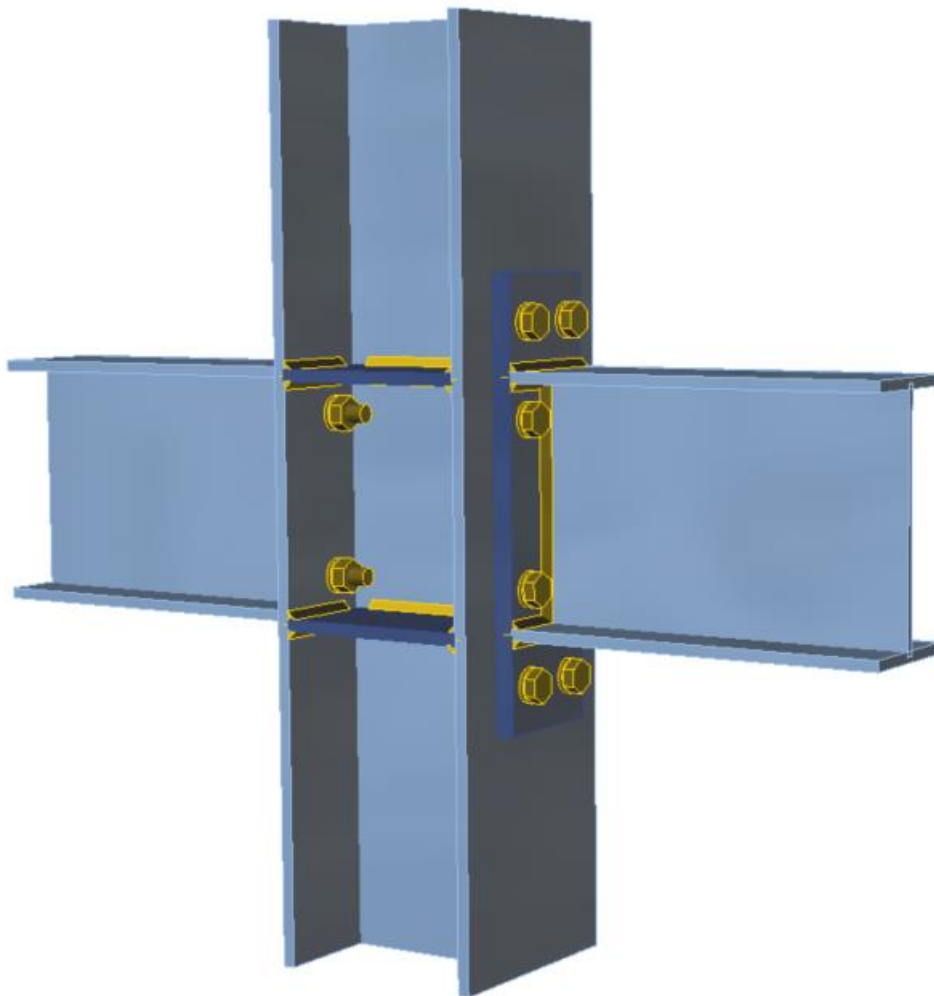


IDEA StatiCa Steel - Tutorial

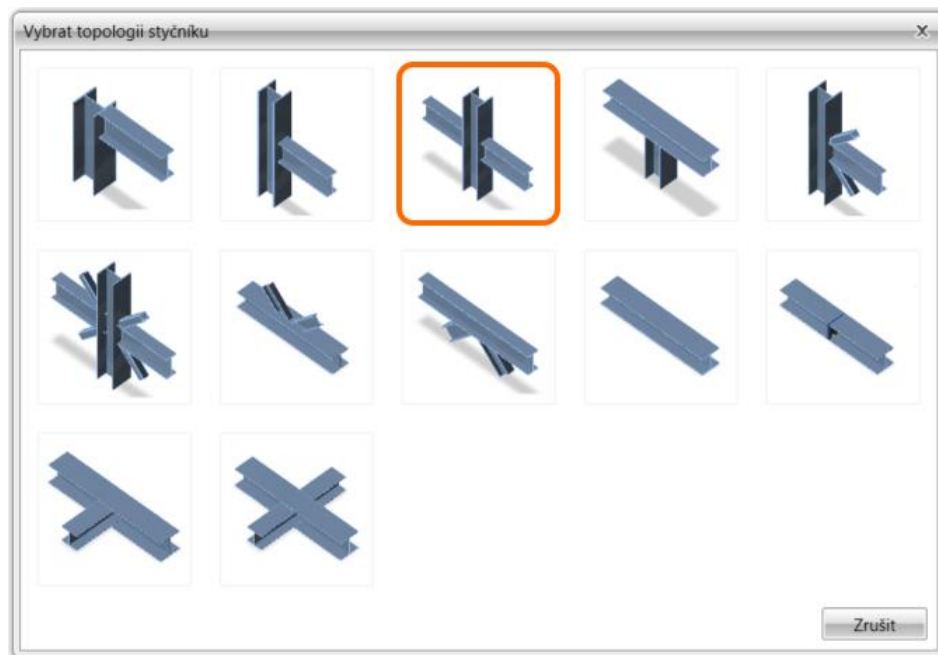
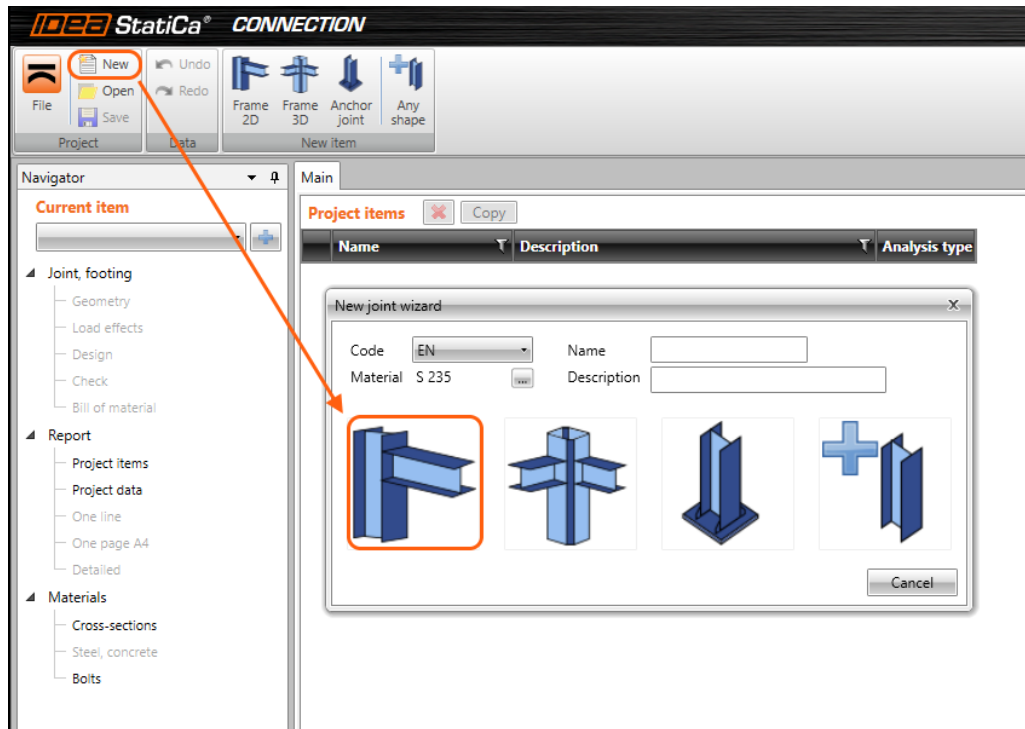


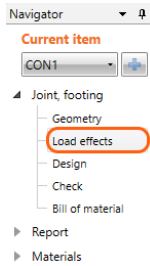
Stiffness analysis

This tutorial will show how to use software IDEA StatiCa to model, design and check a structural steel joint, example being footing with a diagonal.

New project

We create a new project by clicking **New**. Wizard window is opened. We select **2D frame** and **Column with two beams** topology.



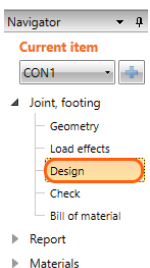
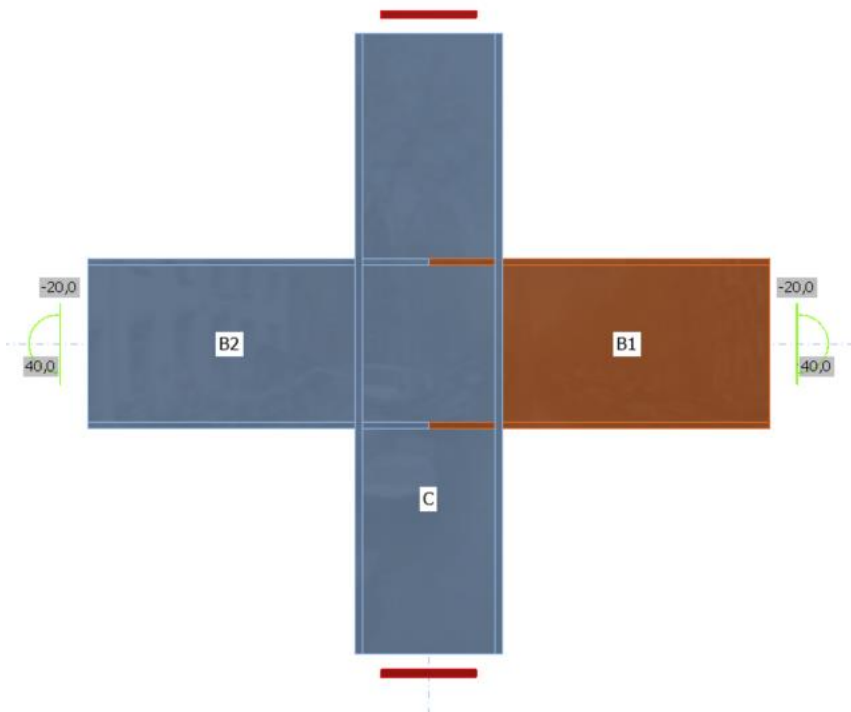


Load effects


One load effect was automatically added by the wizard. We define it by inputting values into the table.

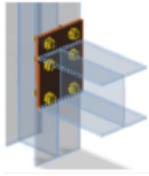
Internal forces

Member	Position	X [mm]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
B1	End	0	0,0	0,0	-20,0	0,0	40,0	0,0
B2	End	0	0,0	0,0	-20,0	0,0	40,0	0,0



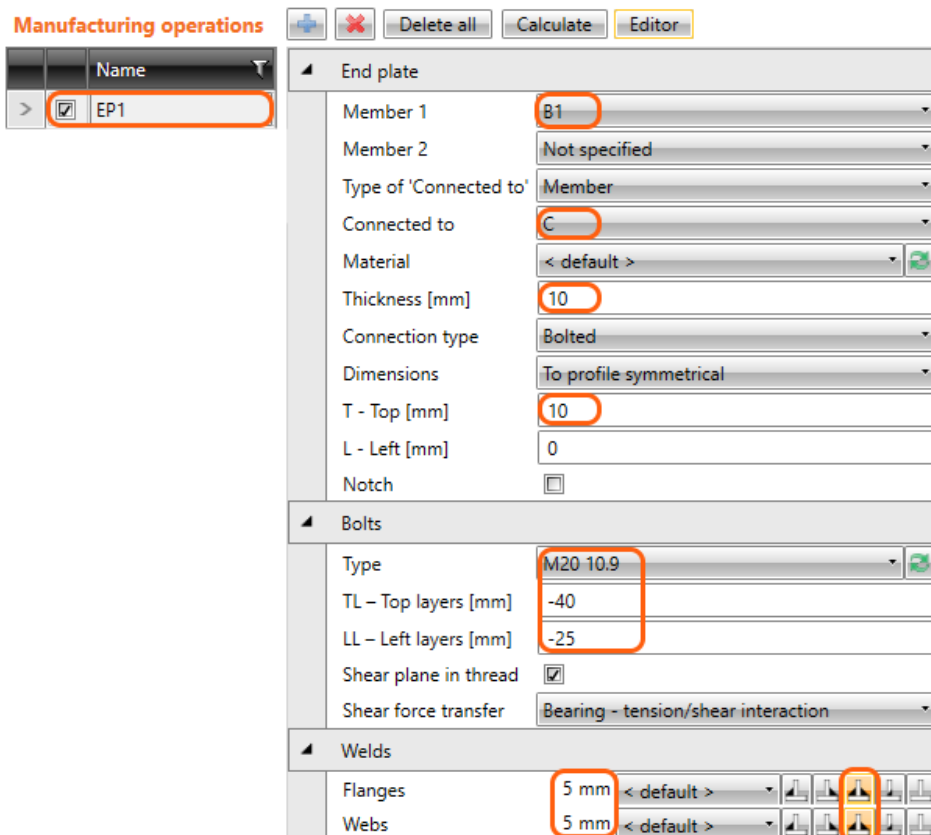
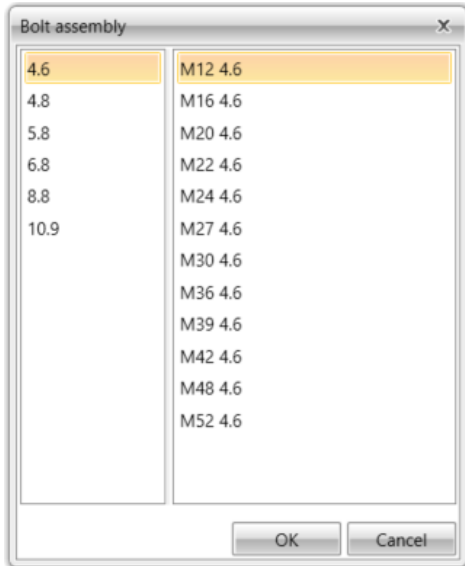
Design

We will define a set of manufacturing operations to model connections between members. A new operation can be added by  button.



End plate

We select default bolt type.





Copy EP1 operation by button from the ribbon and set that on member B2.

Manufacturing operations

	Name
<input checked="" type="checkbox"/>	EP1
<input checked="" type="checkbox"/>	EP2

End plate

Member 1: B2

Member 2: Not specified

Type of 'Connected to': Member

Connected to: C

Material: < default >

Thickness [mm]: 10



Stiffeners

Manufacturing operations

	Name
<input checked="" type="checkbox"/>	EP1
<input checked="" type="checkbox"/>	EP2
<input checked="" type="checkbox"/>	STIFF1

Stiffeners

M – on member: C

R – related to: B1

Position: Both

Material: < default >

Thickness [mm]: 10

Location: Both

X - position [mm]: 0

α - Inclination [°]: 0,0

B – width [mm]: 0

Offset top [mm]: 0

Offset bottom [mm]: 0

Repeat count: 1

Gap [mm]: 0

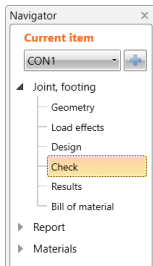
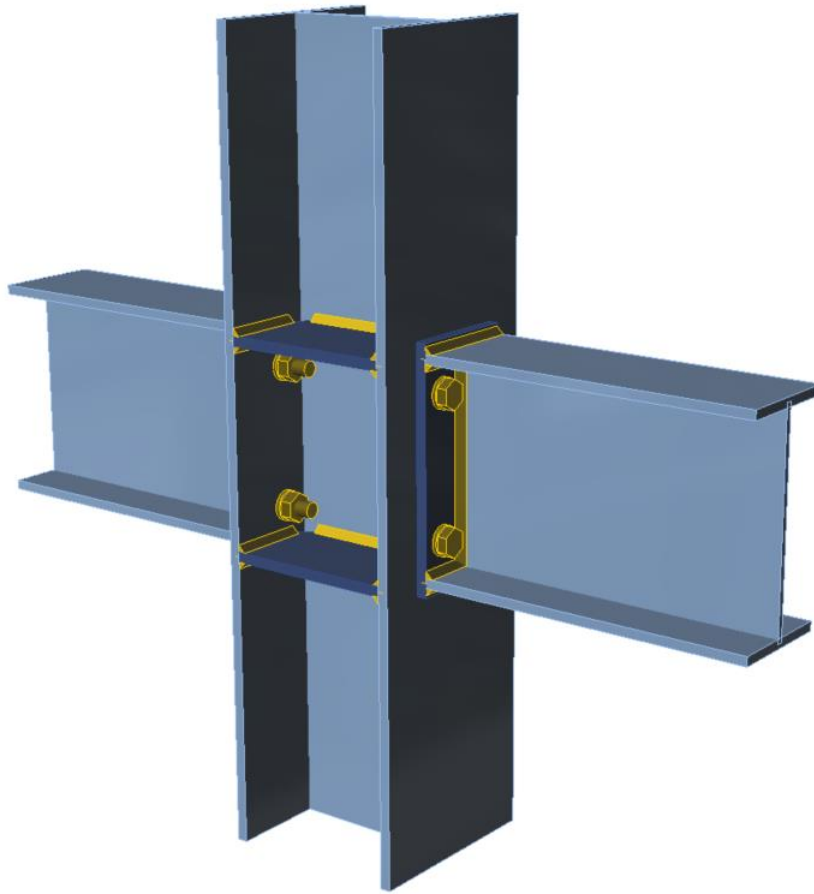
Chamfered corners: ☒

Chamfer cut size [mm]: 0

Welds


All welds: 5 mm < default >

Let's check defined operations of the joint.

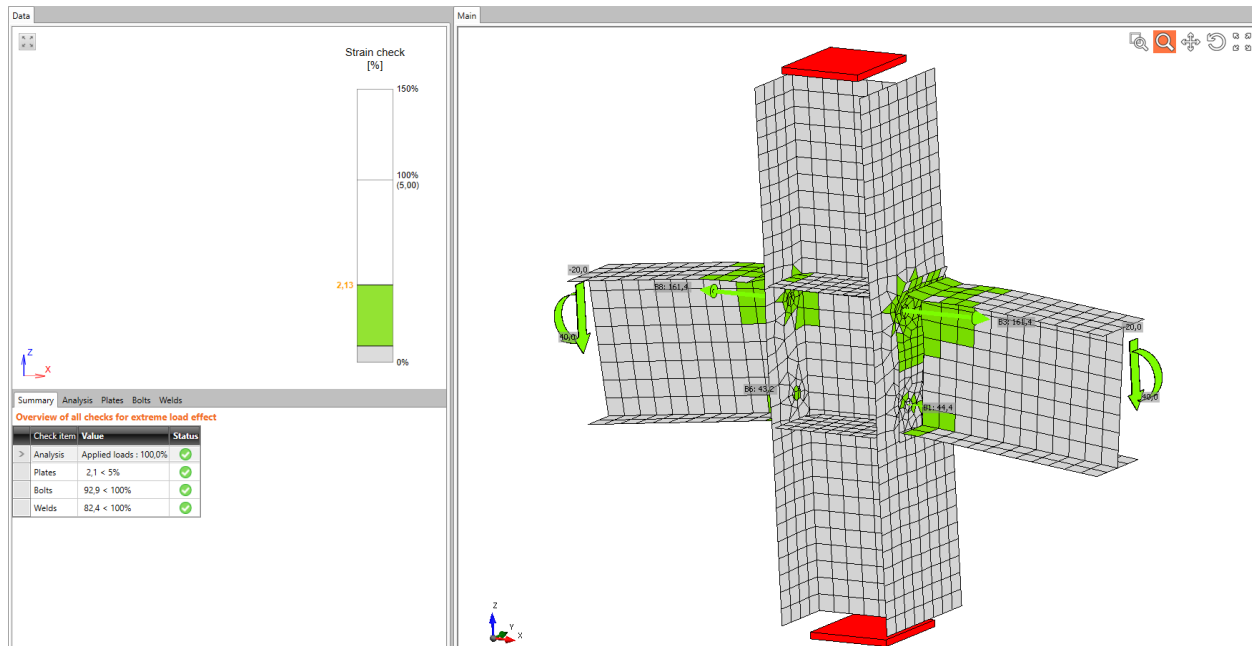


Check of a structural steel joint

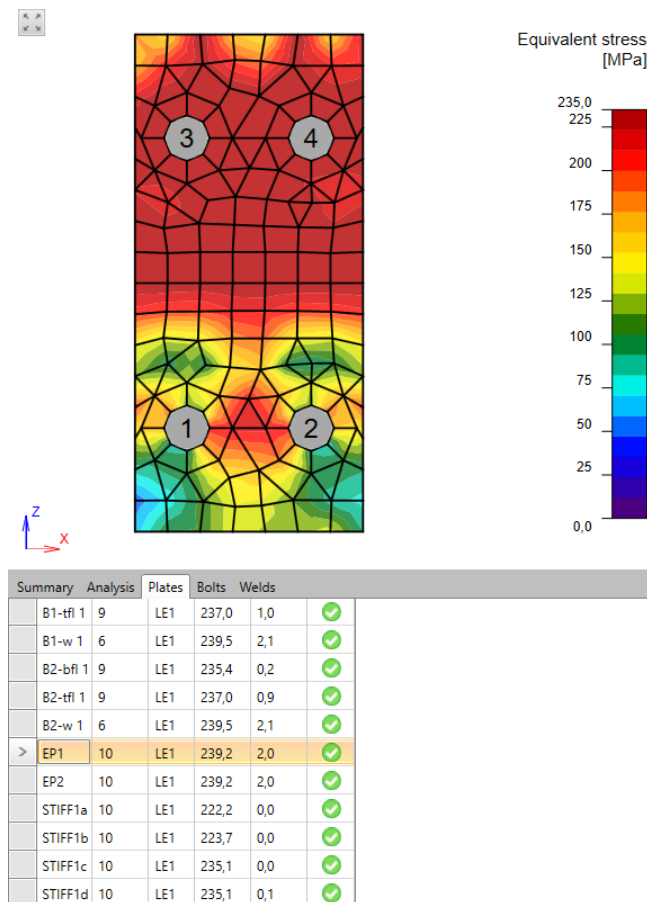


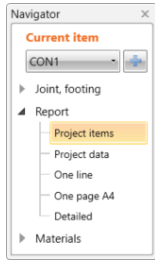
Nonlinear analysis is started by icon  from the top ribbon. Analysis model is automatically generated, calculation is performed and we can check results.

We activate **Strain check**, **Bolt forces**, **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 checked in detailed in the tables and 2D window. For example to display check of plates and stresses we select tab **Plates** and icon **Equivalent stress** from the ribbon.





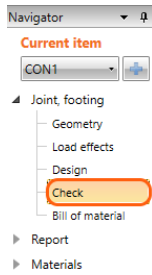
Stiffness analysis settings

To change the analysis type to stiffness we go to **Report** → **Project items** and click by Analysis type of item CON1 to change that to **Stiffness**.

Project items   

Name	Description	Report	Analysis type
> CON1		Yes	Stiffness

And we also set beam **B1** as **Analysed member** in **Geometry**.



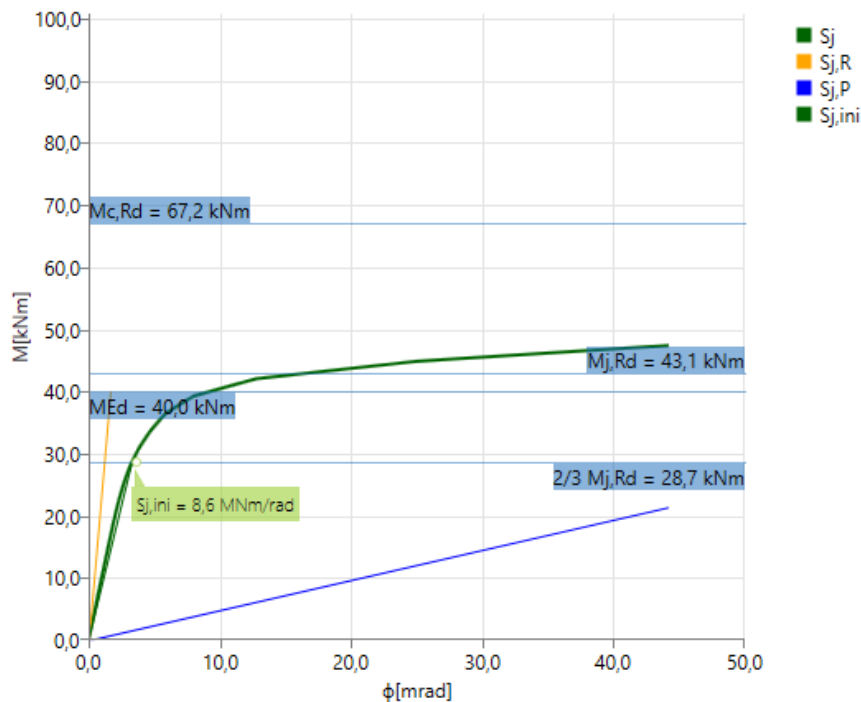
Stiffness analysis



Stiffness analysis is started by icon from the top ribbon.

All values and graph can be checked in detailed by selection of tab **Rotational stiffness**.

Stiffness diagram $M_y - \phi_y$, LE1






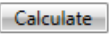

Summary Rotational stiffness

Rotational stiffness of joint component

	Item	Comp.	Loads	MEd [kNm]	Mj,Rd [kNm]	Sj,ini [MNm/rad]	Sjs [MNm/rad]	Φ [mrad]	Φ_c [mrad]	L [m]	Sj,R [MNm/rad]	Sj,P [MNm/rad]	Class
>	B1	My	LE1	40,0	43,1	8,6	4,4	9,0	44,2	6,00	24,3	0,5	Semi-rigid

Modification of connection

Let's modify end plate EP1 to increase rigidity of the connection.

Manufacturing operations     

	Name
> <input checked="" type="checkbox"/>	EP1
<input checked="" type="checkbox"/>	EP2
<input checked="" type="checkbox"/>	STIFF1

End plate

Member 1: B1

Member 2: Not specified

Type of 'Connected to': Member

Connected to: C

Material: < default >

Thickness [mm]: 16

Connection type: Bolted

Dimensions: To profile symmetrical

T - Top [mm]: 80

L - Left [mm]: 0

Notch: ☐

Bolts

Type: M20 10.9

TL - Top layers [mm]: -40 40

LL - Left layers [mm]: -25

Shear plane in thread: ☒

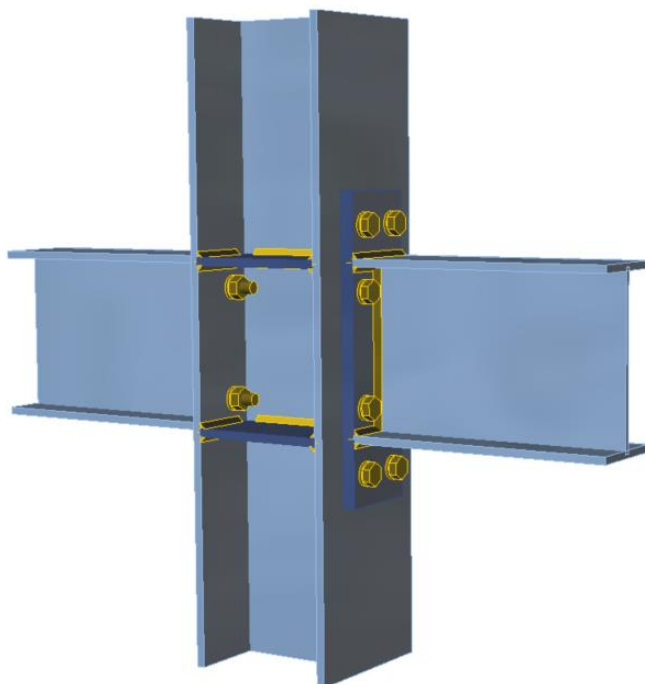
Shear force transfer: Bearing - tension/shear interaction

Welds

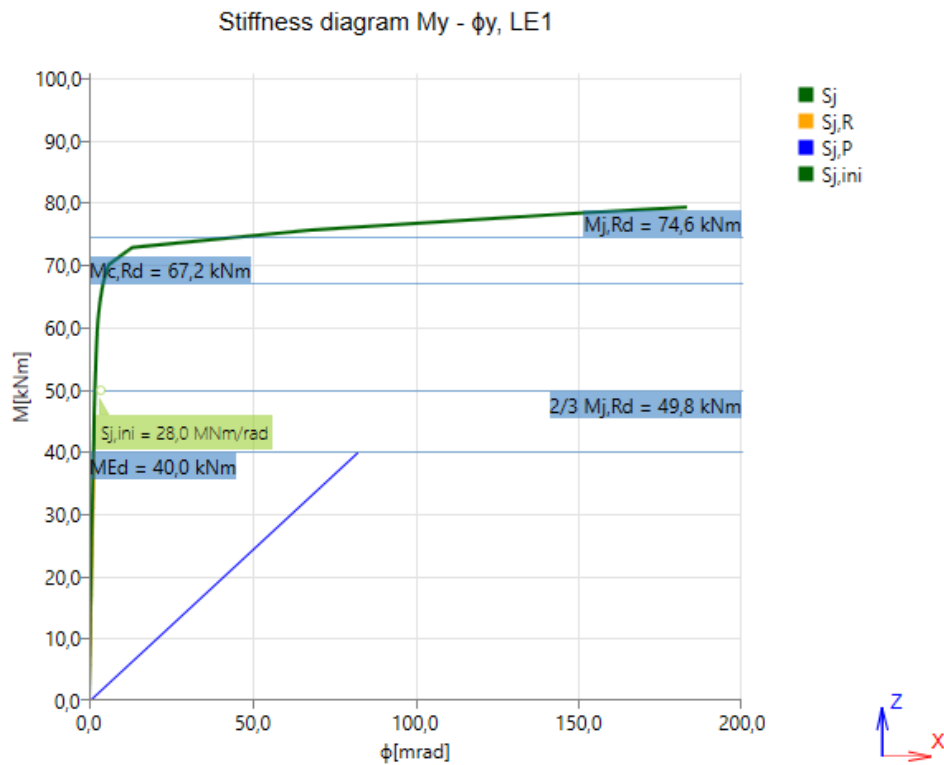
Flanges: 5 mm < default >

Webs: 5 mm < default >

And check updated shape of the joint.



And run stiffness analysis again.

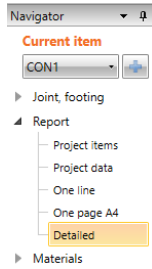


Summary

Rotational stiffness

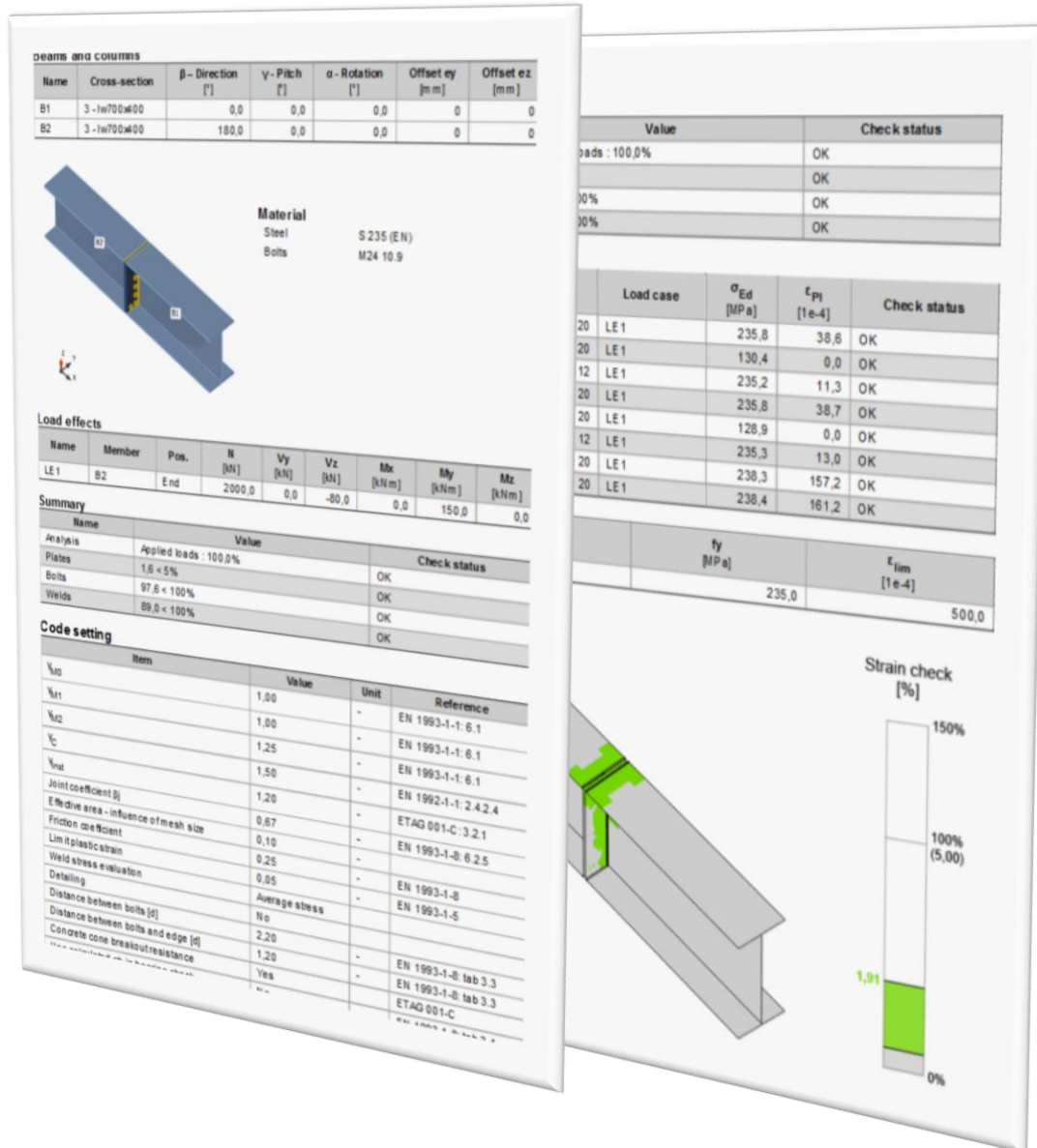
Rotational stiffness of joint component

	Item	Comp.	Loads	M _{Ed} [kNm]	M _{j,Rd} [kNm]	S _{j,ini} [MNm/rad]	S _{j,s} [MNm/rad]	Φ [mrad]	Φ _c [mrad]	L [m]	S _{j,R} [MNm/rad]	S _{j,P} [MNm/rad]	Class
>	B1	My	LE1	40,0	74,6	28,0	29,9	1,3	183,3	6,00	24,3	0,5	Rigid



Report

IDEA StatiCa offers three types of output reports – one line, 1 page and detailed.



Structural steel joint was modelled, designed and checked

Thank you for spending time on this example. For further information please visit our website or drop us an email to info@idea-rs.com.

IDEA StatiCa team

Steel connection design reinvented – any topology, any loading, in minutes. Check of joint/connections acc. to EC/AISC. Unique CBFEM method. Get more resources at www.idea-rs.com and www.ideastatica.com