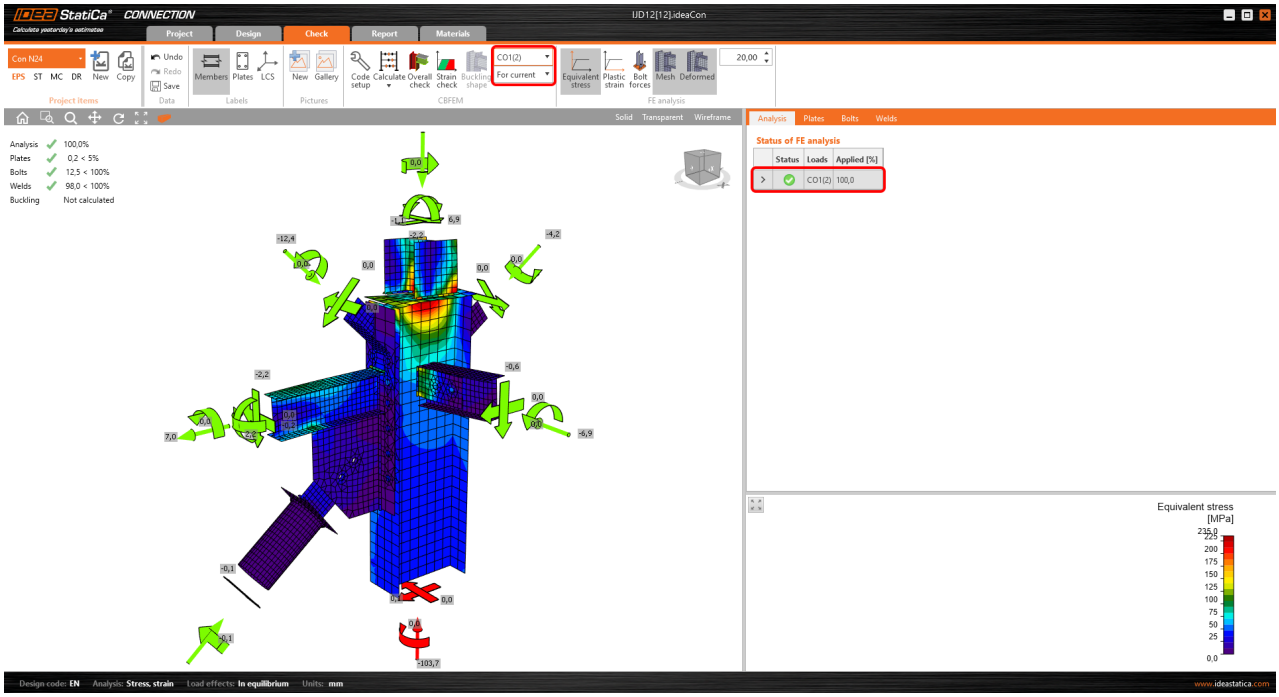
We go to the display tab **Check** and there we activate **Equivalent stress, Bolt forces, Mesh** and **Deformed** and change the deformation scale in the ribbon to get a full picture of what is happening in the joint.



By default, we see the **For extreme** results envelope of all the load effects, so the most unfavorable results are displayed altogether. If we click on any item (e.g. the fin plate), it shows us which load effect is the most unfavorable for it in the ribbon.

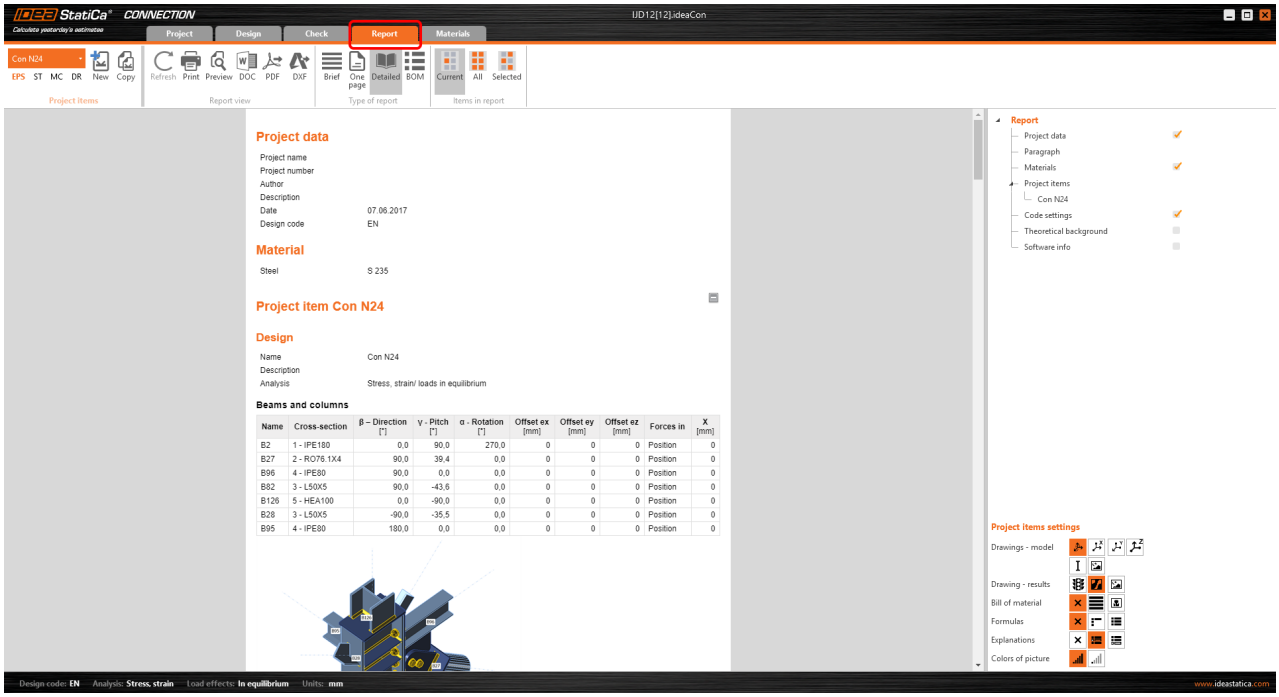


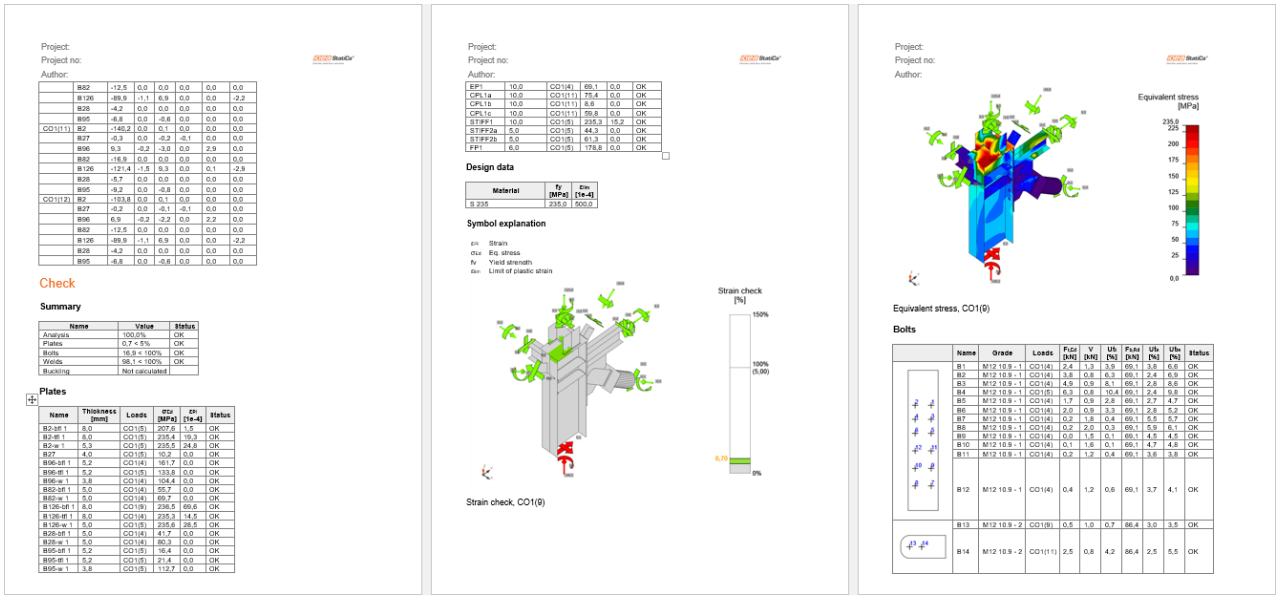
We can also display and browse results for each load effect separately by changing the **For extreme** option to **For current** in the ribbon.



5 Report

At last we go to the tab **Report.** IDEA StatiCa offers fully customizable report to print out or save in editable format.

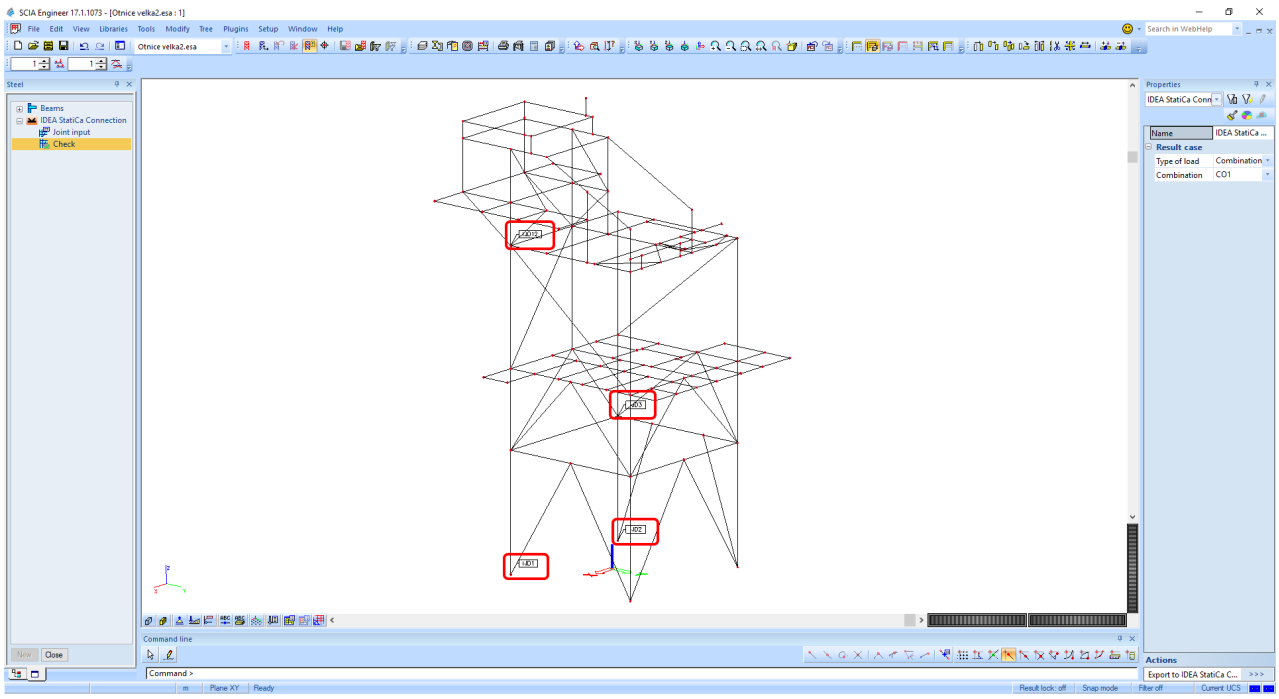




We have imported a joint from SCIA Engineer and designed and code-checked it according to Eurocode.

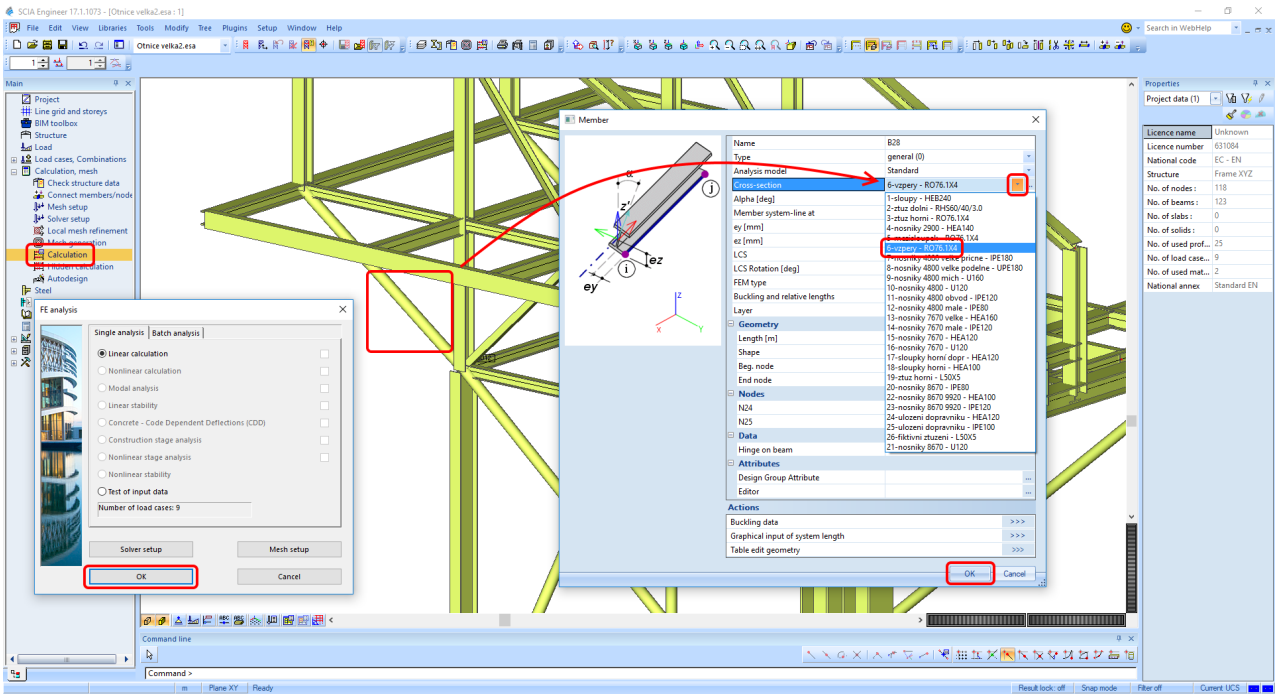
6 How to update the project

We save the project and close the application Connection. All joints exported from a SCIA Engineer project to IDEA StatiCa are kept on the list as a part of SCIA Engineer project file for later updates.

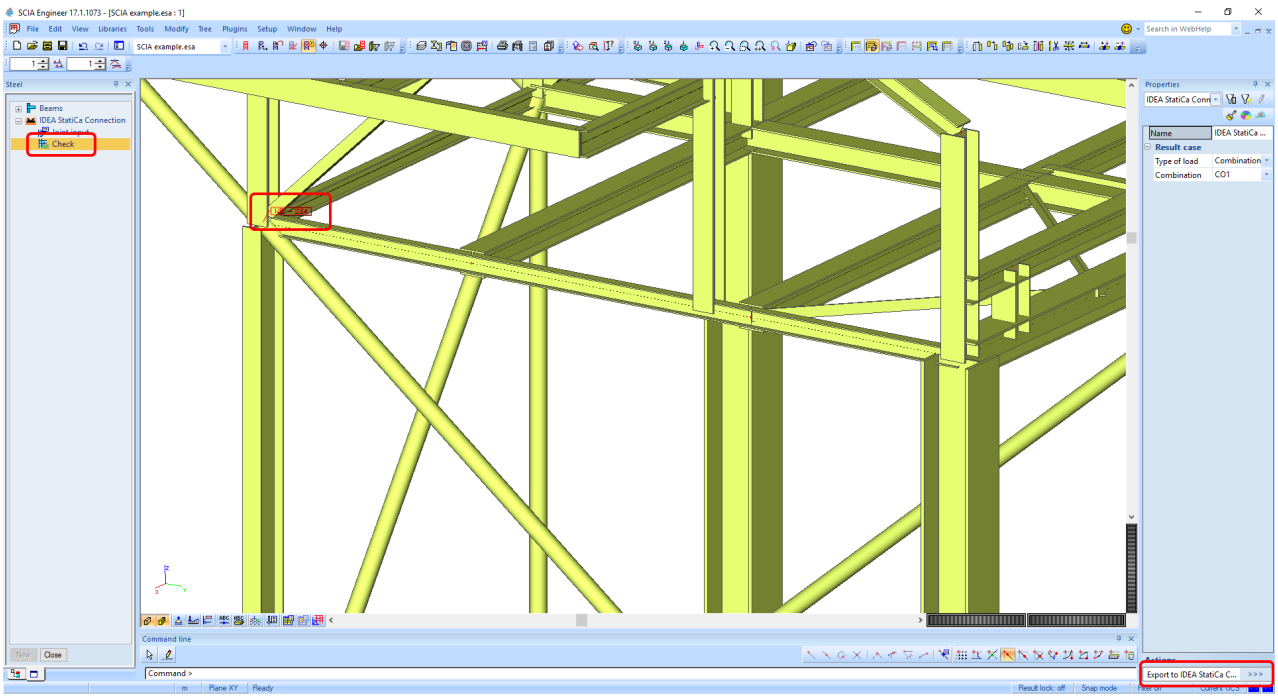


Note: In case you need to send a simple IDEA StatiCa joint model file, you can find it in the Temporary files folder of relevant SCIA Engineer project in the path C:\Users\"username"\ESA17.01\Temp

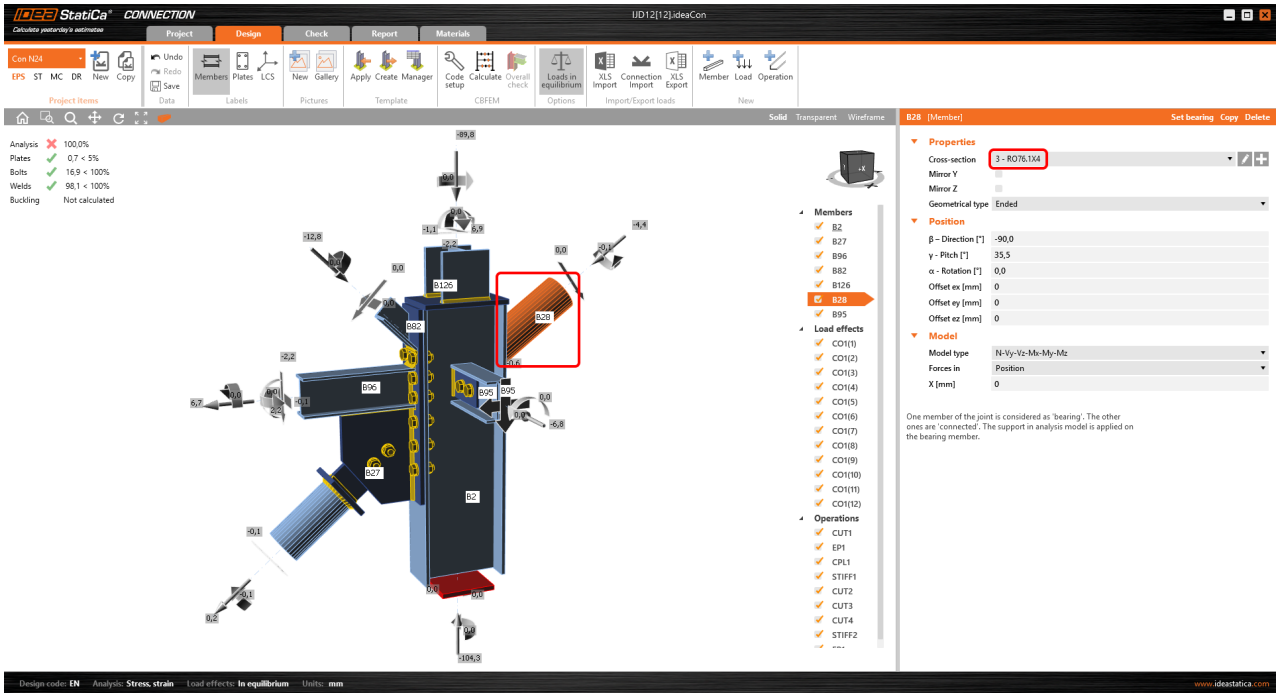
We go back to the SCIA Engineer model, change cross section of one of the diagonals and start the Calculation of linear analysis again.



Then we open again the command **Check**, run the command **Export to IDEA StatiCa Connection** in the Actions menu and select the joint sign in the workspace.



IDEA StatiCa opens and we can see the updated geometry while the design and settings adjust to the new state.



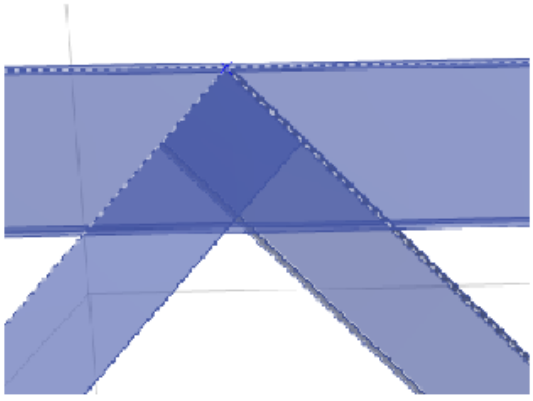
We can adjust the design, recalculate the joint code-check and save the project in IDEA StatiCa again and continue with exporting other joints.

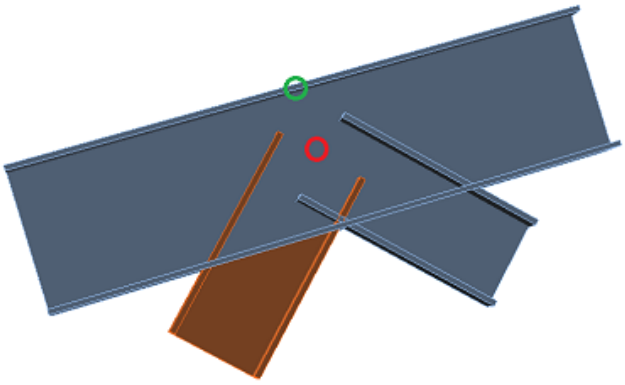
7 Known limitations

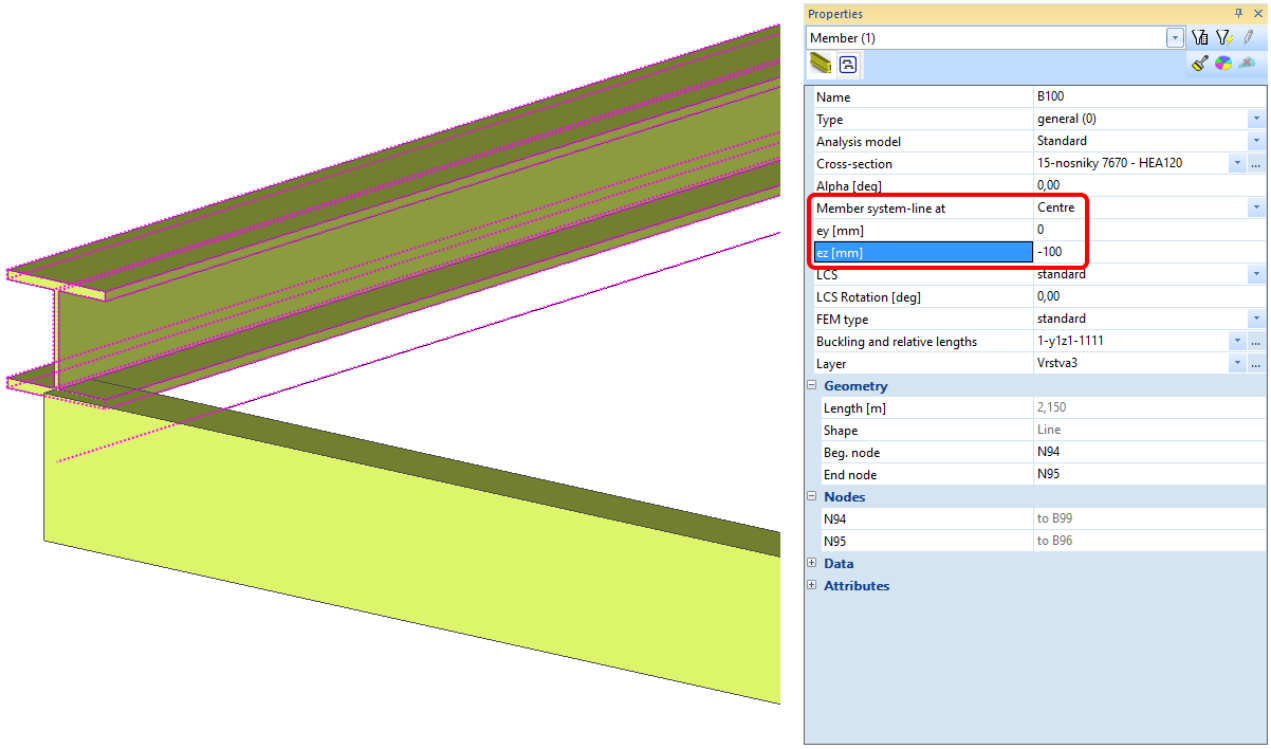
For now, the link works for a wide variety of connections / joints. However, please take into account yet unsupported functionality.

Limitation: Geometrical eccentricity - joint node is not in the central point

Workaround: Import the joint and move the beams manually to proper positions. Imported internal forces include the eccentricity effects, only the geometrical offset is missing.







Keywords:

connection, joint, Eurocode, code-check, SCIA Engineer, BIM link, import, integration, BIM link installer, update link, fin plate, display load effects, for extreme, for current, load combinations, cut, offset, end plate, joint update, export