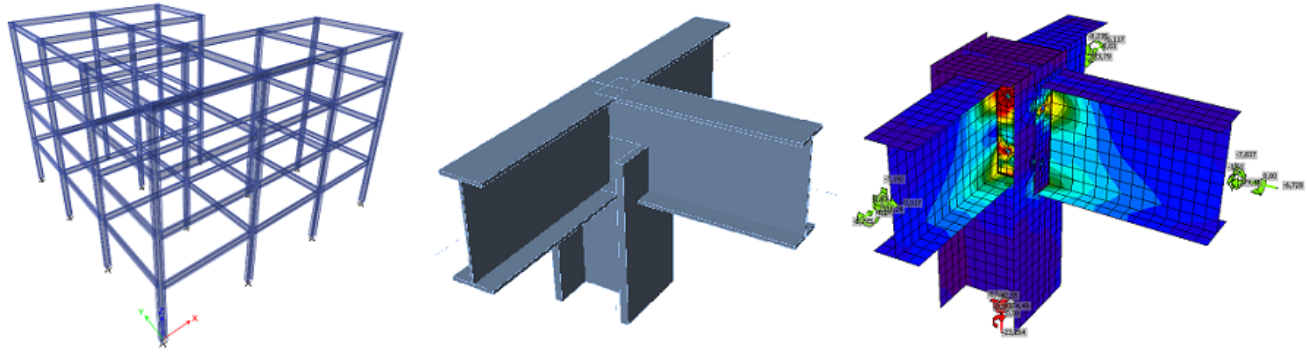


IDEA StatiCa tutorial – ETABS link (AISC)

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.

[Activate the link](#)

[Use the link](#)

[Update the project](#)

[Known limitations](#)

1 How to activate the link

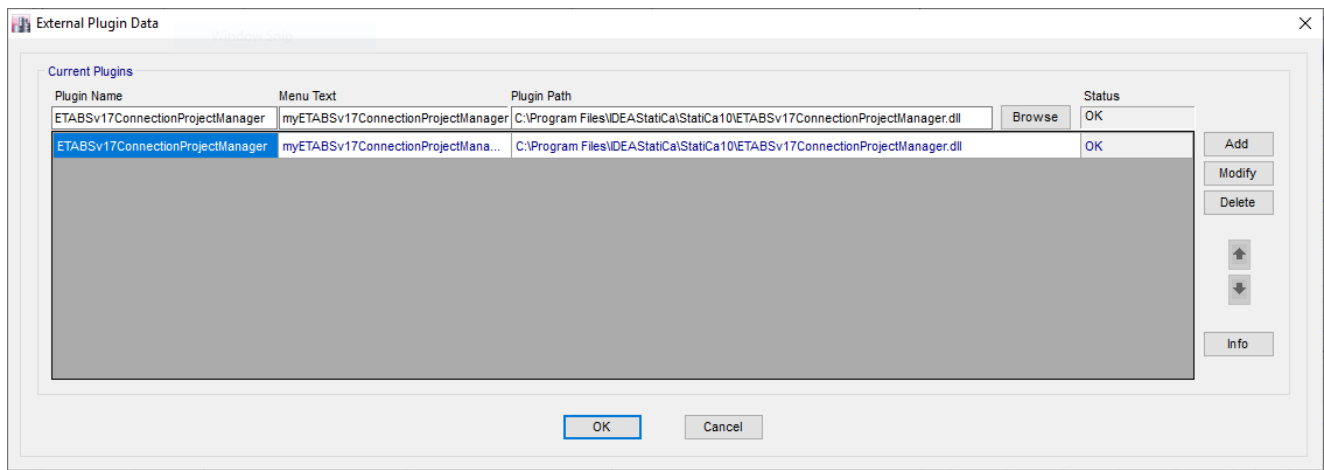
- Install the latest version of IDEA StatiCa, get it in the [Downloads](#).
- Make sure you are using a supported version of ETABS – updates are published in the [BIM section](#)

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

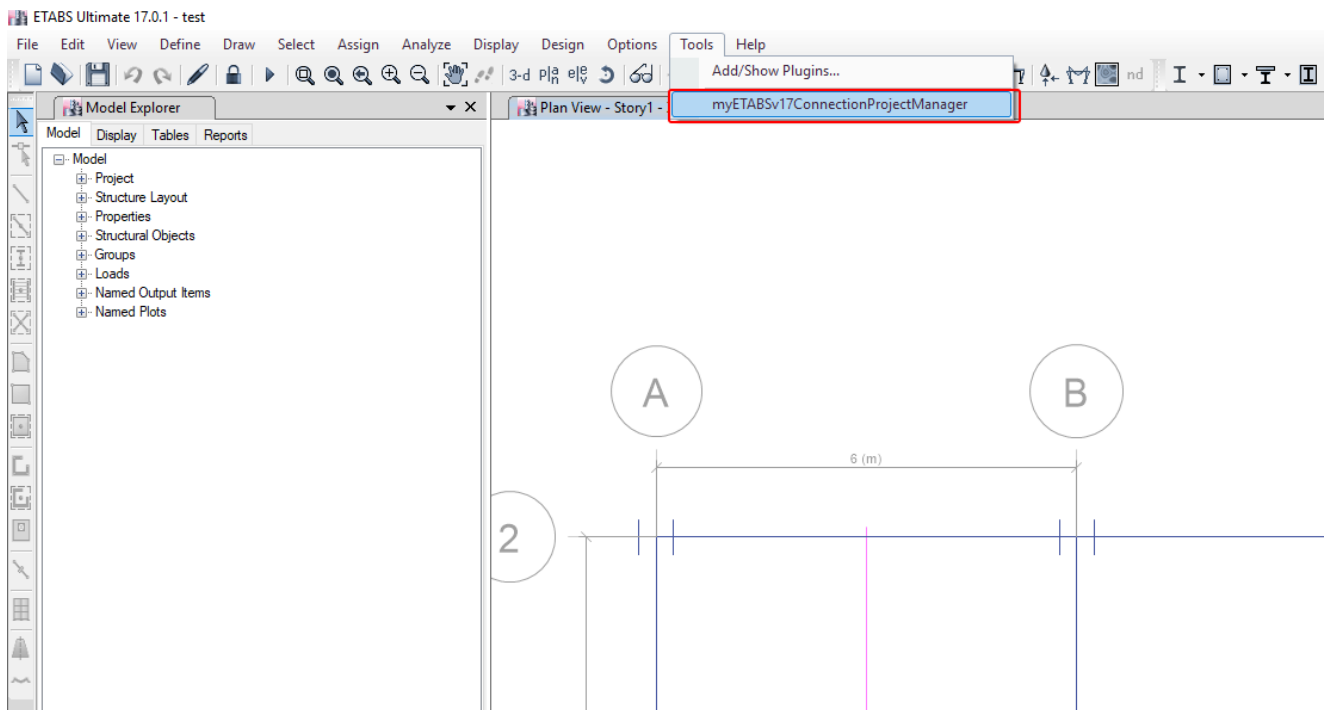
and click **Add**



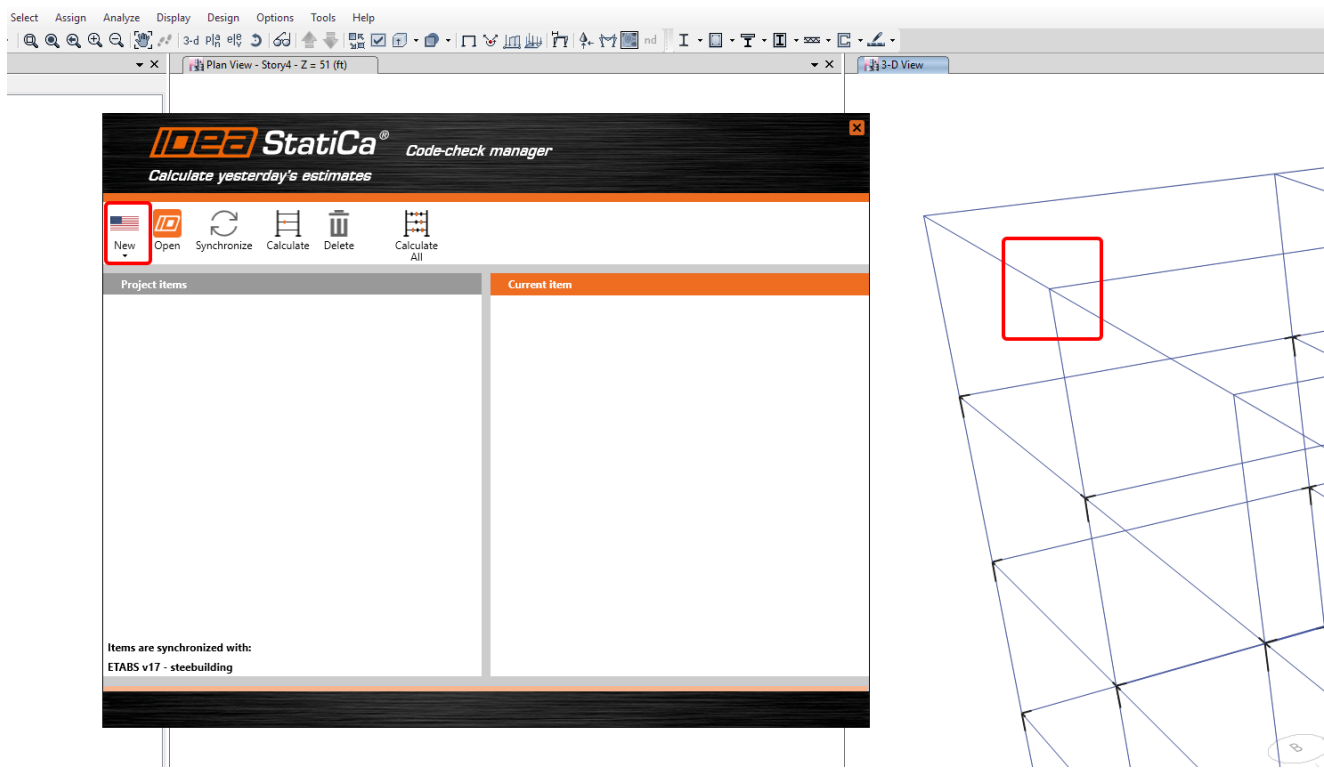
2 How to use the link

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**.

Connection design needs more data to be able to provide a proper design according to national codes. You can use default settings or define them in this wizard.

Design code: AISC

Type of structure: General structure

Default setting:

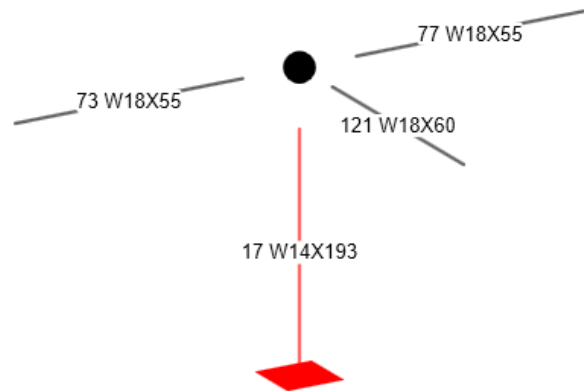
All load combinations are used for the design.

Load combinations are sorted into classes ULS, SLS etc.

Con 21-Node N1

Connected members:

	Cross-section		Role	Type
>	17 W14X193 (W14X193)		Bearing	Ended
	73 W18X55 (W18X55)			Ended
	77 W18X55 (W18X55)			Ended
	121 W18X60 (W18X60)			Ended



Connection design

< Previous

Next >

Cancel

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.

The screenshot shows the IES StatiCa CONNECTION software interface. The main window displays a 3D model of a steel joint. On the right side, there is a table titled "Dead (Load)" showing internal forces for four members. Below the table, there is a section for "Unbalanced forces" with a small table.

Member	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
> 17 W14X193 / End	-4.1	-0.1	-0.6	0.0	1.4	-0.1
73 W18X35 / End	-0.5	0.0	-1.3	0.0	1.7	0.0
77 W18X35 / End	-0.5	0.0	-1.4	0.0	1.8	0.0
121 W18X35 / End	-0.6	0.0	-1.3	0.0	1.4	0.0

X [kN]	Y [kN]	Z [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
0.0	0.0	0.0	0.0	0.0	0.0

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**.

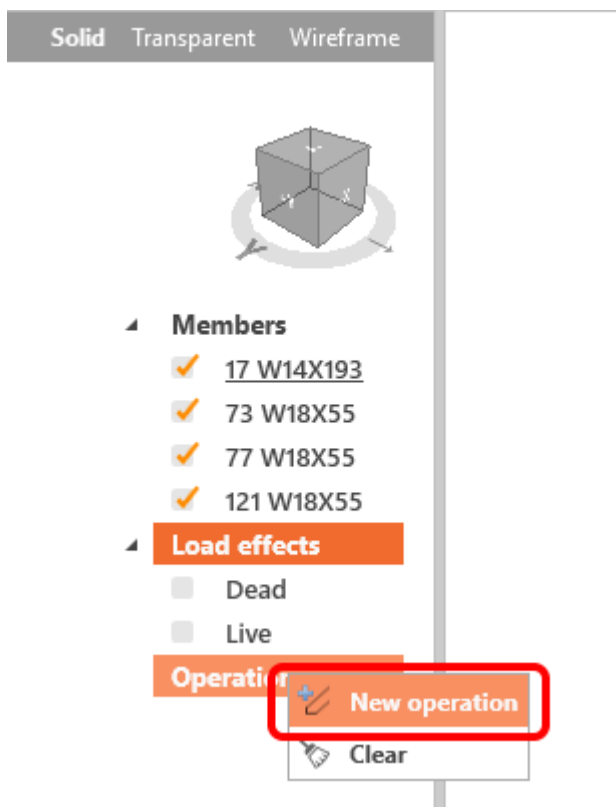
Units

Unit type	Unit	Precision	Format
▲ Main			
Length - Structure	in	3	Decimal
Length - Cross-section	in	3	Decimal
Plate or weld thickness, bolt hole diameter	in	1/16"	Imperial
Angle	°	1	Decimal
Force	kip	3	Decimal
Moment	kip.in	2	Decimal
Stress	ksi	1	Decimal
Coefficient	No symb...	2	Decimal
▲ Material			
Mass	lbm	0	Decimal
Unit Weight	pcf	0	Decimal
Coefficient of Thermal Expansion	-/°F	0	Decimal
▲ Result			
Rotation	mrad	1	Decimal
Strain	1e-4	1	Decimal
Axial Stiffness	kip	0	Decimal
Utilization	%	1	Decimal
Rotational Stiffness	kip-in/deg	1	Decimal

Metric
Imperial
Import
Export

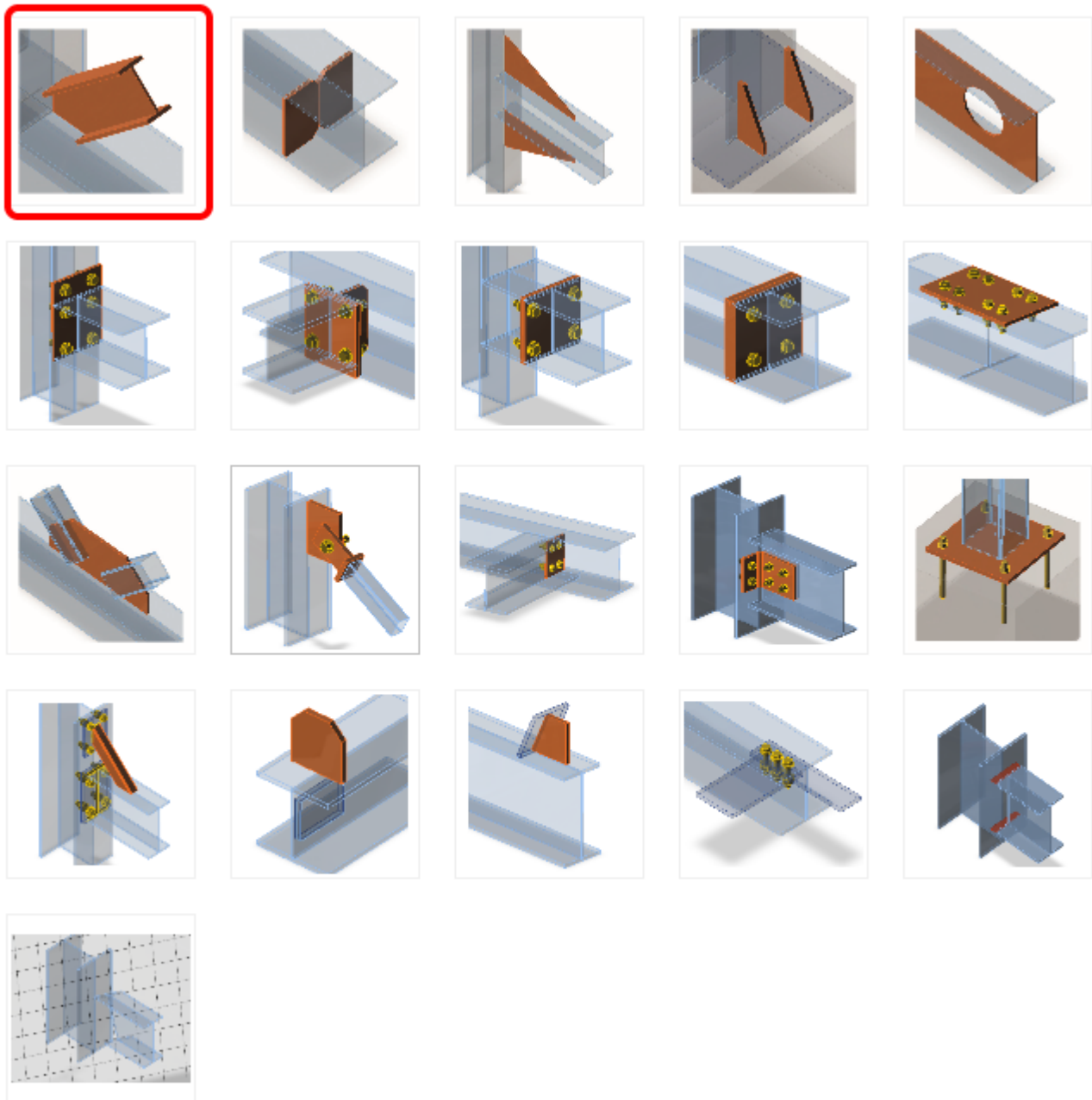
Apply

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**.



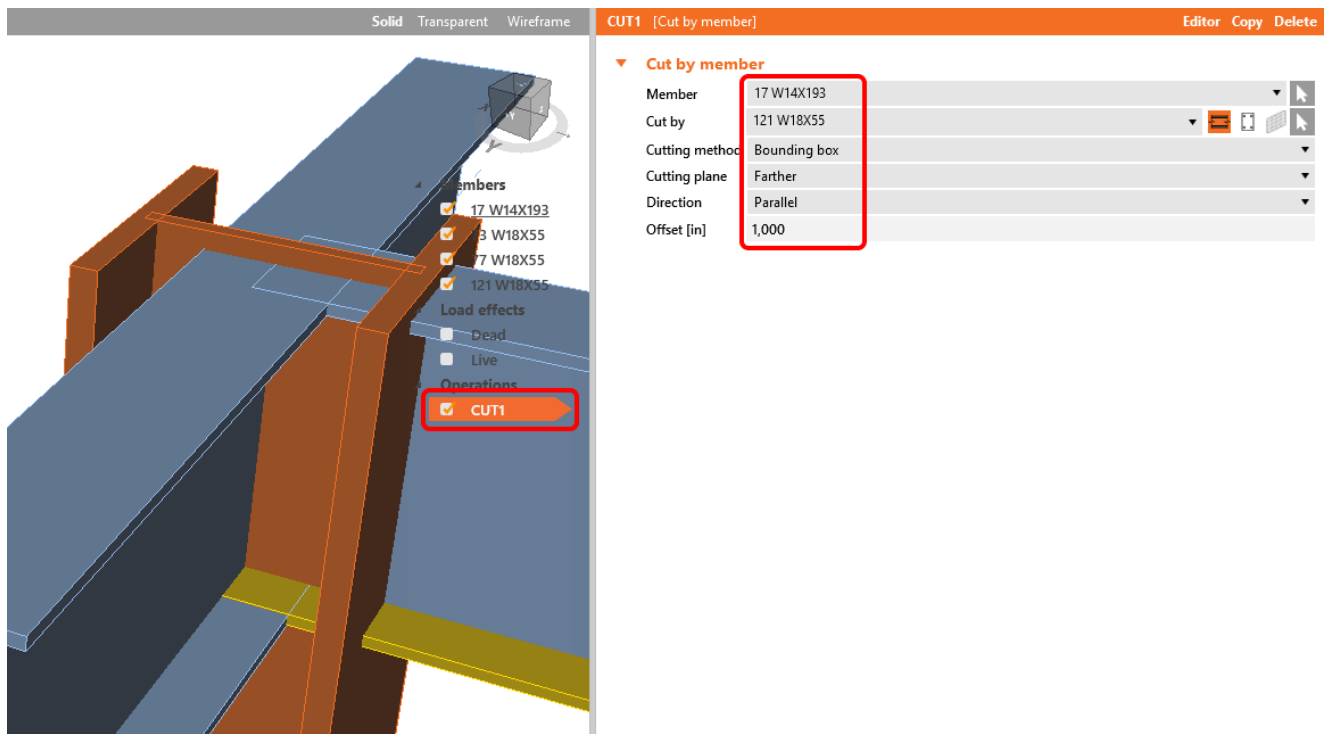
And we add operation **Cut**.

Select operation



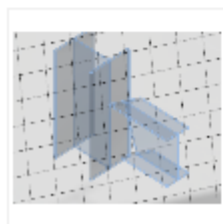
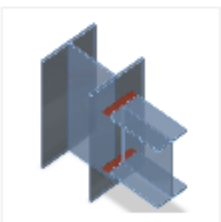
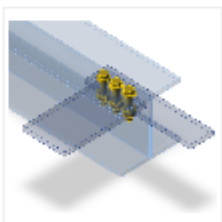
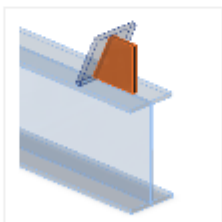
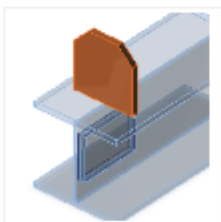
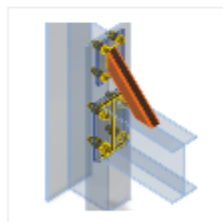
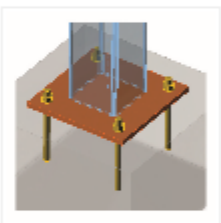
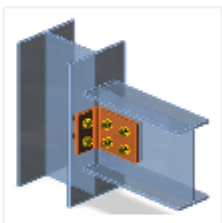
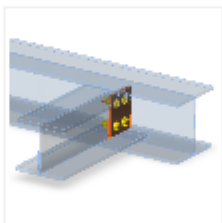
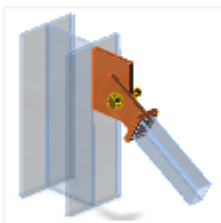
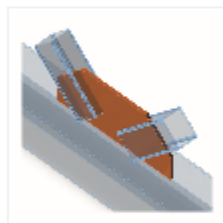
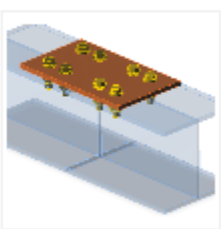
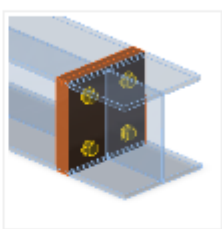
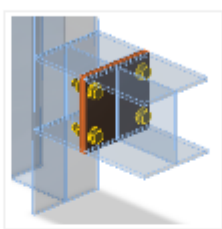
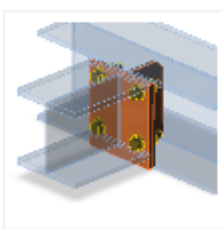
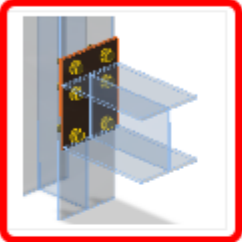
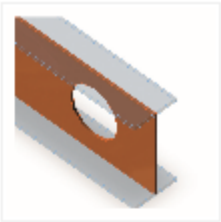
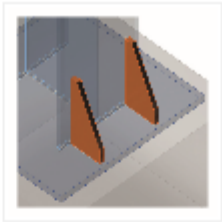
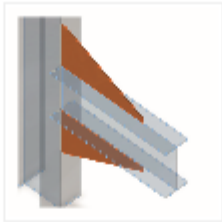
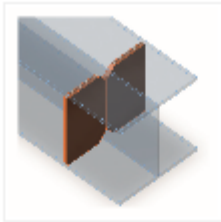
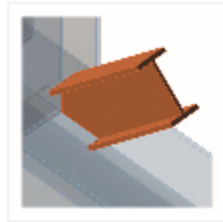
Cancel

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

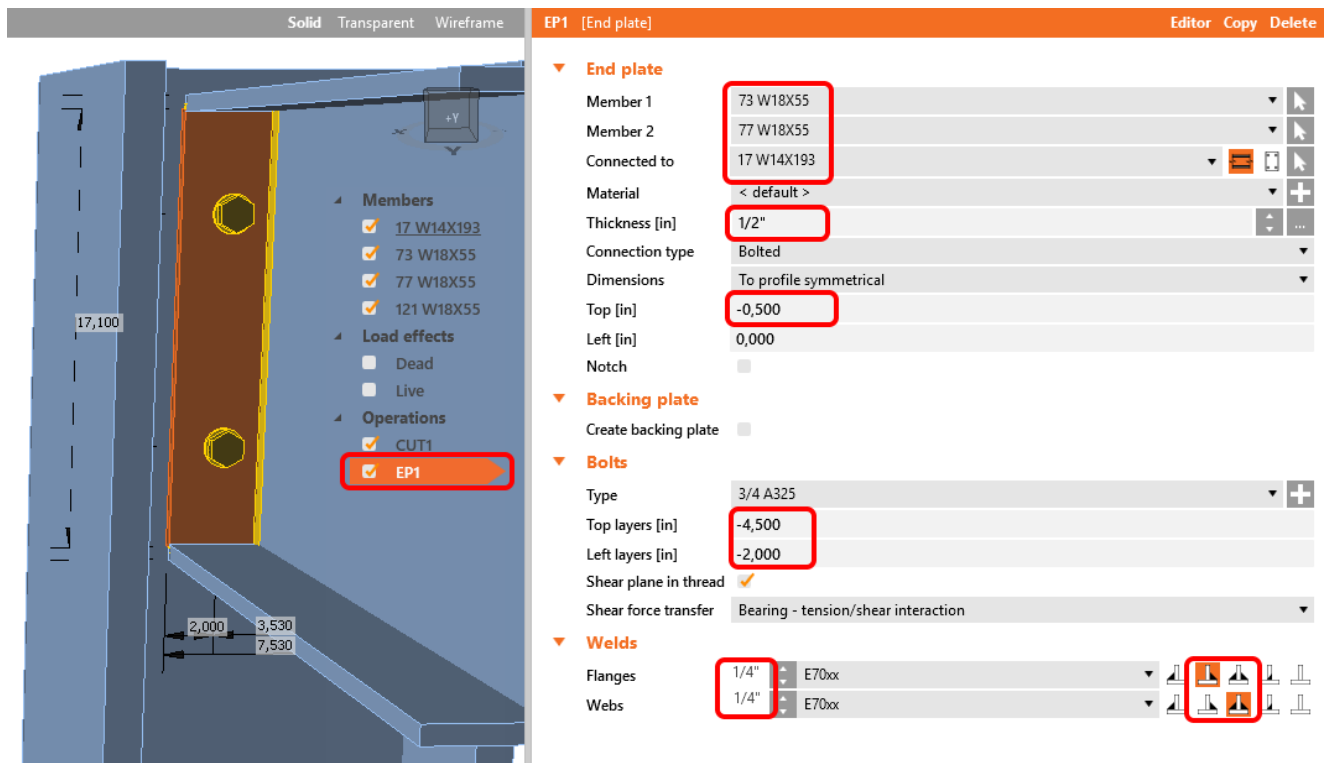
Bolt assembly

✕

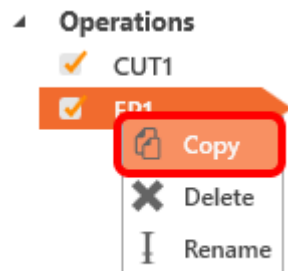
A307	1/2 A325
A325	5/8 A325
A325M	3/4 A325
A490	7/8 A325
A490M	1 A325
	1 1/8 A325
	1 1/4 A325
	1 3/8 A325
	1 1/2 A325

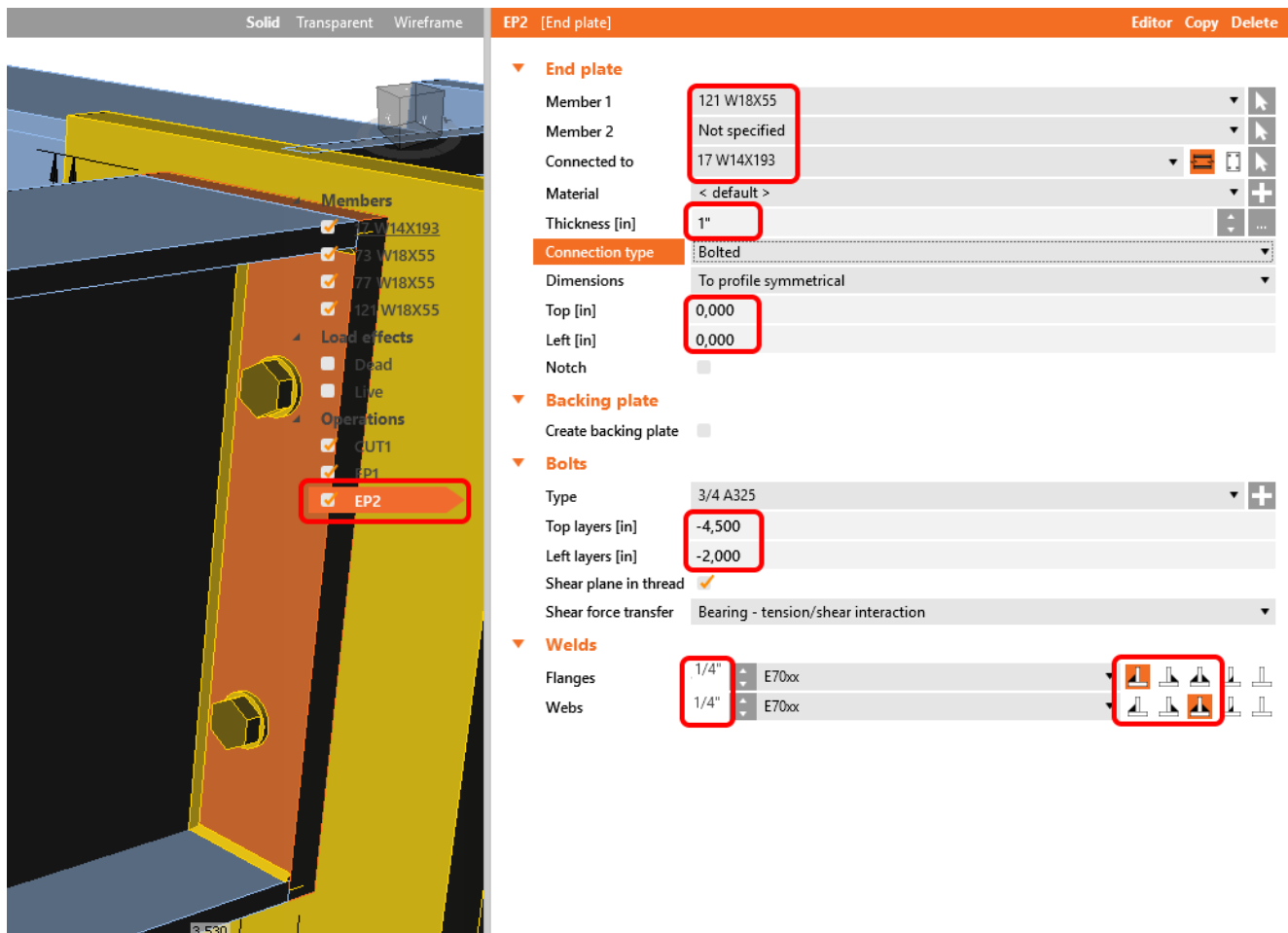
OK

Cancel



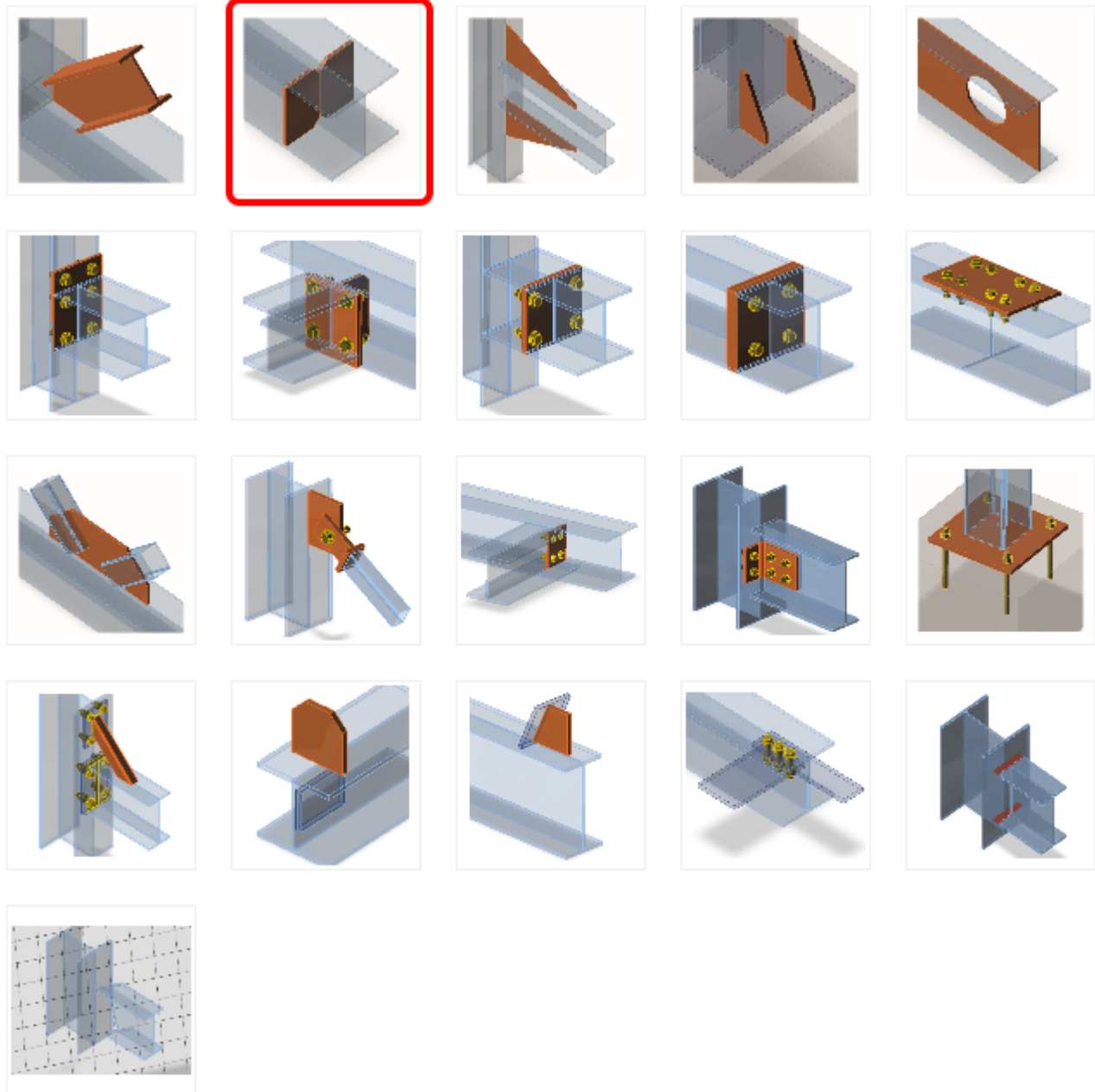
In the next step, **right-click** on **Operation EP1** and select **Copy**. Then change the values according next picture





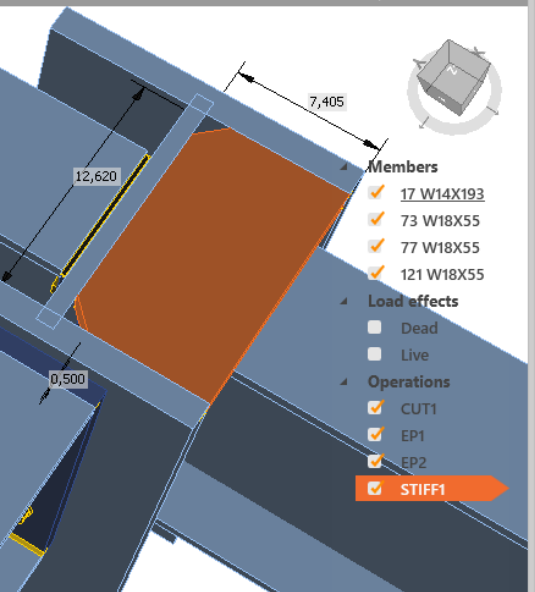
Next, we input a **Stiffener** and fill in the values below.

Select operation



Cancel

Solid Transparent Wireframe
STIFF1 [Stiffeners] Editor Copy Delete



Members

- ☒ 17 W14X193
- ☒ 73 W18X55
- ☒ 77 W18X55
- ☒ 121 W18X55

Load effects

- ☐ Dead
- ☐ Live

Operations

- ☒ CUT1
- ☒ EP1
- ☒ EP2
- ☒ STIFF1

Stiffeners

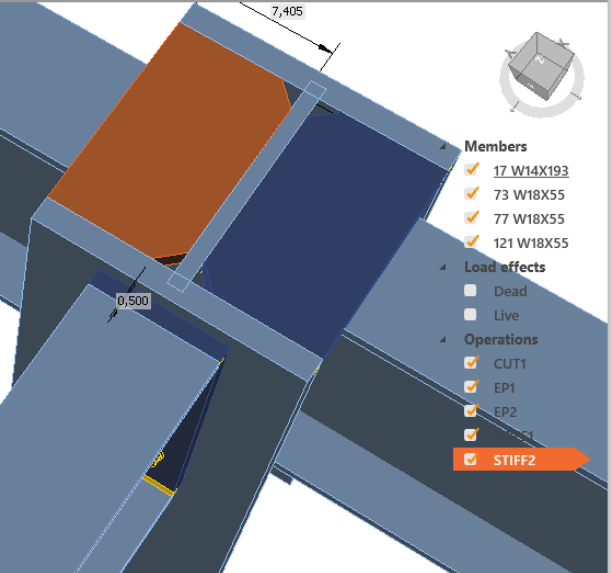
On member	17 W14X193
Related to	Not specified
Material	< default >
Thickness [in]	1"
Location	Rear
X - position [in]	0,500
α - Inclination [°]	0,0
Width [in]	0,000
Offset top [in]	0,000
Offset bottom [in]	0,000
Repeat count	1
Gap [in]	0,000
Chamfered corners	<input checked="" type="checkbox"/>
Chamfer cut size [in]	0,000

Welds

All welds 1/4" E70xx

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

Solid Transparent Wireframe
STIFF2 [Stiffeners] Editor Copy Delete



Members

- ☒ 17 W14X193
- ☒ 73 W18X55
- ☒ 77 W18X55
- ☒ 121 W18X55

Load effects

- ☐ Dead
- ☐ Live

Operations

- ☒ CUT1
- ☒ EP1
- ☒ EP2
- ☒ STIFF2

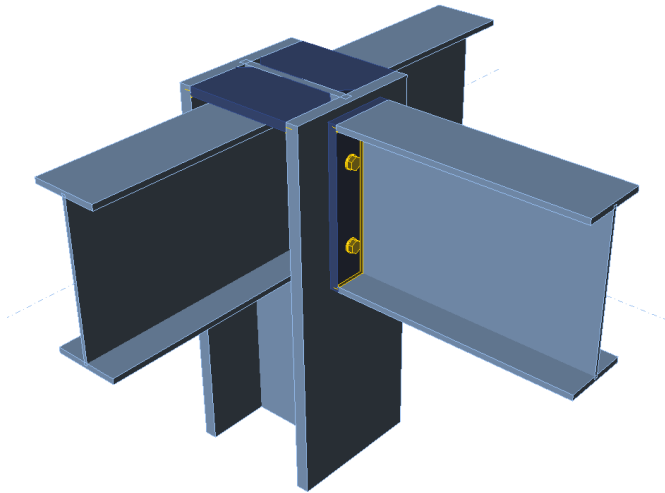
Stiffeners

On member	17 W14X193
Related to	Not specified
Material	< default >
Thickness [in]	1"
Location	Front
X - position [in]	0,500
α - Inclination [°]	0,0
Width [in]	0,000
Offset top [in]	0,000
Offset bottom [in]	0,000
Repeat count	1
Gap [in]	0,000
Chamfered corners	<input checked="" type="checkbox"/>
Chamfer cut size [in]	0,000

Welds

All welds 1/4" E70xx

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.

StatiCa CONVECTION

Project Design Check Report Materials

Can 21

Project Items

Analysis 100.0%
Plates 0.9 < 5%
Bolts 89.3 < 100%
Welds 94.4 < 100%
Buckling Not calculated

Members

- 17 W34X193
- 77 W18X55
- 73 W18X55
- 121 W18X55

Load effects

- Dead
- Live

Operations

- CUT1
- EP1
- EP2
- STIFF1
- STIFF2

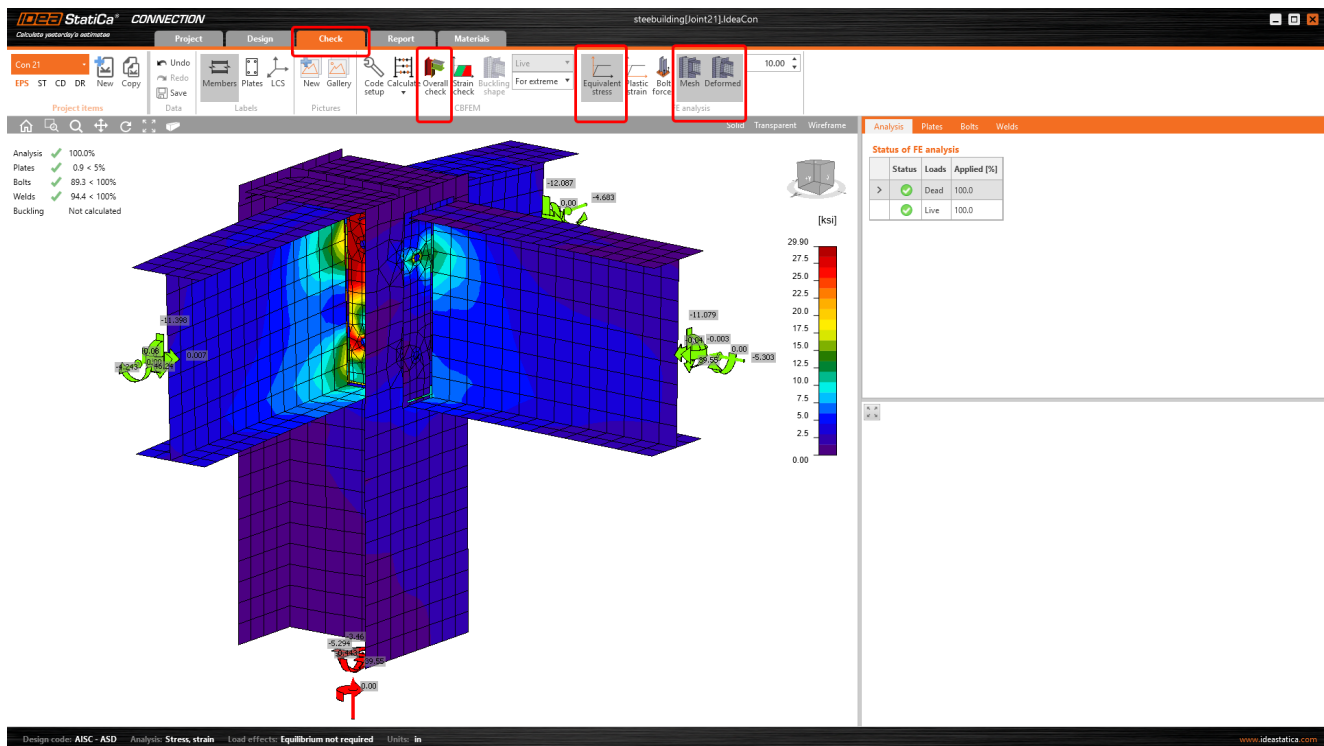
Member	N [kip]	Vy [kip]	Vz [kip]	Mx [kip-ft]	My [kip-ft]	Mz [kip-ft]
77 W18X55 / End	-4.683	-0.003	-12.087	0.00	49.70	-0.04
73 W18X55 / End	-4.243	0.007	-11.398	0.00	46.24	0.06
121 W18X55 / End	-5.303	-0.003	-11.079	0.00	39.55	-0.04

Values in disabled cells are not taken into account in CBFEM analysis. Members can be loaded only by that components of internal forces which are defined in member 'Model type'.

Design code: AISC - ASD Analysis: Stress, strain Load effects: Equilibrium not required Units: in

www.ideastatica.com

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.



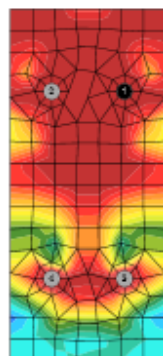
All values can be reviewed in detailed in the tables and 2D window. For example, to display check of bolts select tab **Bolts** tab.

Check of bolts for extreme load effect

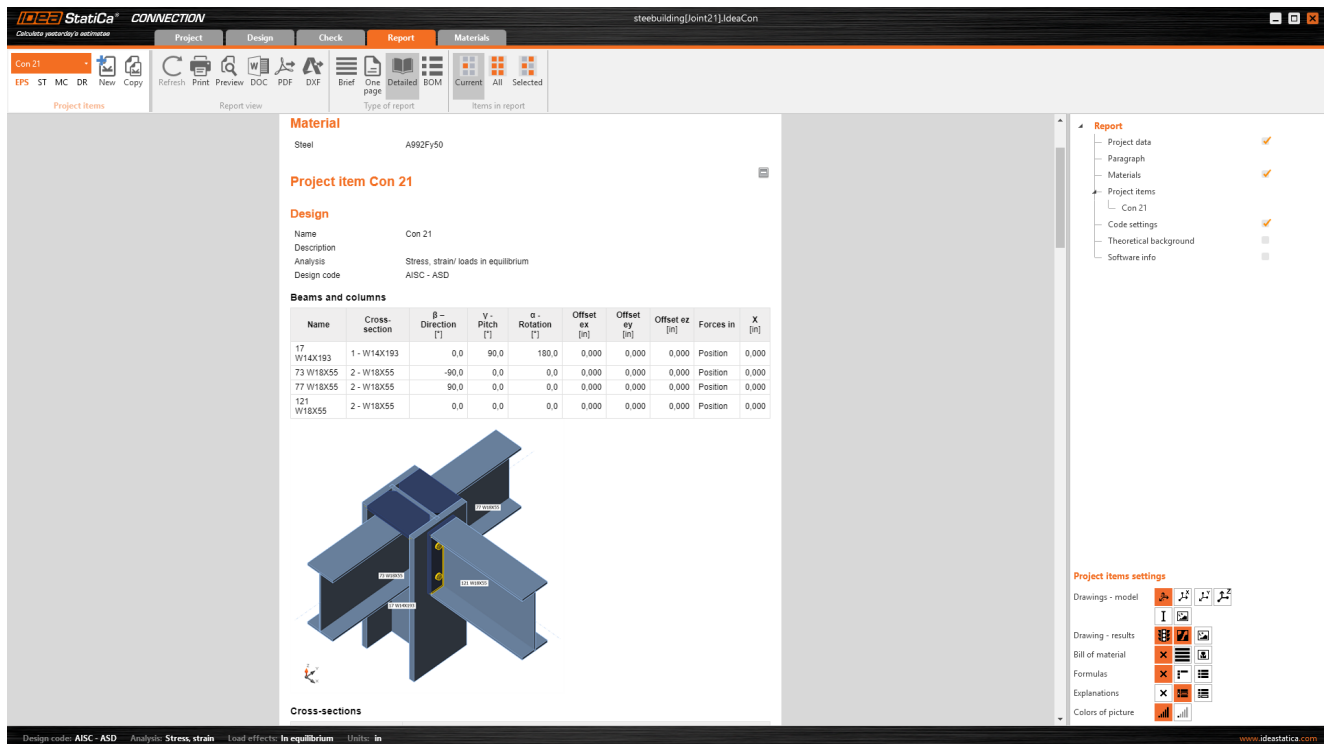
	Status	Item	Loads	Ft [kip]	V [kip]	Bearing Rn/Ω [kip]	Utt [%]	Uts [%]	Uts [%]
-	✓	B1	Live	17.674	2.285/1.909	52.062	89.0	19.2	-
<p>Tension resistance check (AISC 360-16: J3-1)</p> $\frac{R_n}{\Omega} = \frac{F_u A_b}{\Omega} = 19.863 \text{ kip} \geq F_t = 17.674 \text{ kip}$ <p>Where:</p> <p>$F_u = 89.9 \text{ ksi}$ – nominal tensile stress from AISC 360-16 Table J3.2</p> <p>$A_b = 0.442 \text{ in}^2$ – gross bolt cross-sectional area</p> <p>$\Omega = 2.00$ – safety factor for tension and shear</p>									
+	✓	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

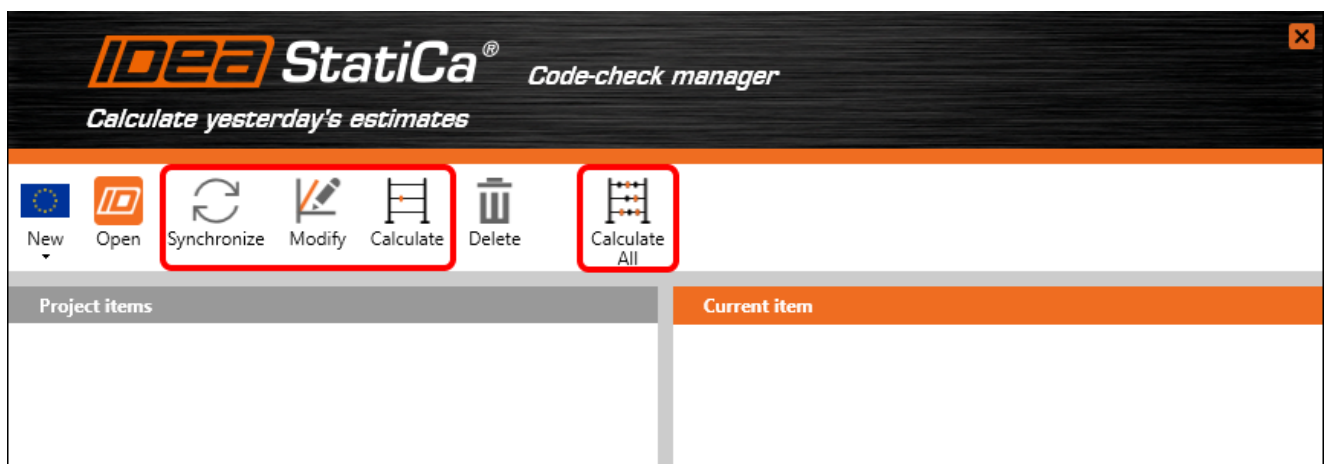


At last we go to the tab **Report**. IDEA StatiCa offers fully customizable report to print out or save in editable format.



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.

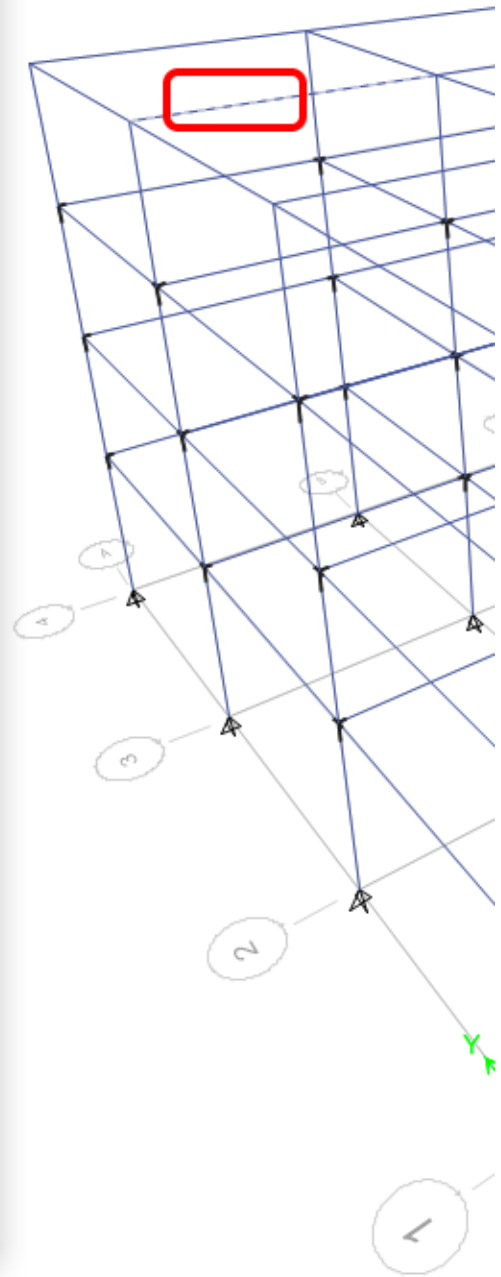
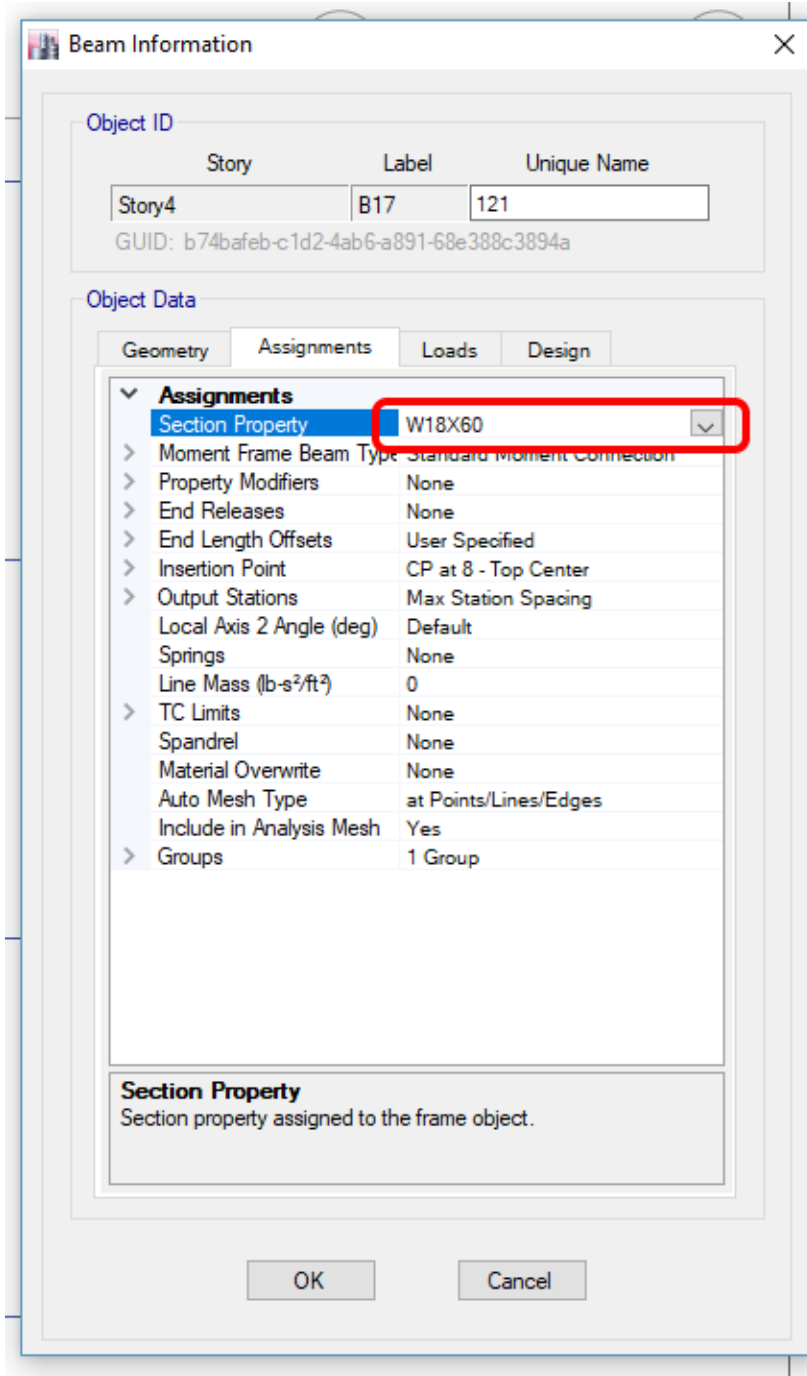
Calculate all - Synchronize and calculate all items and provide new set of results.

Note

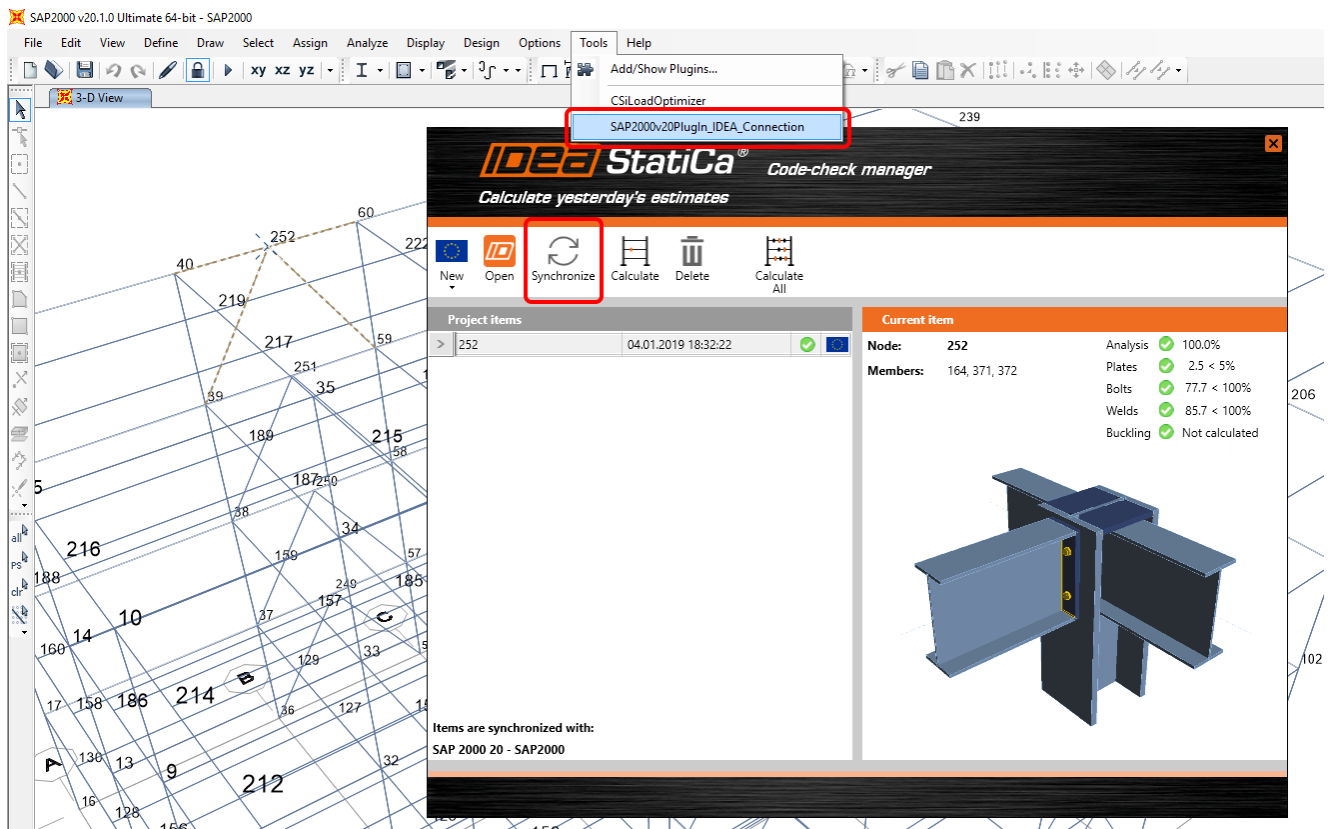
Kindly be aware that IDEA StatiCa syncs with a model of the 3rd party application, not the other way around. If we **add operations in IDEA StatiCa** and then use the options described above (Synchronize; Modify; Calculate; Calculate all), the additionally added operations will be deleted.

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**.

Connection design needs more data to be able to provide a proper design according to national codes. You can use default settings or define them in this wizard.

Design code: AISC

Type of structure: General structure

Default setting:

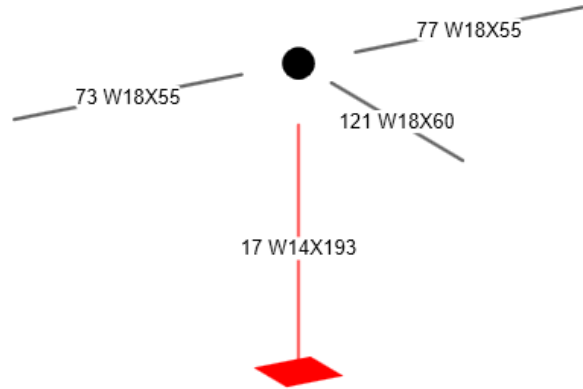
All load combinations are used for the design.

Load combinations are sorted into classes ULS, SLS etc.

Con 21-Node N1

Connected members:

	Cross-section		Role	Type
>	17 W14X193 (W14X193)		Bearing	Ended
	73 W18X55 (W18X55)			Ended
	77 W18X55 (W18X55)			Ended
	121 W18X60 (W18X60)			Ended



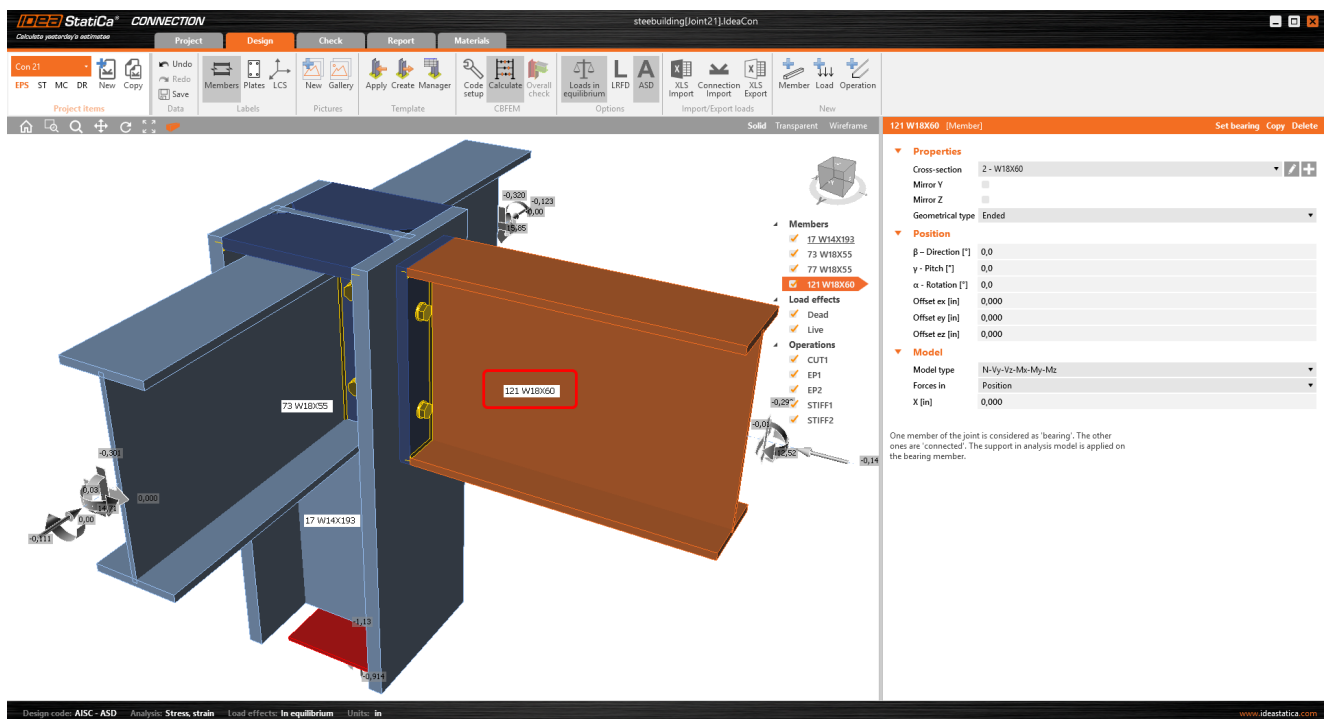
Connection design

< Previous

Next >

Cancel

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

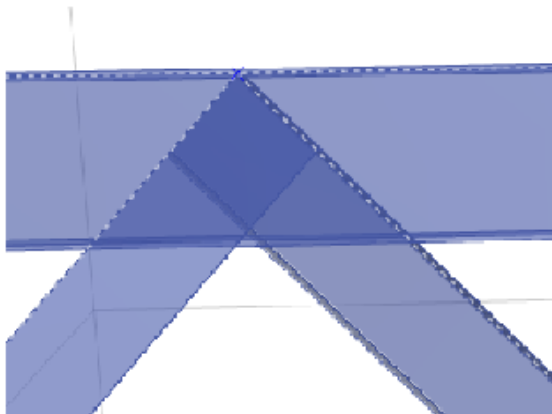


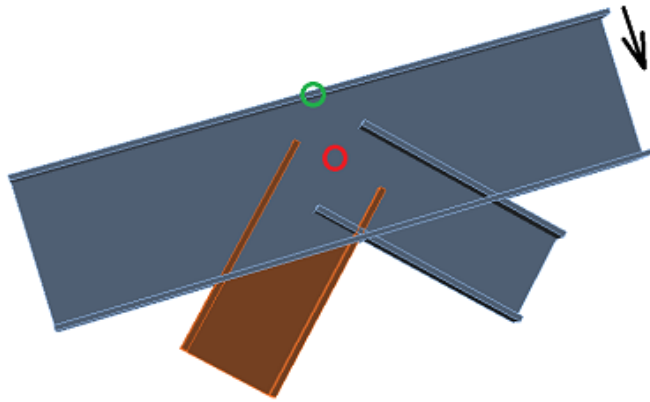
We have imported, designed and code-checked a steel joint according to AISC.

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





Frame Assignment - Insertion Point

Cardinal Point

8 (Top Center) ▾

☐ Mirror about Local 2
 ☐ Mirror about Local 3

Frame Joint Offsets from Cardinal Point

Coordinate System

Local ▾

	End-I	End-J	
1	0	0	in
2	0	0	in
3	0	0	in

☒ Do not transform frame stiffness for offsets from centroid for non-P/T floors

Reset Defaults

OK

Close

Apply

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