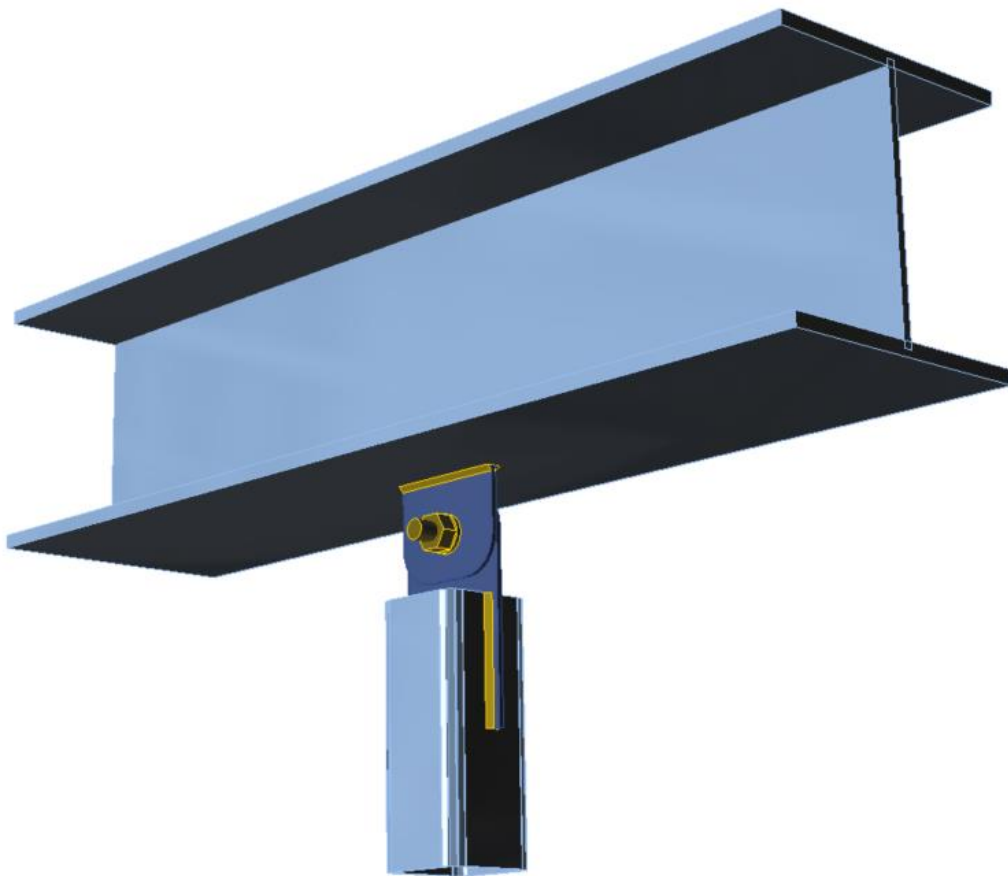# IDEA StatiCa Steel - Tutorial

Buckling analysis

Welcome to IDEA StatiCa tutorial. We will show how to use software IDEA StatiCa to model, design and check buckling of a structural steel joint, example being connection of strut to beam.
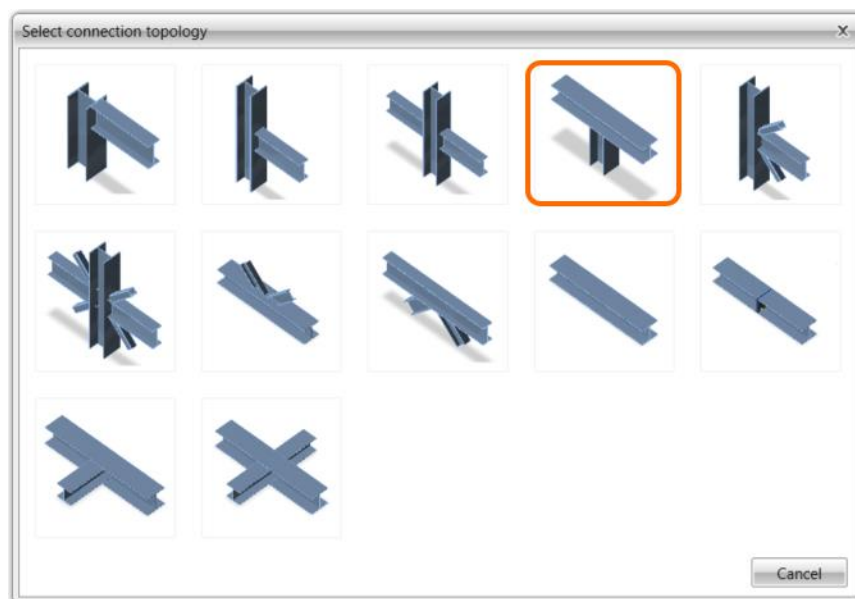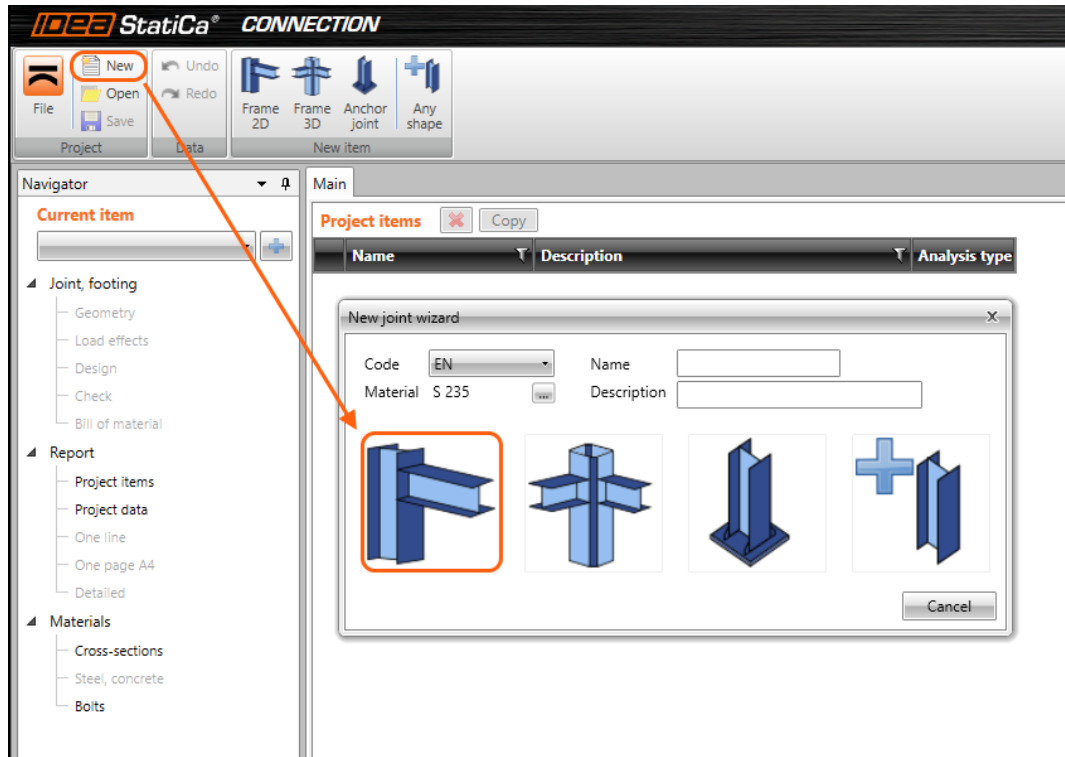
Launching application

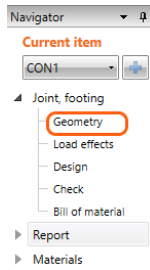Let's launch IDEA StatiCa and select application **Connection**.

New project

We create a new project by clicking **New.** Wizard window is opened. We select **2D frame shape** and **column to beam** topology.
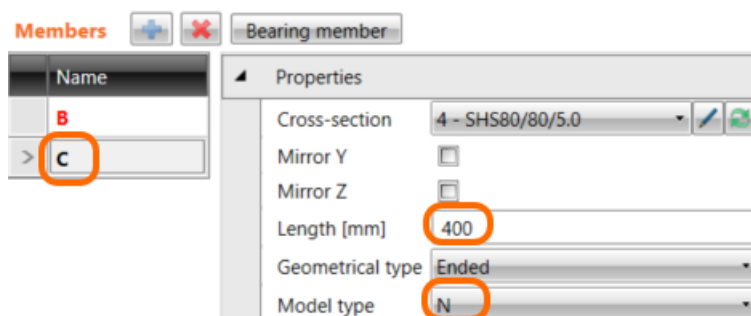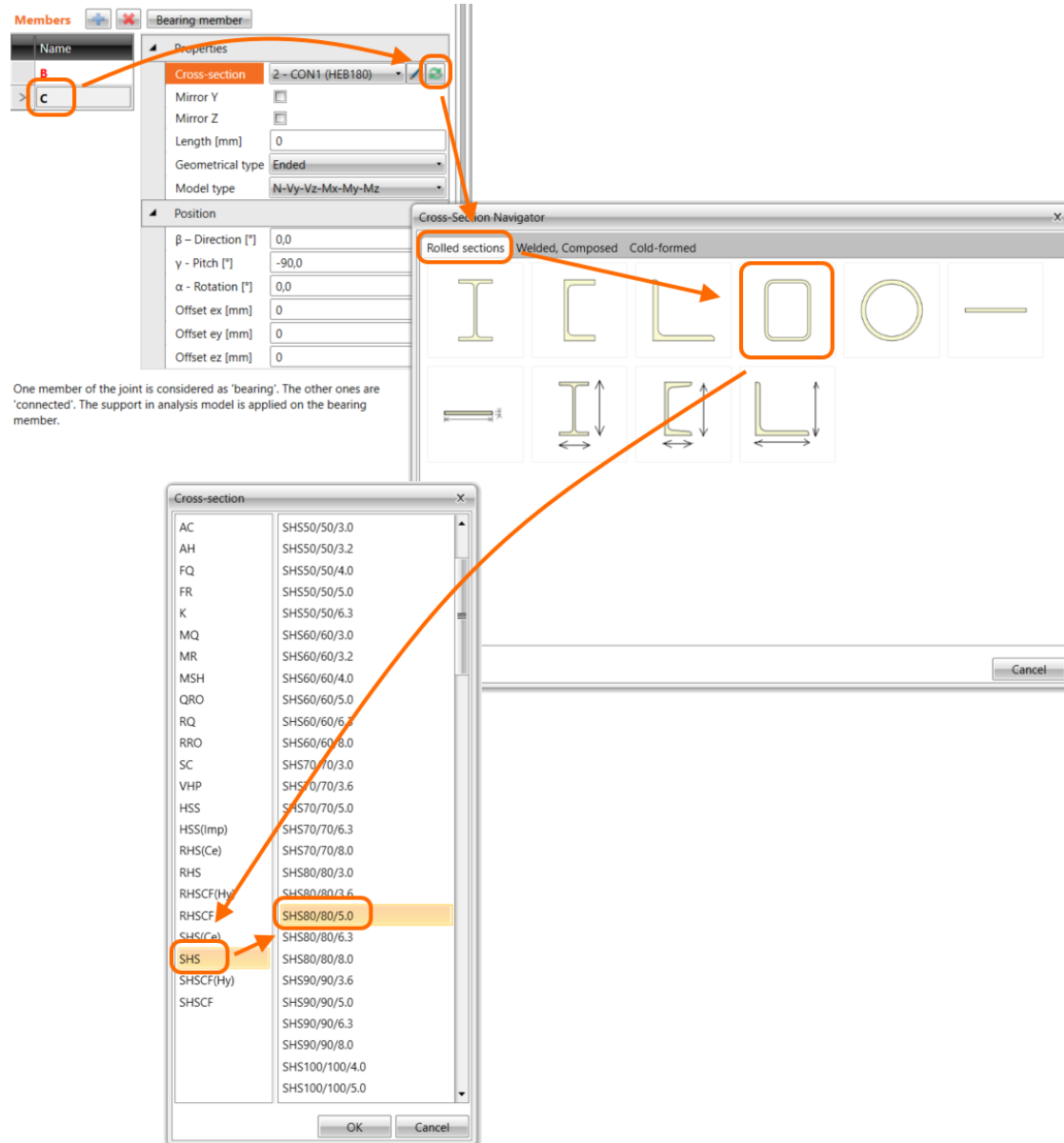
Navigator on the left will guide us through the whole project.
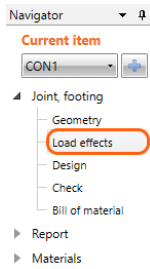
Geometry

We start with defining the **Geometry**. Two beams were automatically added. We change cross-sections of member **C** to **SHS 80/50/5**. We also change its **Length** to **400mm** and set **Model type** to "**N**".

One member of the joint is considered as 'bearing'. The other ones are 'connected'. The support in analysis model is applied on the bearing member.
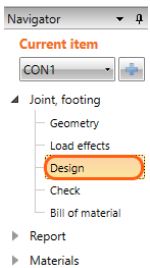
Load effects

Let's continue with **Load effects**. One load effect was automatically added by the wizard. We input value on Normal force into the table. More load cases can be added.

Internal forces [Clean] [Copy X position]

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

Design

Next step is the **Design**. To model connection between beams, we click on **Add button** and select operation **Connecting plate**.

We select default bolt type.

Bolt assembly

| 4.6	M12 4.6
4.8	M16 4.6
5.8	M20 4.6
6.8	M22 4.6
8.8	M24 4.6
10.9	M27 4.6
	M30 4.6
	M36 4.6
	M39 4.6
	M42 4.6
	M48 4.6
	M52 4.6

[OK] [Cancel]
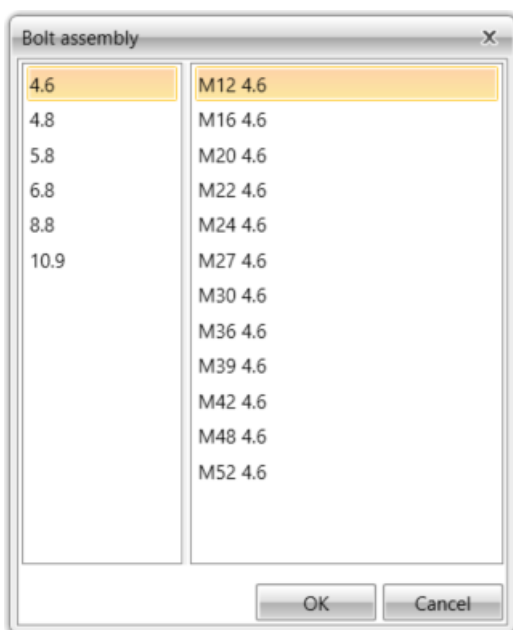
Operation is added and we update its parameters (plate thickness and dimensions, bolt size and position and weld size and type)

**Manufacturing operations**   ✚ ✖ | Delete all | Calculate | Editor ▾

| Name ▼ |

◢ Connecting plate

| Member | C ▾ |
| Connected to | New plate ▾ |

◢ New gusset plate

R1 Type	Member ▾
R1 - related to	B ▾
R2 Type	None ▾
Material	< default > ▾
Thickness [mm]	4
B – width [mm]	100
H – depth [mm]	80
X – position [mm]	0

◢ Connection

X - position [mm]	200
Material	< default > ▾
Thickness [mm]	0
Alignment	Front ▾
Type	Notched member ▾
L - plate length [mm]	80
B – plate width [mm]	0
O – Overlap [mm]	100
E – plate excentricity [mm]	0
Connection type	Bolted ▾

◢ Welds

| Plate | 3 mm | < default > ▾ | ⊿ ⊿ ⊿ ⊿ ⊿ |
| Tongue | 3 mm | < default > ▾ | ⊿ ⊿ ⊿ ⊿ ⊿ |

◢ Bolts

| Type | M16 10.9 ▾ |
Reference line	Member x-axis ▾
Rows [mm]	0
Positions [mm]	30
Grid	Regular ▾
Shear plane in thread	☑
Shear force transfer	Bearing - tension/shear interaction ▾
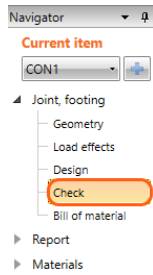
Check of a structural steel joint

Now we move to the **Check** section of the Navigator. We click button **Code setup** from the ribbon and activate **Buckling analysis**.
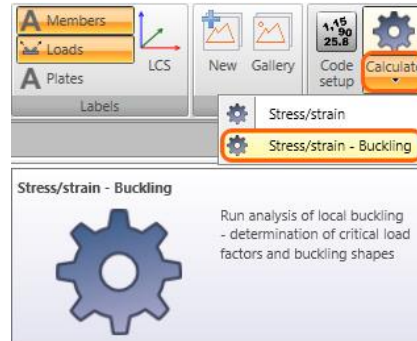

Then we start nonlinear analysis by clicking **Calculate** in the ribbon. Analysis model is automatically generated, Stress/strain calculation is performed and we can start **Bucking analysis**.

We activate **Buckling shape**, **Mesh** and **Deformed** from the ribbon to get a full picture of buckling shape of the joint. Everything is displayed in the 3D window.

To check a shape of calculated **Factor of critical load,** we select tab **Buckling** and choose a particular raw from the table.

Loads	Shape	Factor
LE1	1	3,93
	2	11,12
	3	21,45
	4	29,89
	5	39,81
	6	47,91
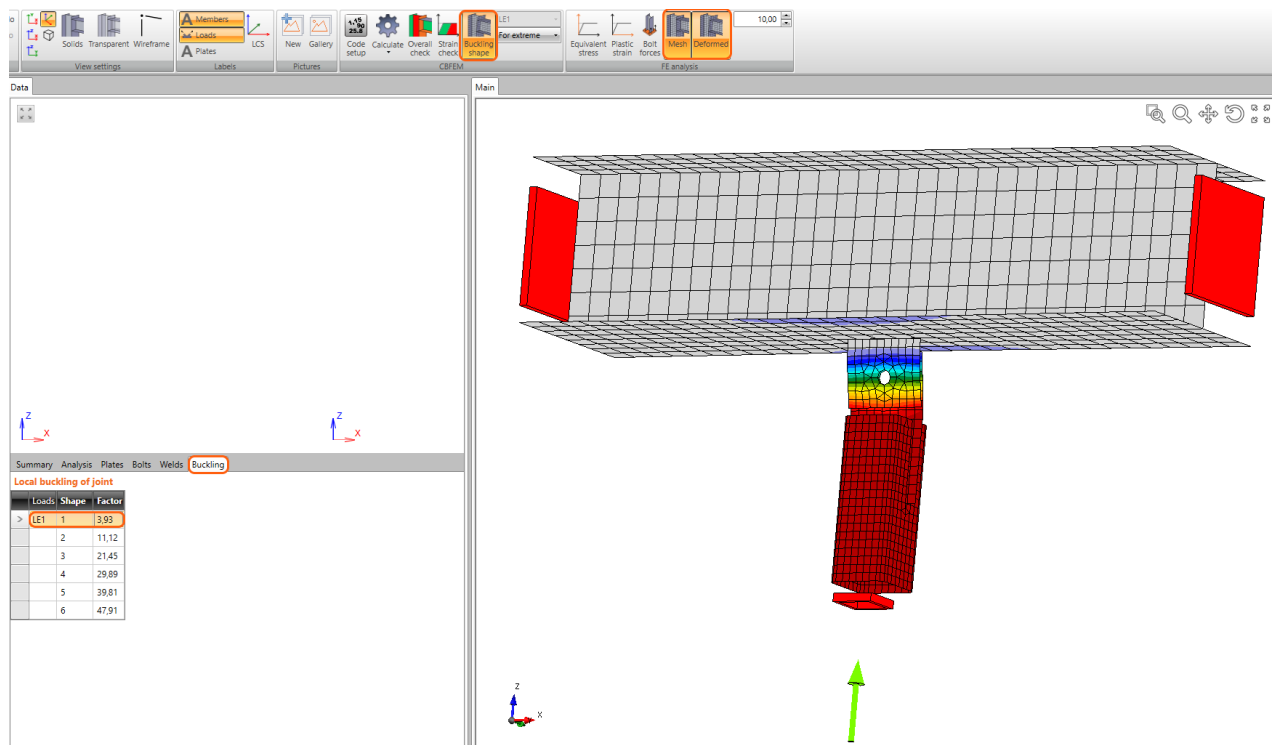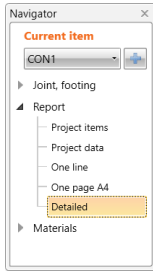
Report

The final step of the project is to generate the **Report.** IDEA StatiCa offers three types of output reports – one line, one page and detailed. We will choose the **Detailed** report that contains materials, geometry, all checks. We can also add bill of material and theoretical background to explain underlying methods and references to selected design code.
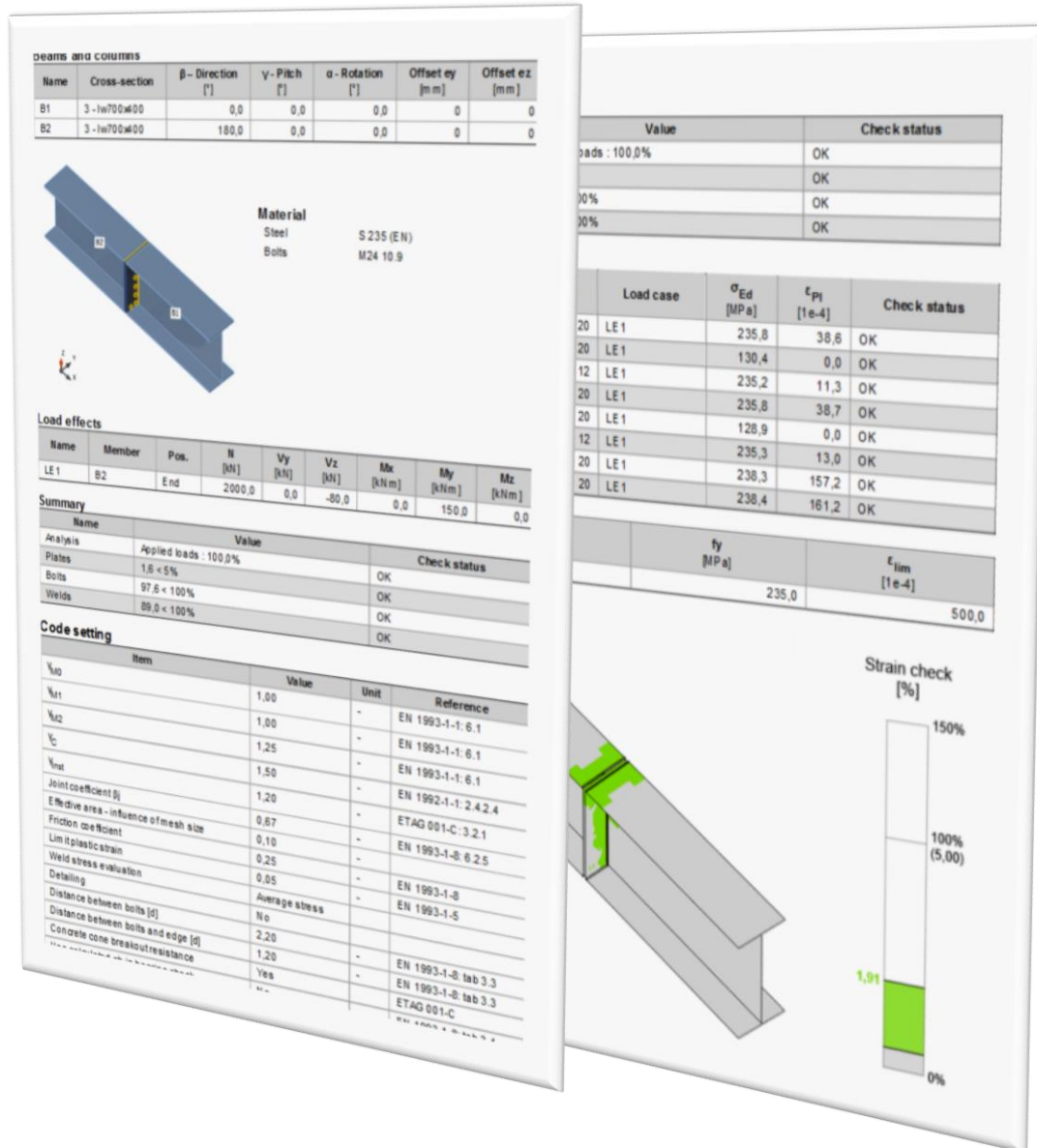
Structural steel joint was modelled, designed and checked

Thank you for spending time on this tutorial. For further information please visit our website.