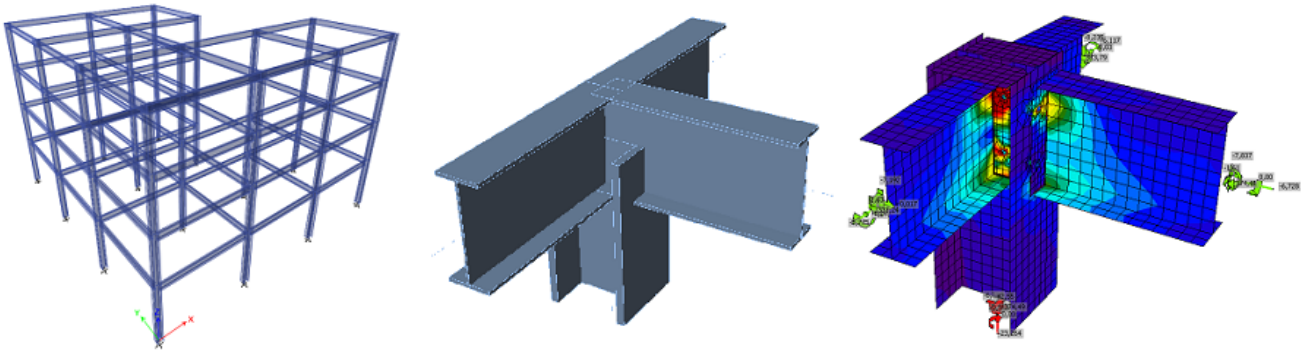
IDEA StatiCa tutorial – ETABS link (AISC)

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 ETABS and IDEA StatiCa Connection.

1 How to activate the link

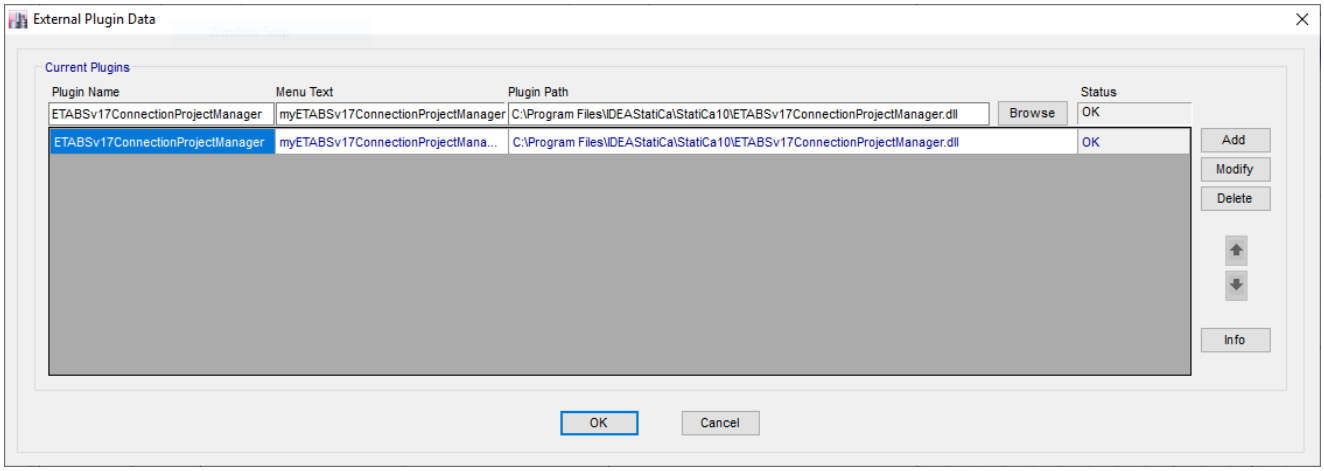
- Install the latest version of IDEA StatiCa, get it in the Downloads.
- Make sure you are using a supported version of ETABS – updates are published in the BIM section

After installation of both programs, start ETABS and Click Tools > Add/Show Plugins to open the Plugin Manager dialog. This option lets you install and add add-ins (programs) to the appropriate places in the ETABS menu.

Browse for

C:\Program Files\IDEAStatiCa\StatiCa10\ETABSv17ConnectionProjectManager.dll

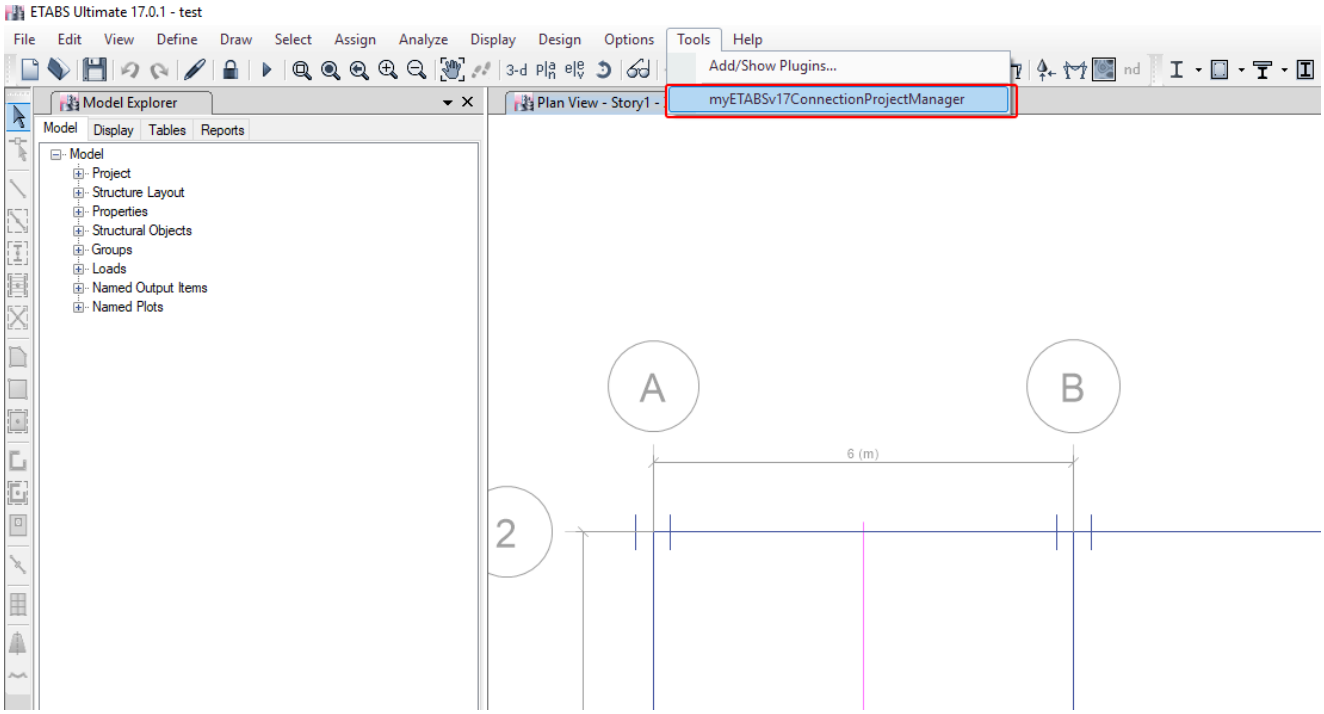
and click **Add**



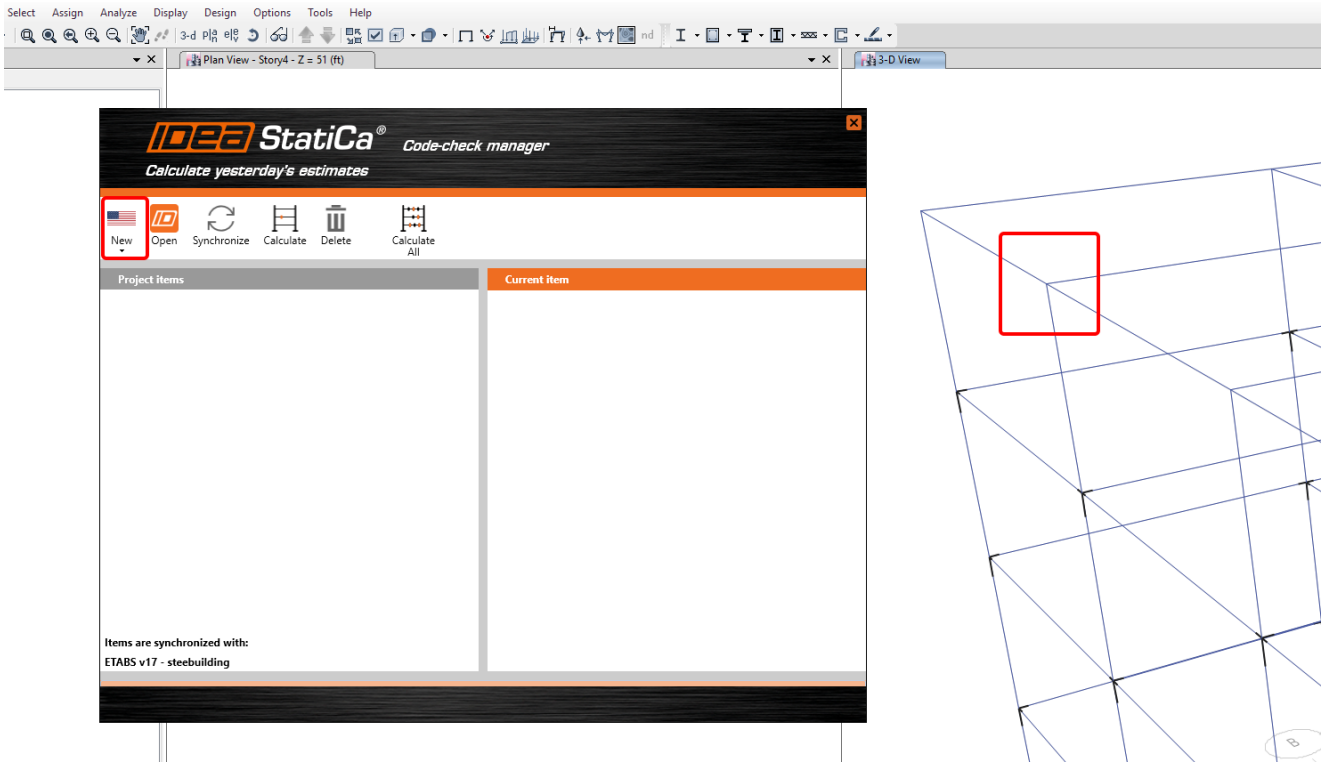
2 How to use the link

Open attached project in **ETABS** and run analysis.

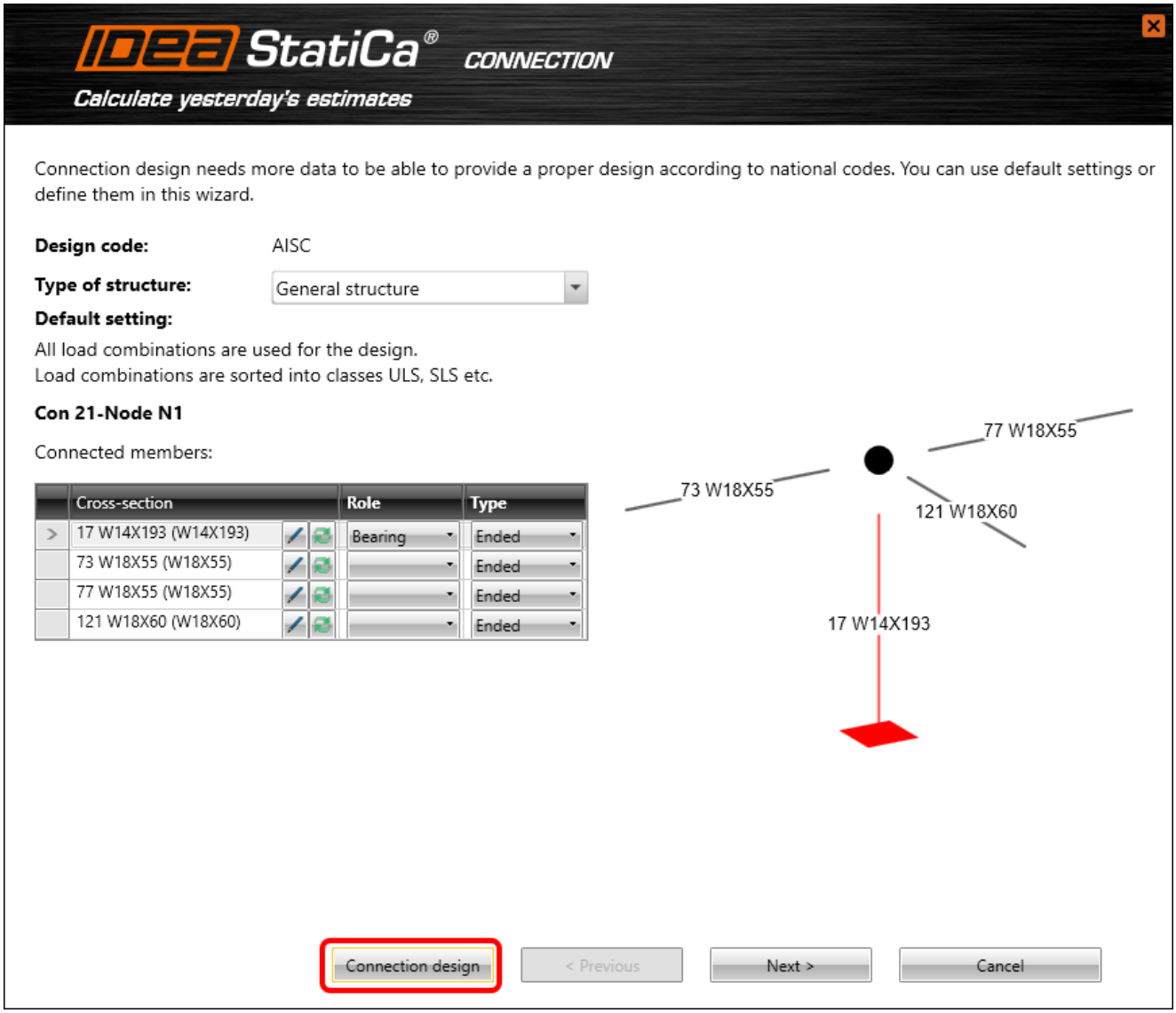
Go to menu item **Tools** and run the export command we have just defined.



The Code-check manager opens, we select the joint in ETABS model and by choosing a code we export it for the CBFEM analysis in IDEA StatiCa.

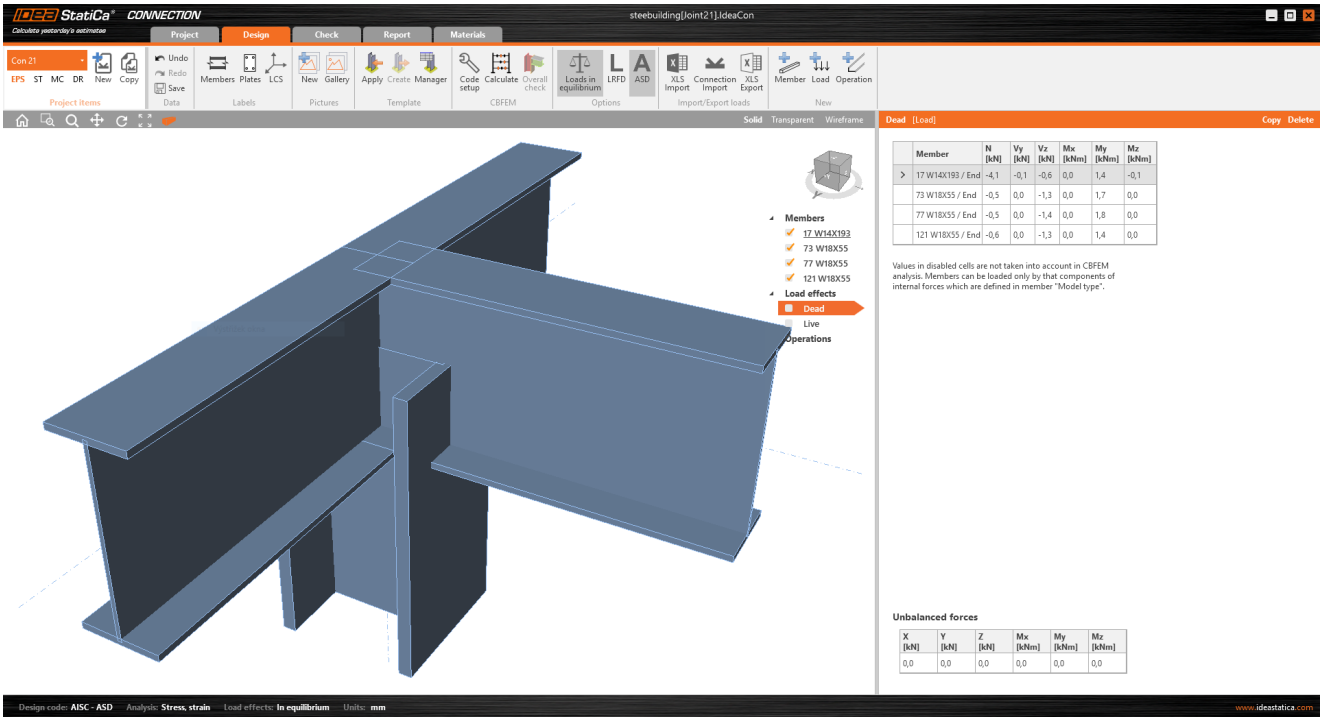


And we can go directly to **Connection design**.



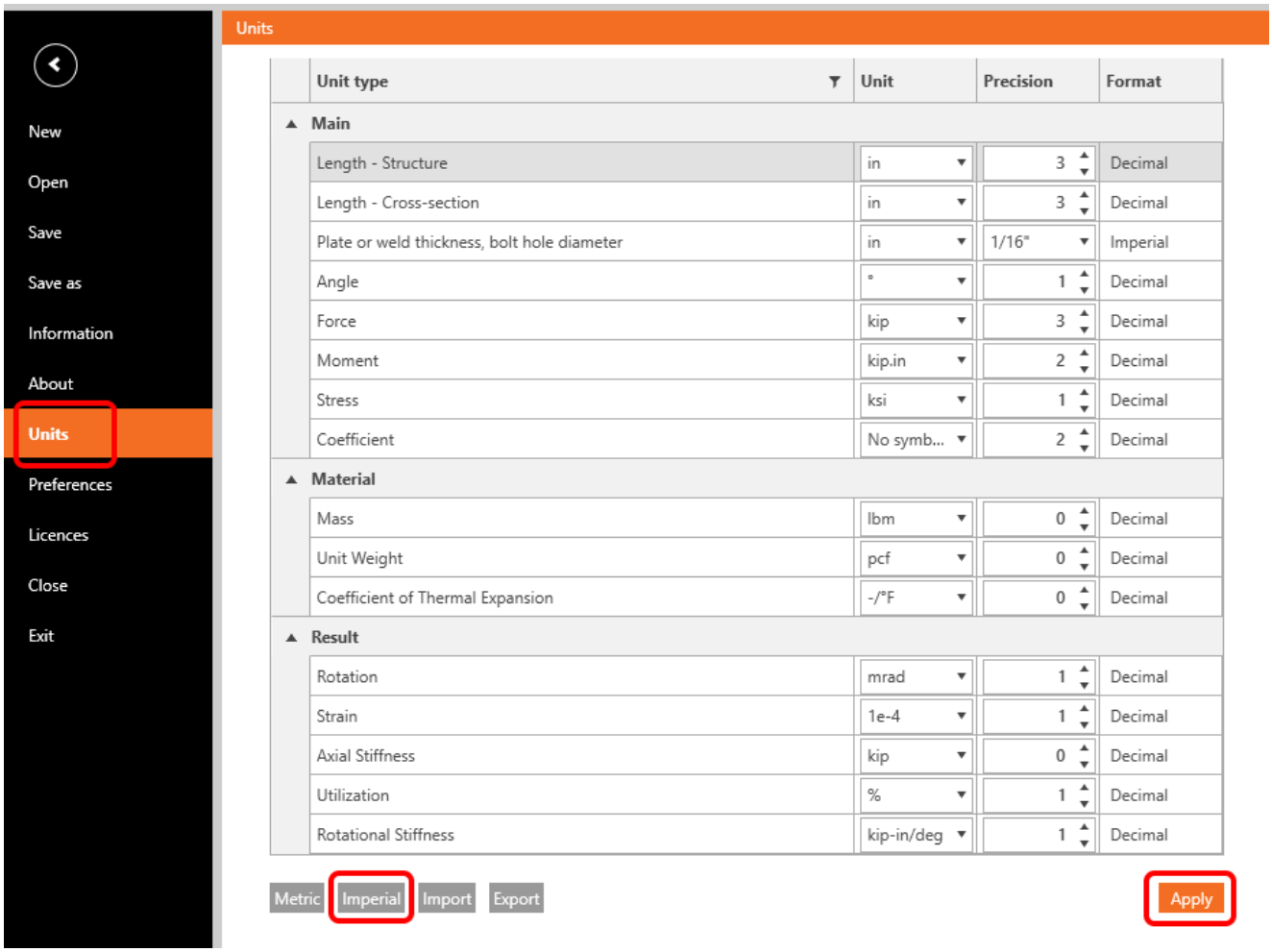
3 Design

Automatic data transfer is started and IDEA StatiCa Steel Connection with generated project is launched. All members and load effects were added automatically.

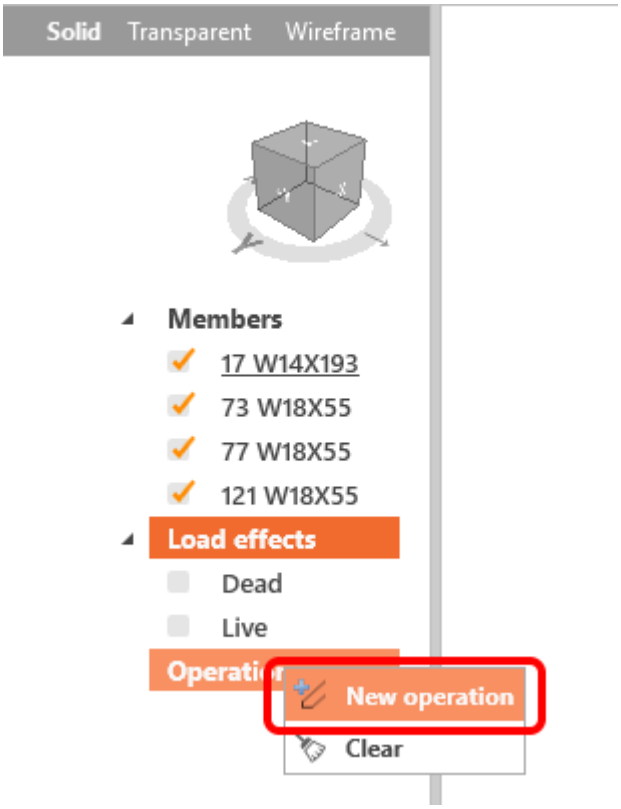


Before we start with designing the joint, we check the setting of units and set them to Imperial if necessary.

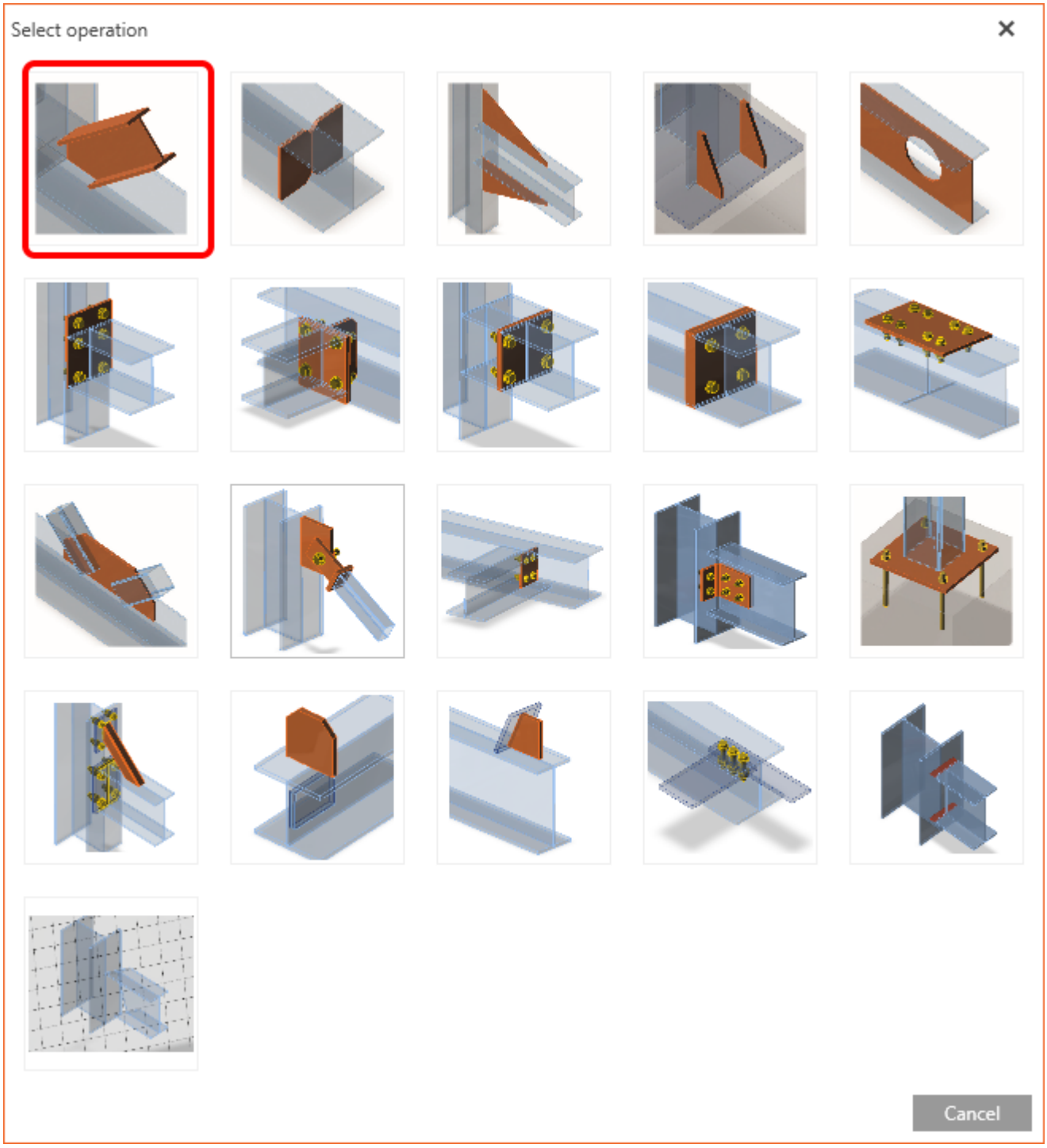
We go to **Project,** select **Units**, click **Imperial** and approve by **Apply**.



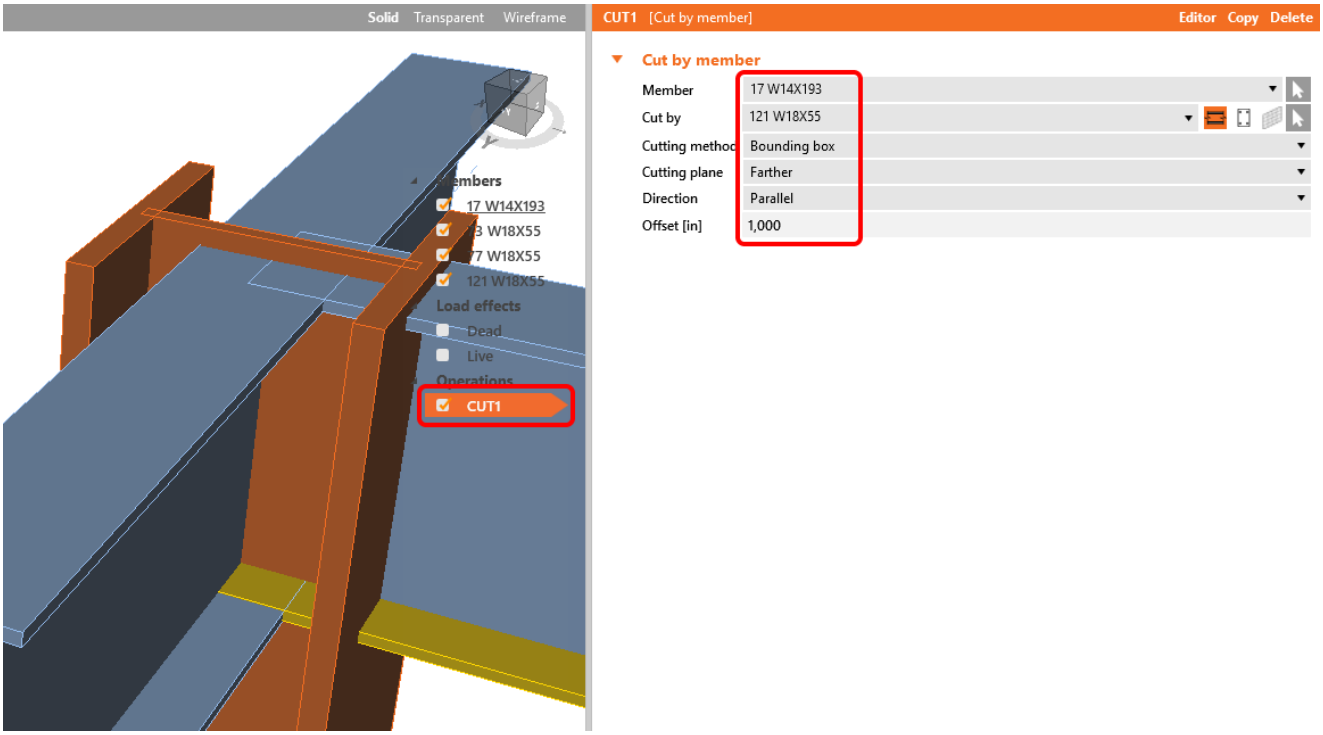
Now we will define a set of manufacturing operations to model connections between members. In the items tree we right-click on **Operations** and select the option **New operation.**



And we add operation **Cut**.

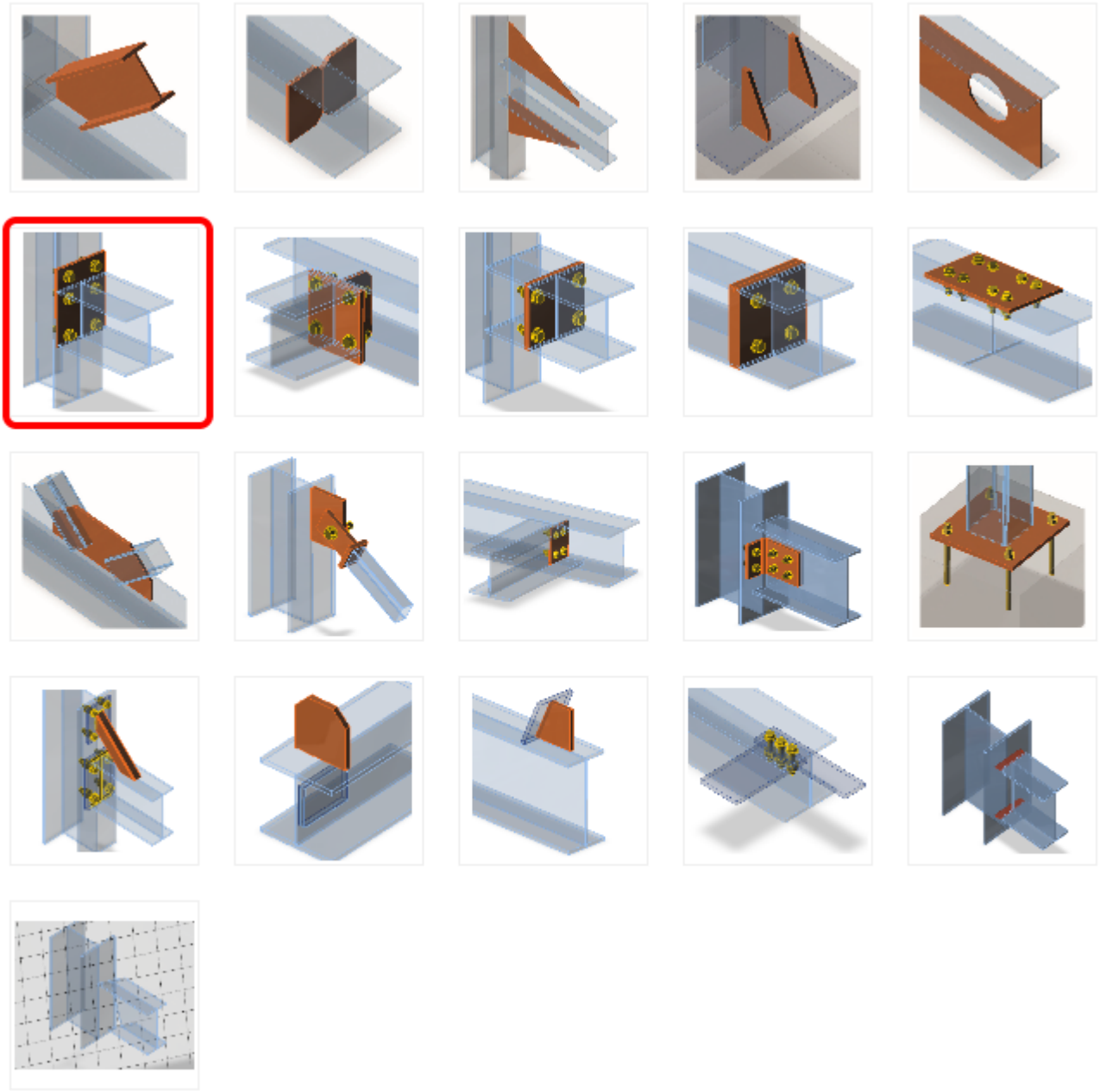


And set properties of **CUT1** as follows:

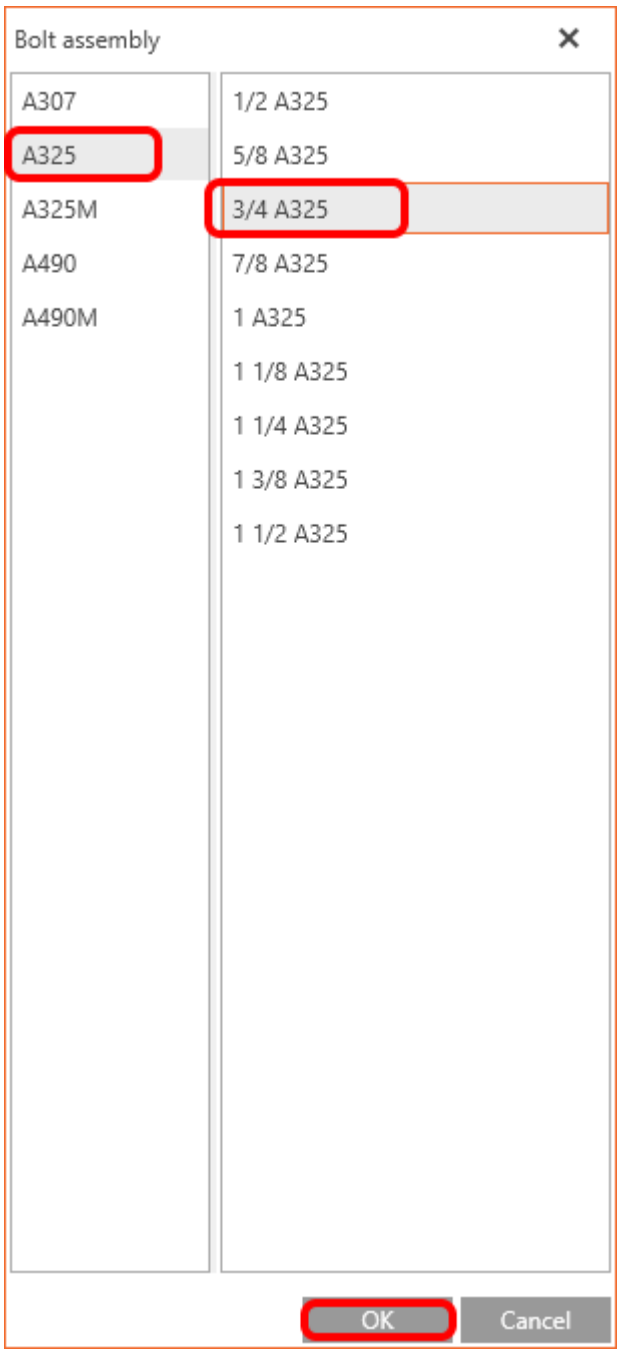


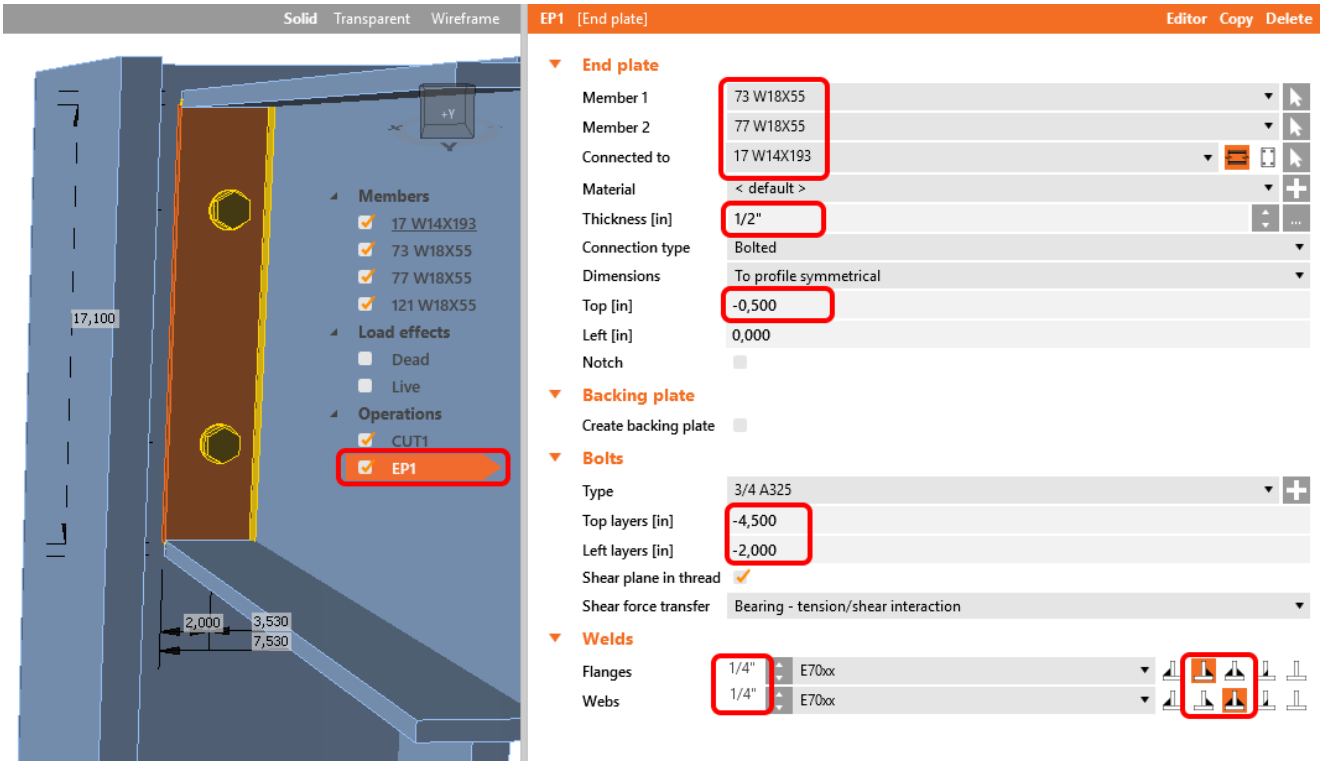
In the next step we use again **right-click** on **Operation** and add the manufacturing operation **End plate** and fill in the values below.

Select operation

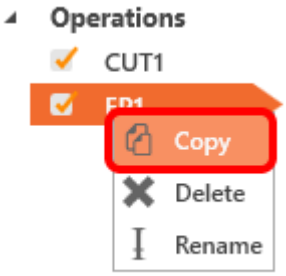


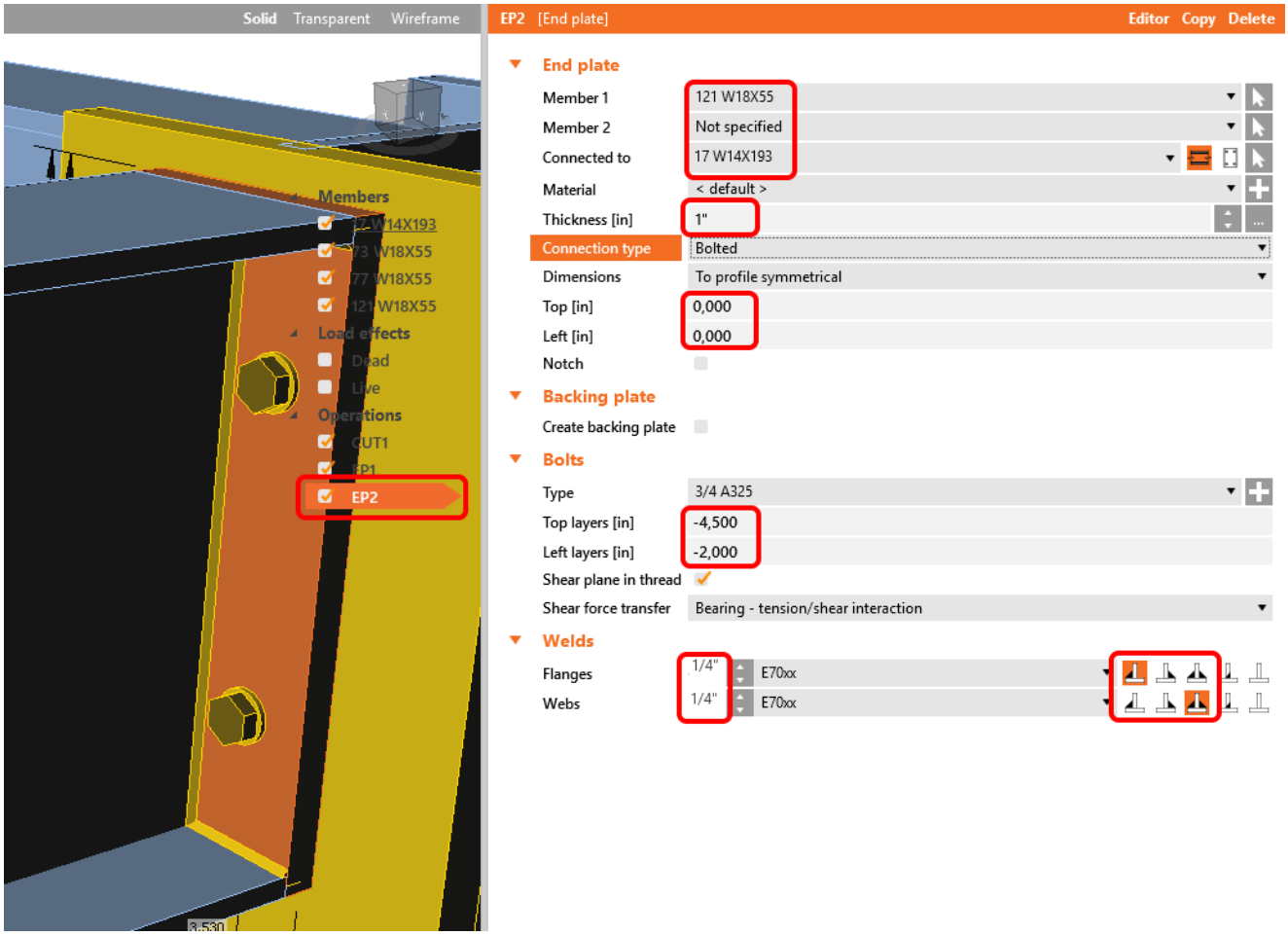
Cancel


Bolt assembly
A307
A325
A325M
A490
A490M
1/2 A325
5/8 A325
3/4 A325
7/8 A325
1 A325
1 1/8 A325
1 1/4 A325
1 3/8 A325
1 1/2 A325
OK
Cancel



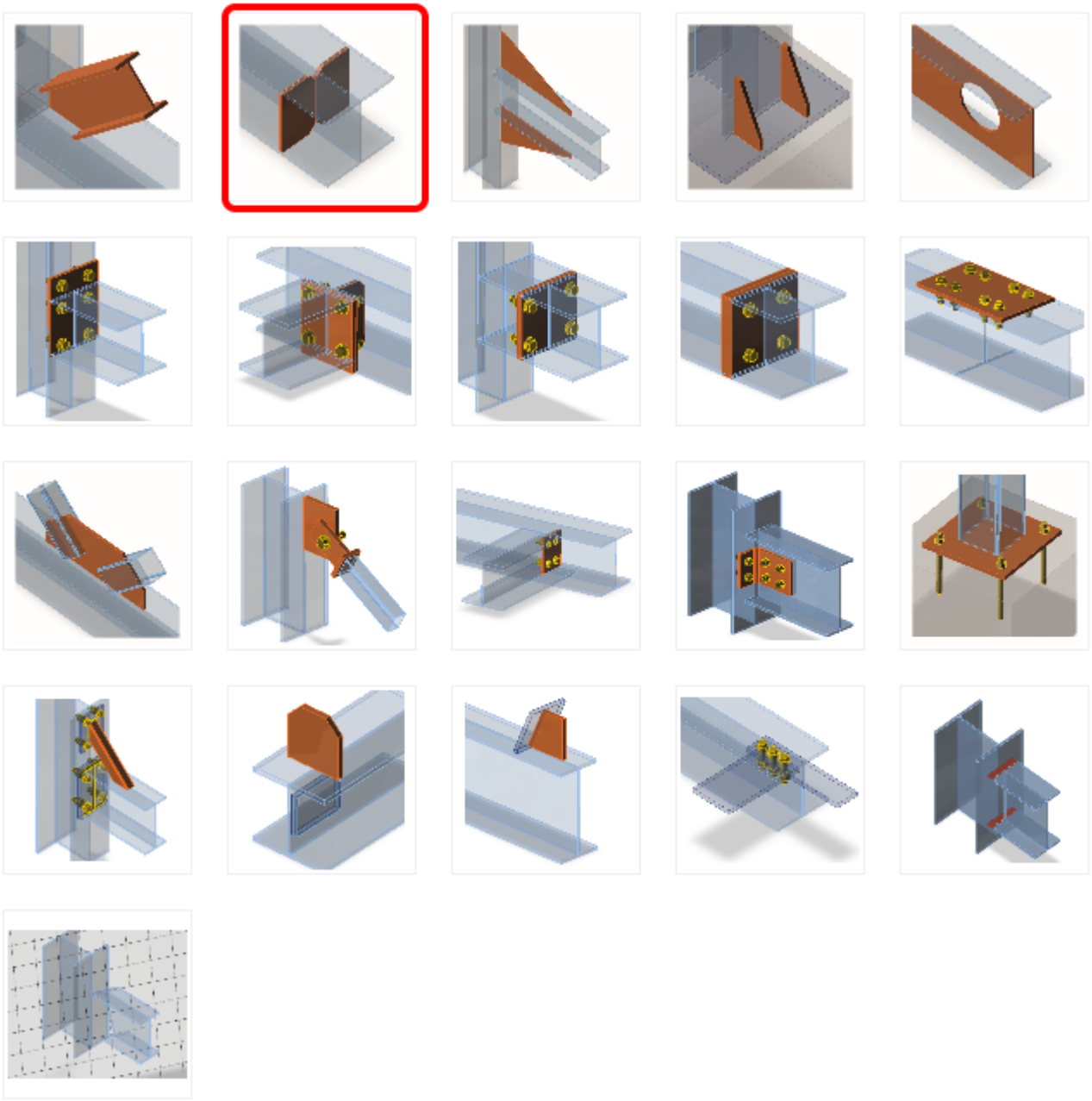
In the next step, **right-click** on **Operation EP1** and select **Copy**. Then change the values according next picture



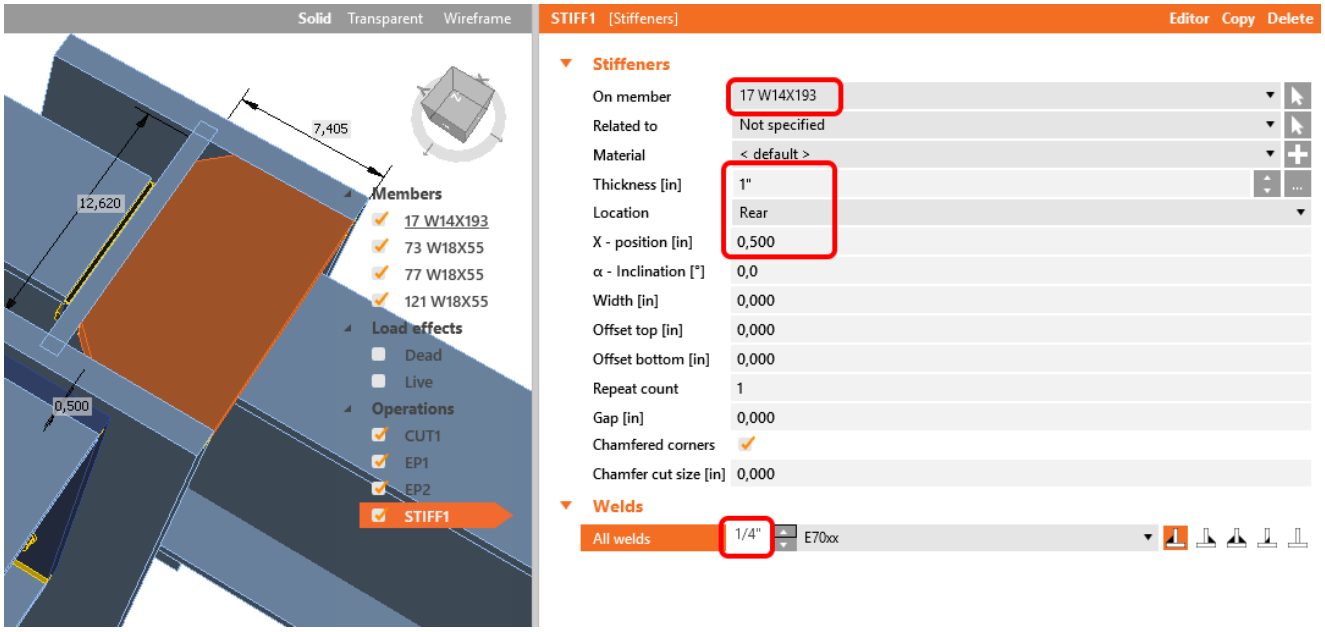


Next, we input a **Stiffener** and fill in the values below.

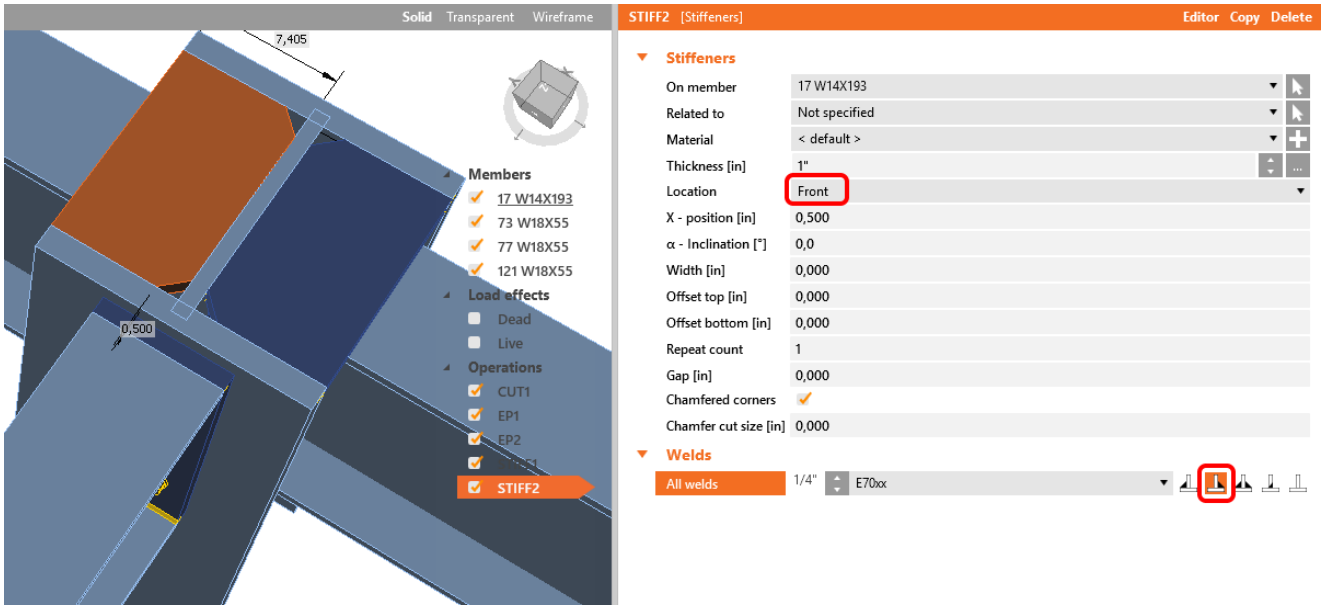
Select operation



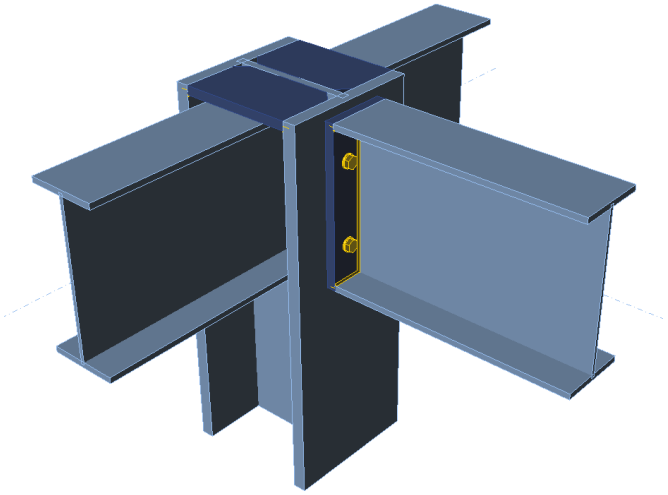
Cancel



Copy the Operation **STIFF1** and change the Location to **Front.**

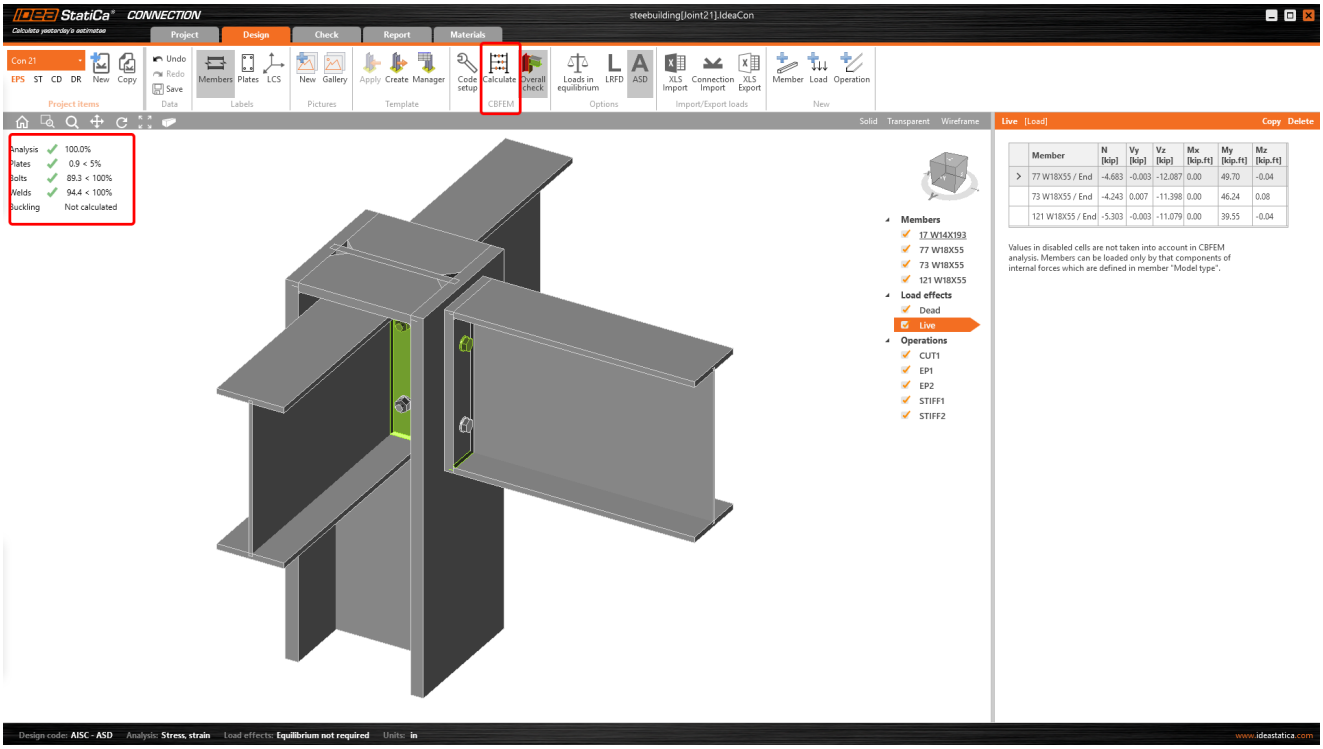


Finally, we can check the design of the joint.

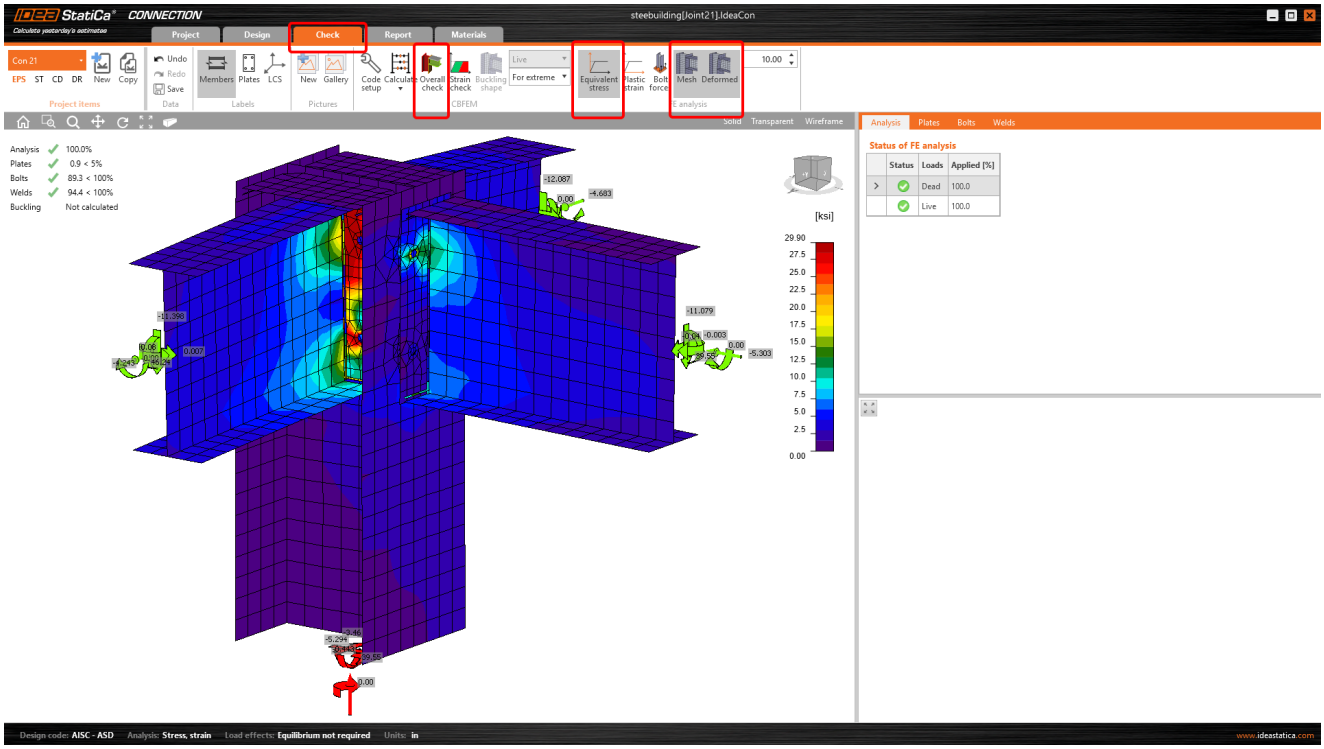


4 Check

The analysis based on CBFEM is started by icon **Calculate** from the top ribbon. Analysis model is automatically generated, calculation is performed and we can check results.



Activate **Overall check, Equivalent stress, Mesh** and **Deformed** from the ribbon to get a full picture of what is happening in the joint. Everything is displayed in the 3D window.



All values can be reviewed in detailed in the tables and 2D window. For example, to display check of bolts select tab **Bolts** tab.

Analysis | Plates | Bolts | Welds

Check of bolts for extreme load effect

		Status	Item	Loads	Ft [kip]	V [kip]	Bearing Rn/Ω [kip]	Utt [%]	Uts [%]	Utts [%]
	−	✓	B1	Live	17.674	2.285/1.909	52.062	89.0	19.2	-

Tension resistance check (AISC 360-16: J3-1)

$\frac{R_n}{\Omega} = \frac{F_{nt} \cdot A_b}{\Omega} =$ 19.863 kip ≥ $F_t =$ 17.674 kip

Where:

F_{nt} = 89.9 ksi – nominal tensile stress from AISC 360-16 Table J3.2

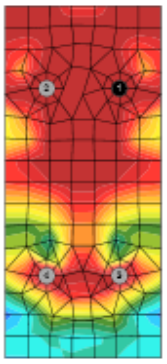
A_b = 0.442 in^2 – gross bolt cross-sectional area

Ω = 2.00 – safety factor for tension and shear

	+	✓	B2	Live	17.738	2.378/2.005	52.062	89.3	20.0	-
	+	✓	B3	Live	10.863	4.115/3.876	52.062	54.7	34.5	57.3
	+	✓	B4	Live	10.884	4.336/4.119	52.062	54.8	36.4	58.5
	+	✓	B5	Live	12.783	2.620	58.497	64.4	22.0	-
	+	✓	B6	Live	12.994	2.618	58.497	65.4	22.0	-
	+	✓	B7	Live	1.822	2.927	58.497	9.2	24.6	-
	+	✓	B8	Live	1.843	2.941	58.497	9.3	24.7	-

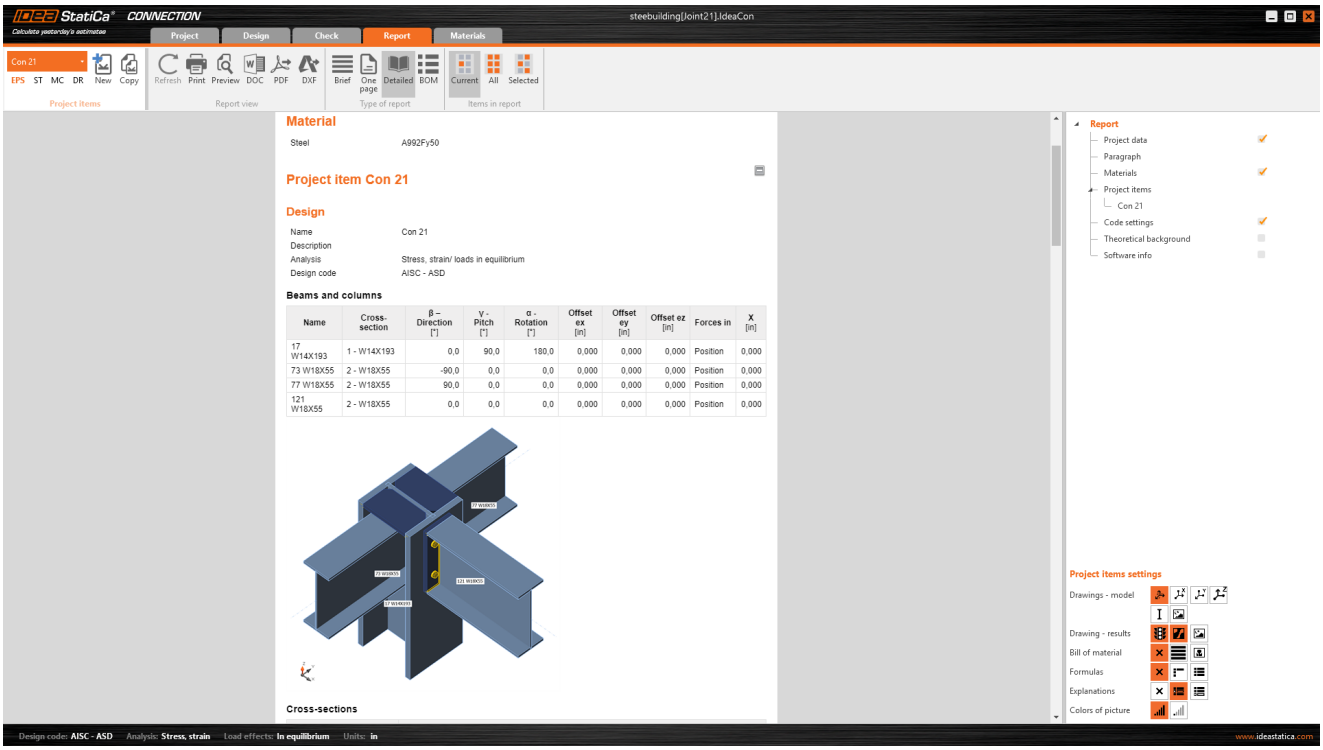
Design data

	Grade	Tension Rn/Ω [kip]	Shear Rn/Ω [kip]
>	3/4 A325 - 1	19.863	11.918



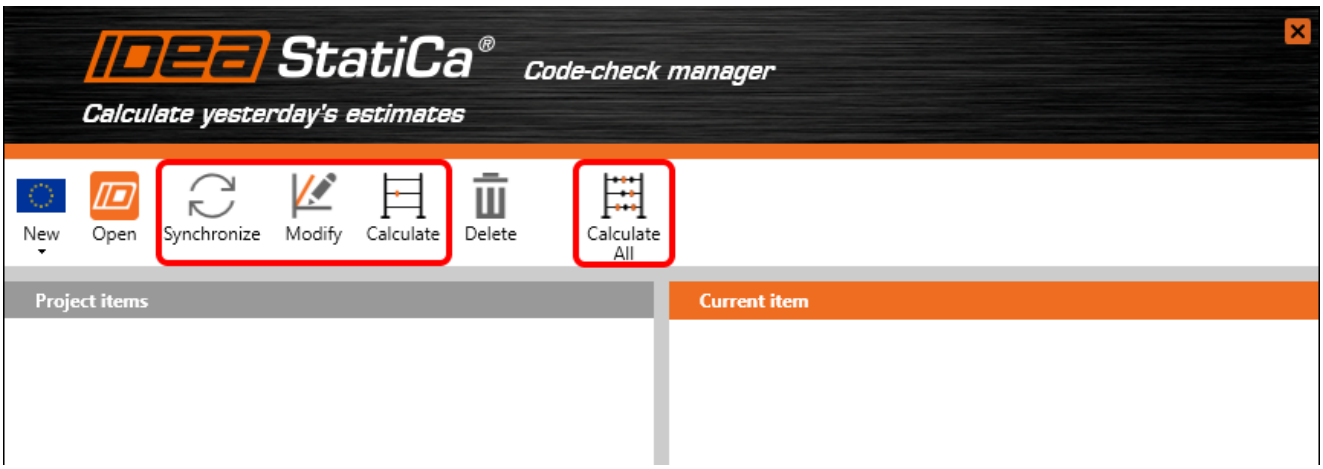
5 Report

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, designed and code-checked a steel joint according to AISC.

6 Synchronize models



Code-check manager is a BIM tool to export and synchronize connections from other programs. It is launched directly in the 3rd party applications via a command/icon.

Synchronize - IDEA StatiCa detects changes in already imported entities (changes in thickness, changes in cross-section, modification of properties of welds, bolts, etc.) and updates the project in IDEA StatiCa Connection.

Modify - IDEA StatiCa imports the whole joint again with modified topology (different number of members, new entities, etc.).

Calculate - Synchronize and calculate current item and provide a new set of results.

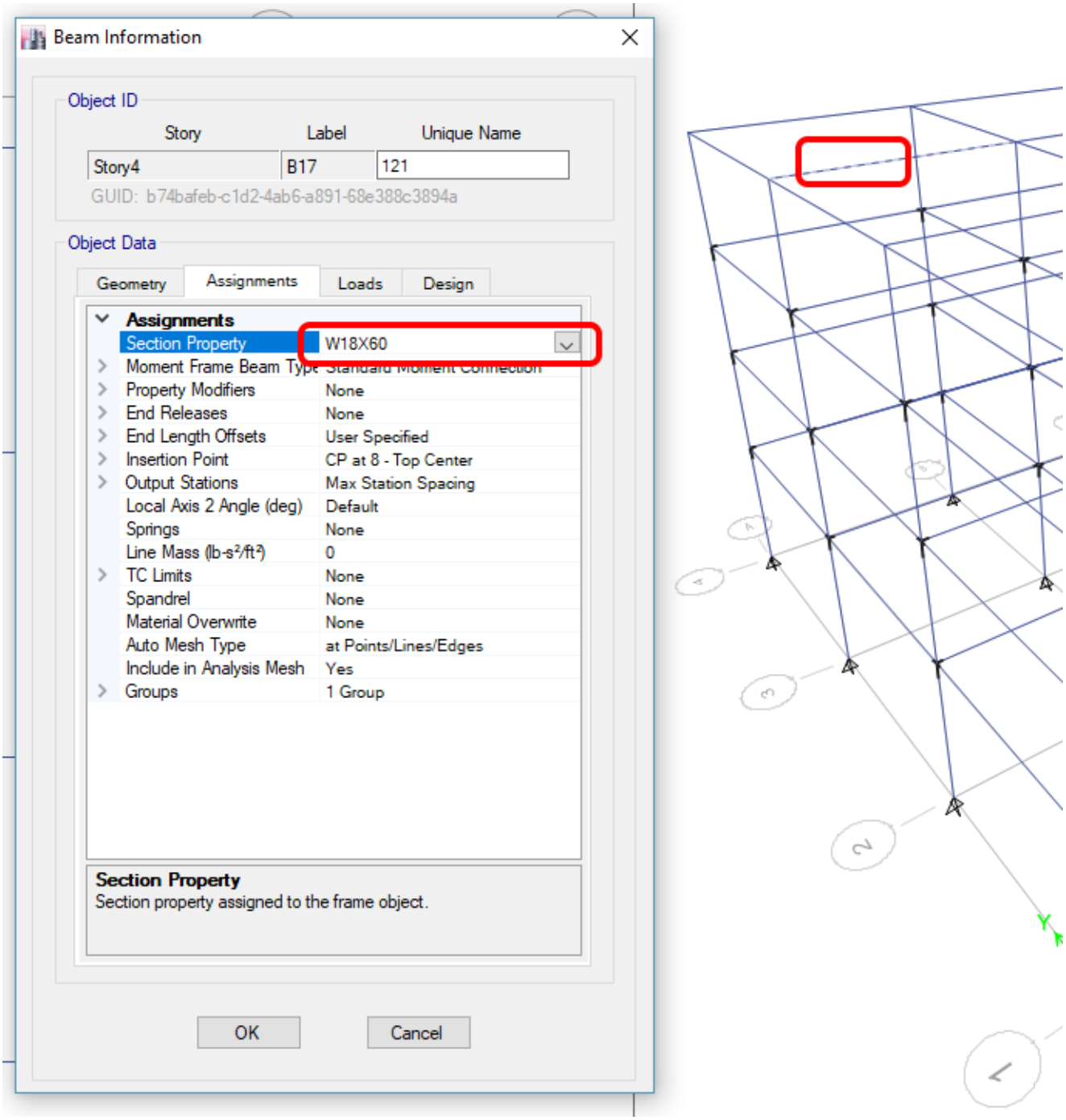
Calculate all - Synchronize and calculate all items and provide new set of results.

Note

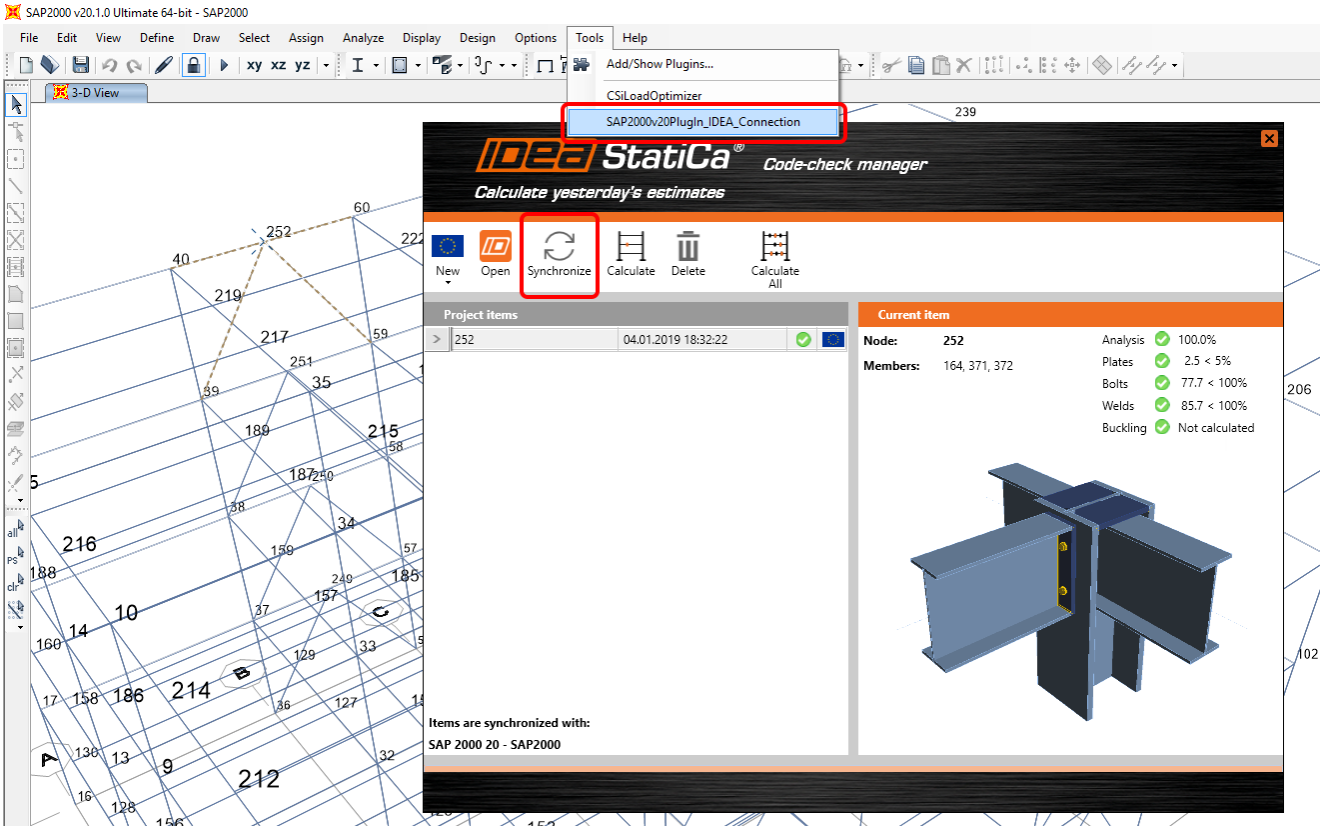
Kindly be aware that IDEA StatiCa syncs with a model of the 3rd party application, not the other way around. If we **add operations in IDEA StatiCa** and then use the options described above (Synchronize; Modify; Calculate; Calculate all), the additionally added operations will be deleted.

We save the project in IDEA StatiCa and close the application Connection. All joints exported from ETABS project to IDEA StatiCa are kept on the list inside ETABS.

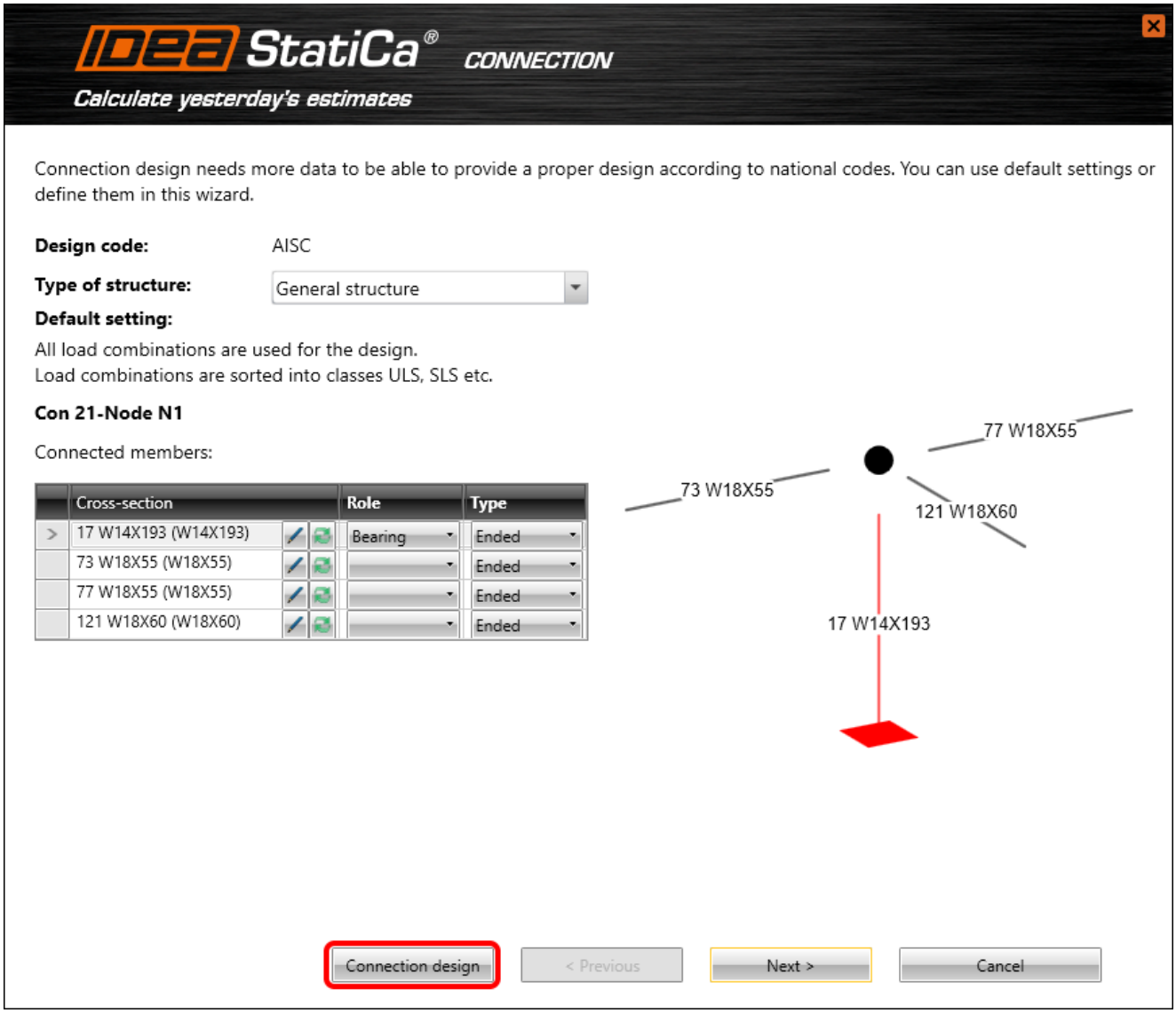
Unlock the model and change the cross-section of member **121** from W18x655 to W18x60.



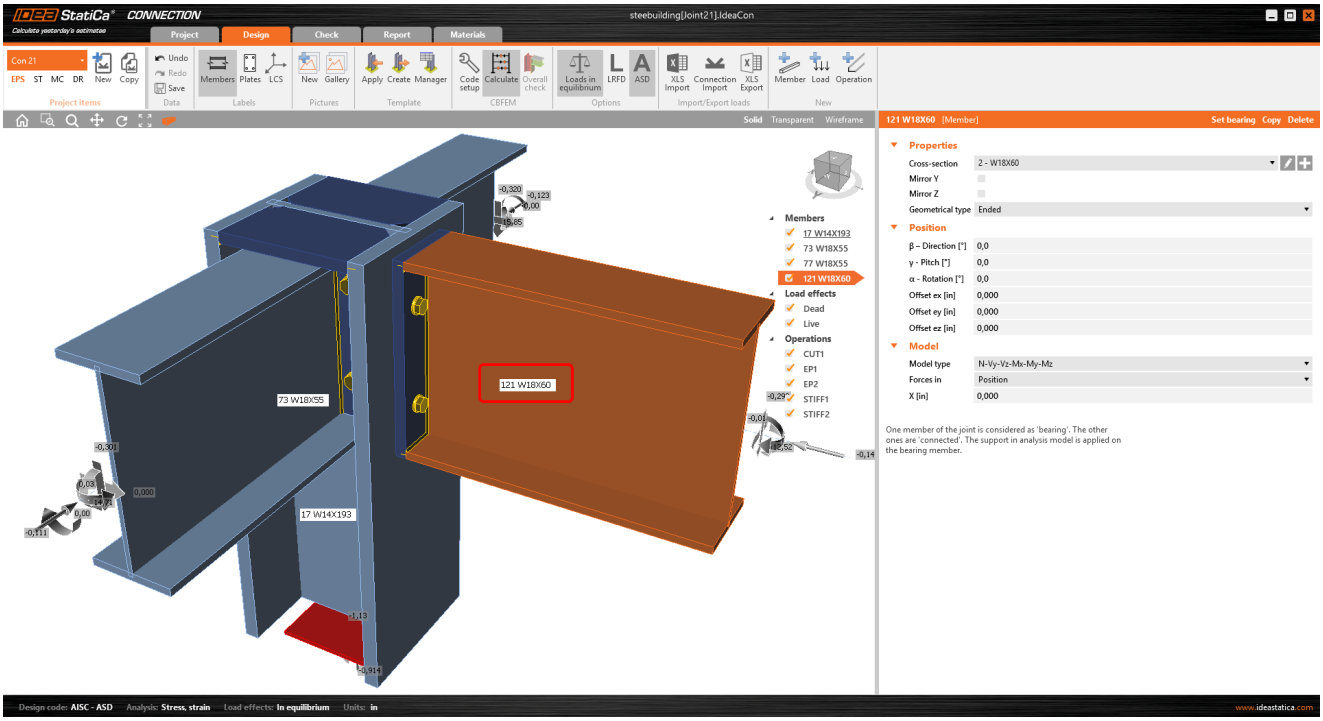
We run the analysis, select the export command in the upper ribbon and in the Code-check manager click on **Synchronize**.



In the next window just select **Connection design**.



As you can see, the cross-section of the Member 121 has been changed, but all previous operations remained.

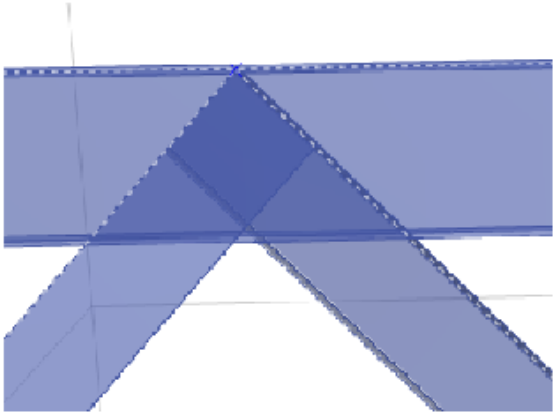


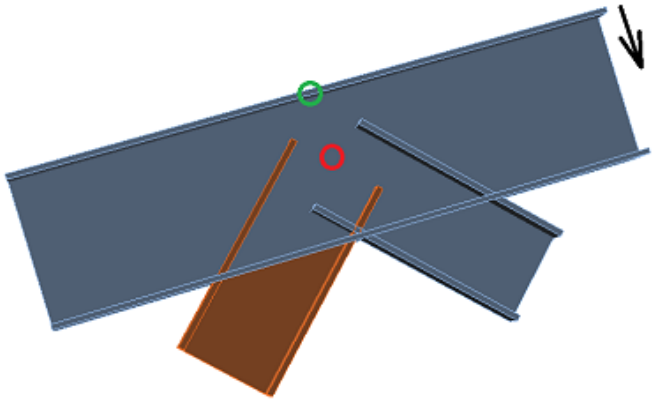
We have imported, designed and code-checked a steel joint according to AISC.

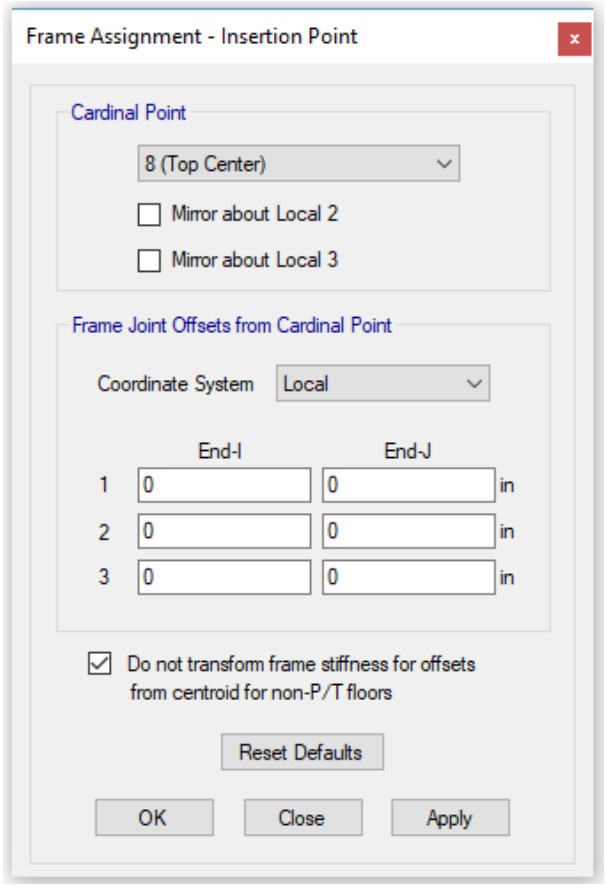
7 ETABS known limitations

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

Eccentricity - Centroid is not set as Cardinal point







Workaround: Import the whole joint and manually move the beams with eccentricity to the proper position.

Keywords:

connection, joint, AISC, analysis, end plate, cut, ETABS, BIM, BIM link, code-check manager, CBFEM