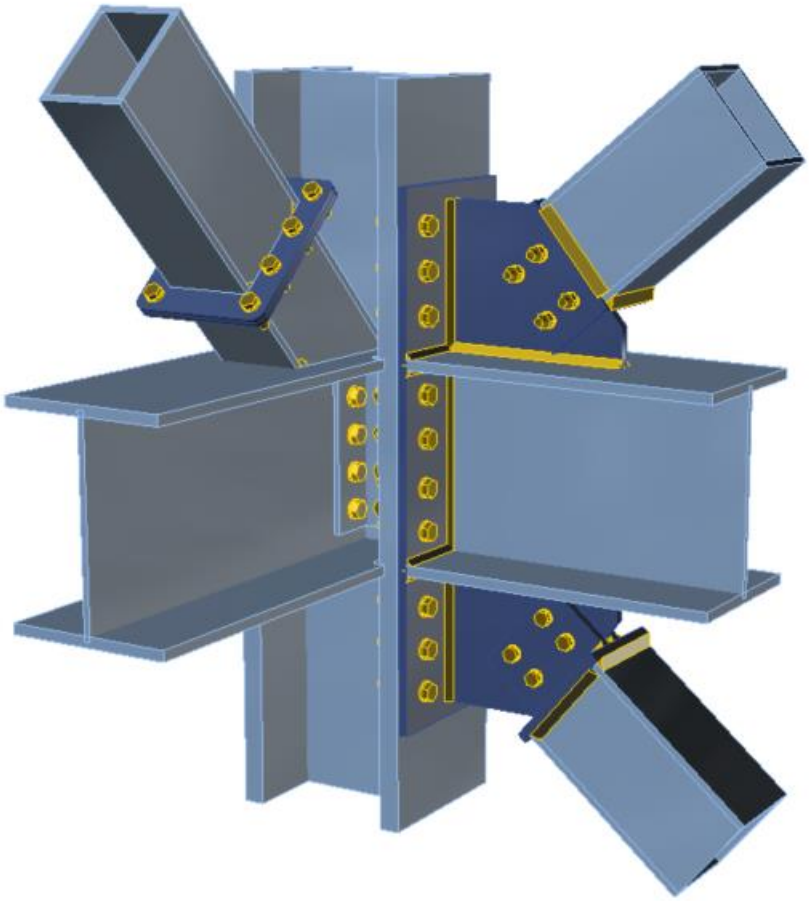
IDEA StatiCa®
Calculate yesterday's estimates

IDEA StatiCa Steel - Tutorial



Robot Structural Analysis link

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

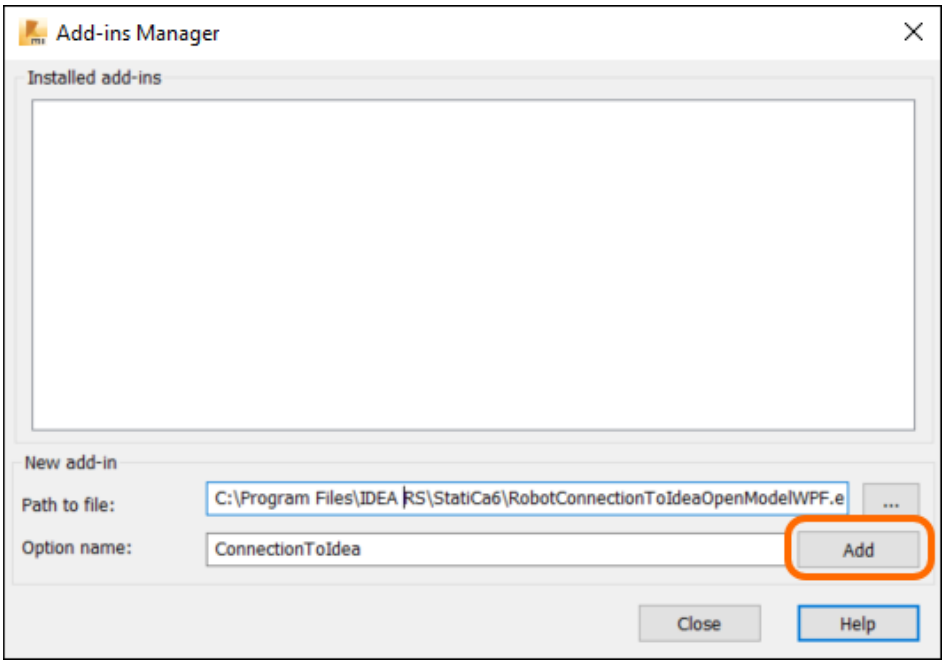
This tutorial will show how to activate and use the link between Robot Structural Analysis and IDEA StatiCa Connection.

Robot Structural Analysis

How to activate the link

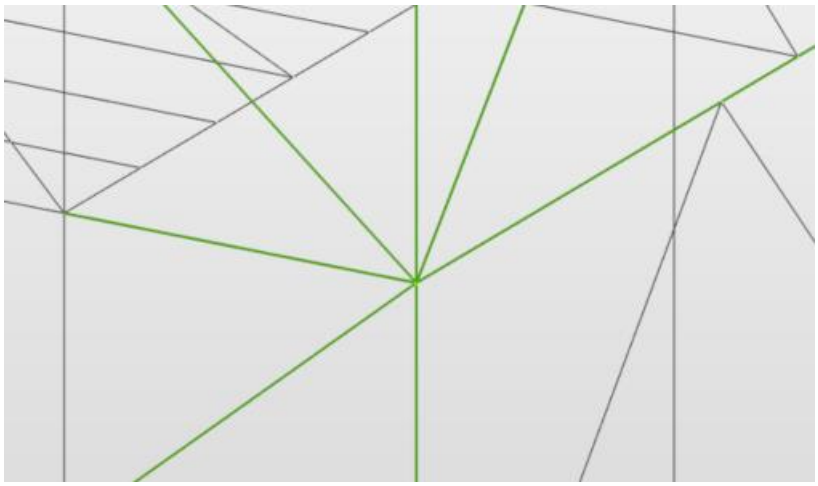
1. Robot Structural Analysis Professional (RSAPRO) **2016**, version 29.0.0.5650 (x64) or later must be installed on the computer.
2. Download and install the latest version of **64-bit IDEA StatiCa Steel .**
   Enhanced edition is required.

After installation of both programs, start RSA and Click Add-ins > **Add-ins Manager** to open the Add-ins Manager dialogue. This option lets you install and implement add-ins (programs) to the appropriate places in the Robot menu. Browse to folder where IDEA StatiCa is installed (c:\Program Files\IDEA RS\StatiCa6) and open file: **RobotConnectionToIdeaOpenModelWPF.exe** Finally click **Add** and **Close** buttons.

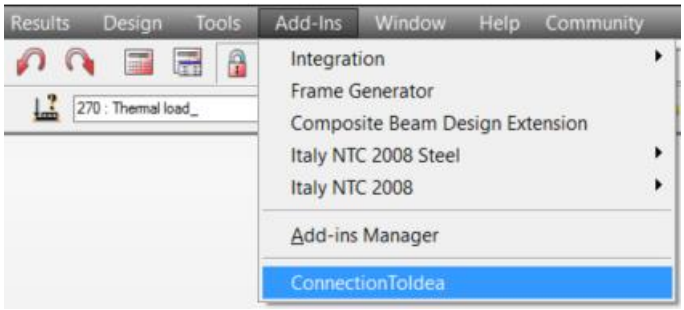


How to use the link

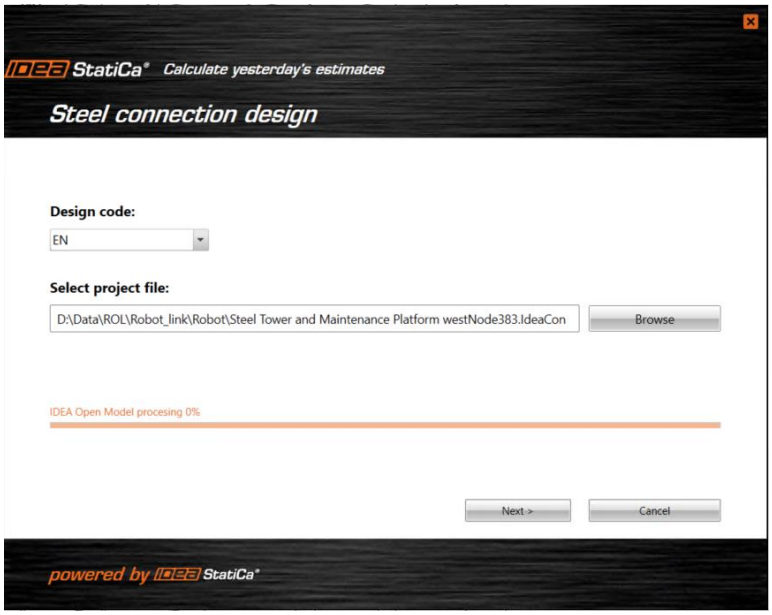
1. Open attached project in Robot Structural Analysis.
2. Select a joint **node** and connected **members**.



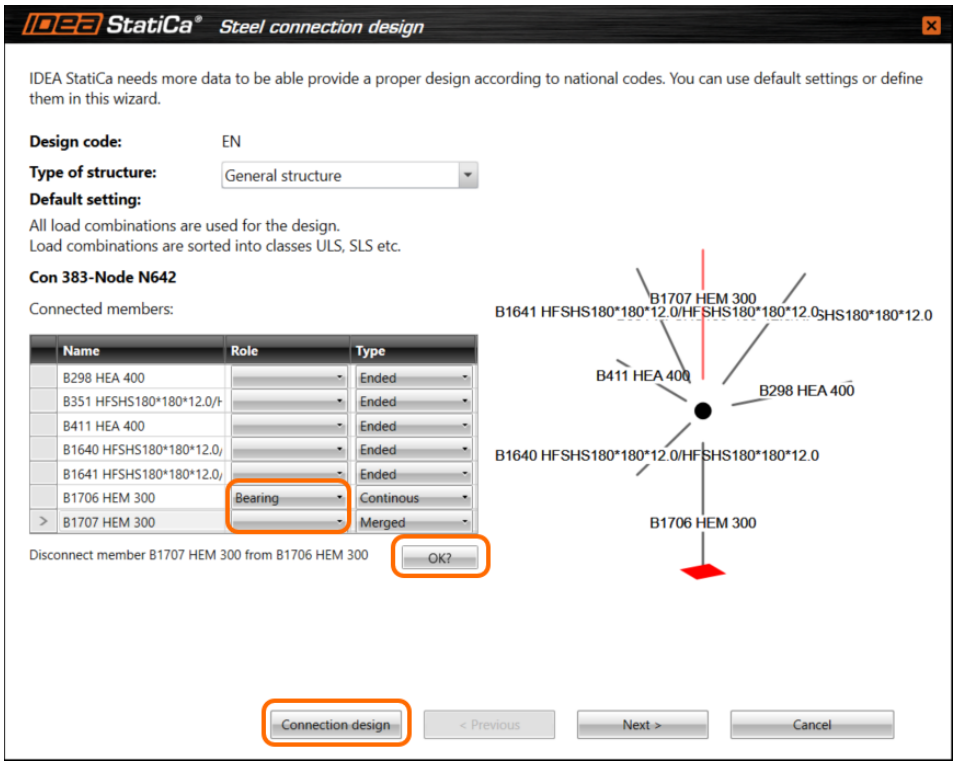
3. Start IDEA Connection add-inn from the menu.



4. IDEA StatiCa joint import wizard is launched. Set file name and path of IDEA StatiCa Connection project to save and click button **Next**

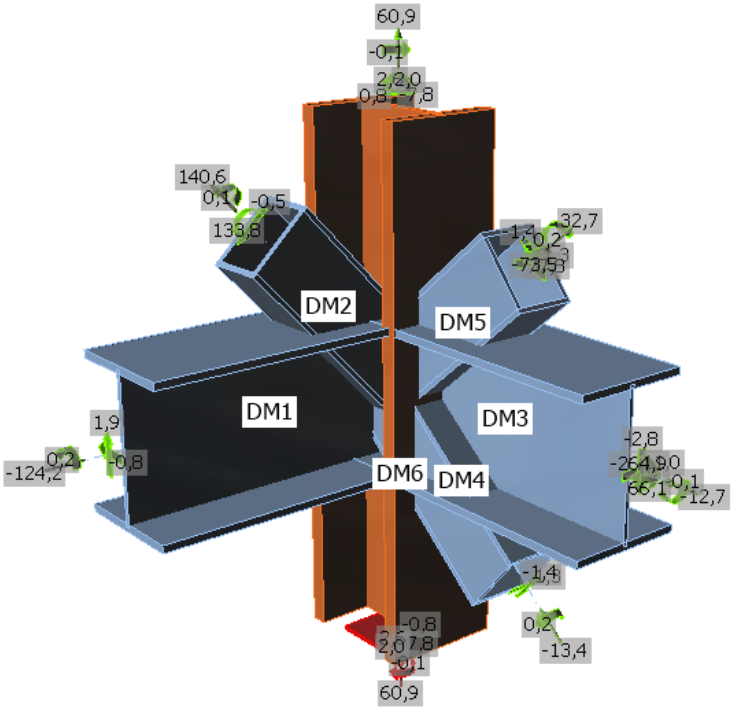


5. Select bottom column member and set it as **Bearing** and merge column members by **OK** button. Finish settings by **Connection design** button.

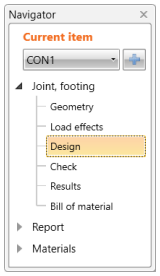


IDEA StatiCa Connection

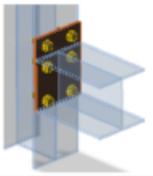
Automatic data transfer is started and IDEA StatiCa Connection with generated project is launched. All members and load effects were added automatically.



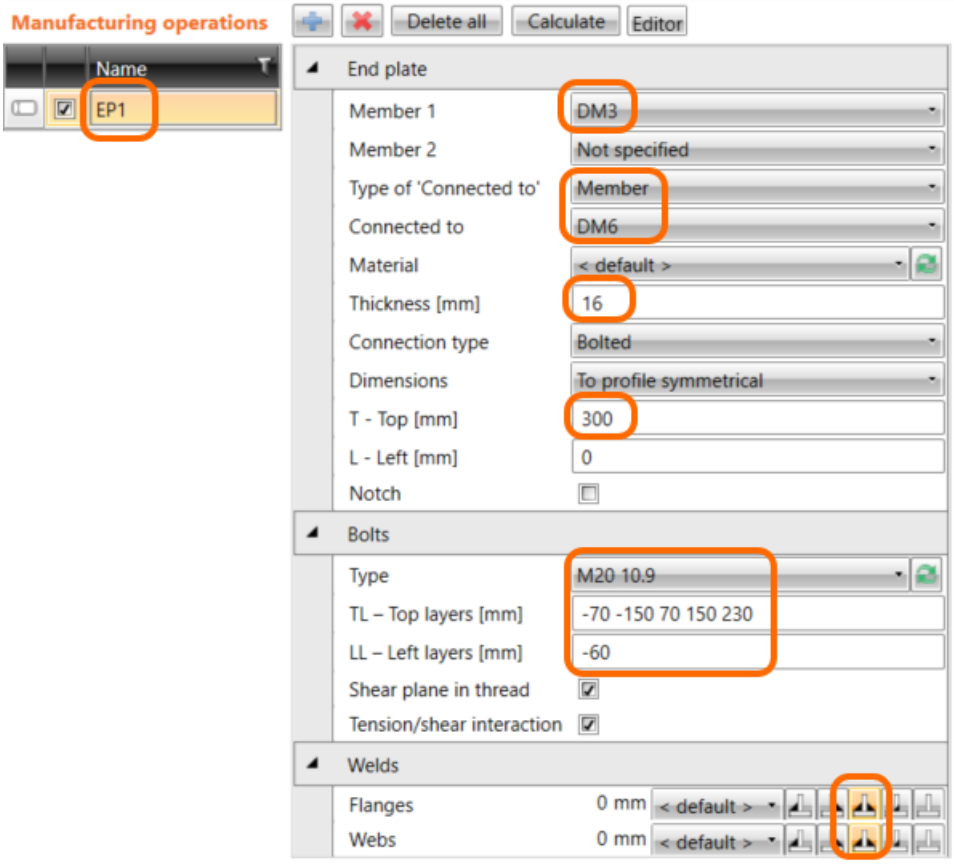
Design



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



End plate



Connecting plate

Manufacturing operations

Delete all | Calculate | Editor

Name
EP1
CPL1

Connecting plate

Member	DM5
Connected to	New plate

New gusset plate

R1 Type	Member
R1 - related to	DM3
R2 Type	Plate
R2 – related also to	EP1
Material	< default >
Thickness [mm]	0
B – width [mm]	350
H – depth [mm]	270
X – position [mm]	0
Shape	Rectangular

Connection

X - position [mm]	600
Material	< default >
Thickness [mm]	0
Alignment	Front
Type	Cap plate 2x
L - plate length [mm]	150
B – plate width [mm]	0
E – plate excentricity [mm]	0
Thickness of cap [mm]	0
Cap plate offset (LL RR TT BB) [mm]	10
Cap plate shape	Rectangular
Connection type	Bolted

Welds

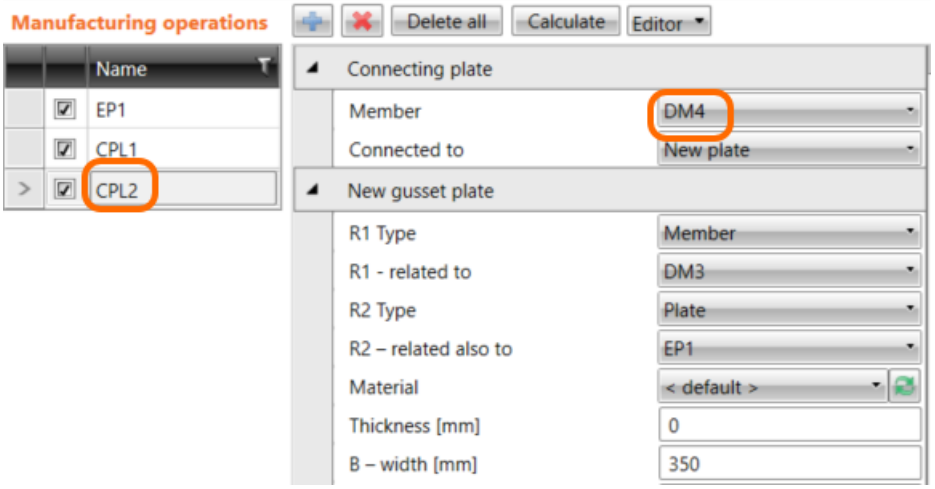
Plate	0 mm
Cap plate	0 mm
Tongue	0 mm

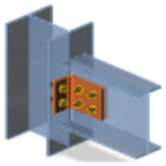
Bolts

Type	M16 10.9
Reference line	Member x-axis
Rows [mm]	50 ; -50
Positions [mm]	30 60
Grid	Regular
Shear plane in thread	☑
Tension/shear interaction	☑

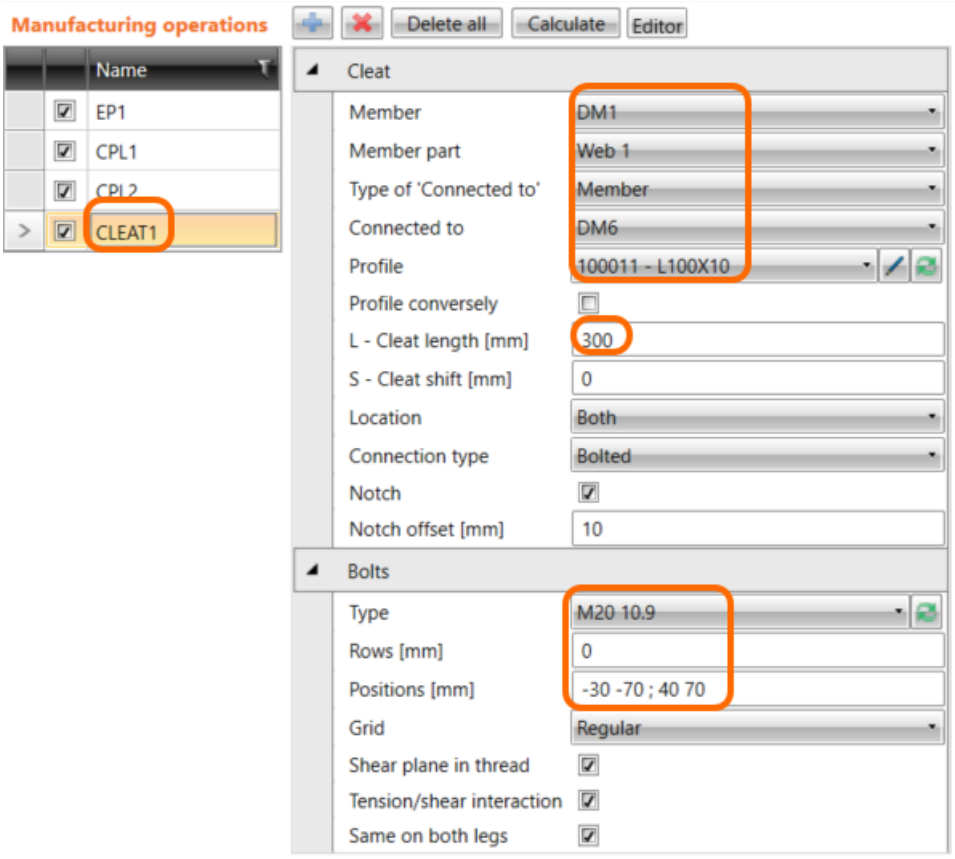
Copy CPL1 operation by 

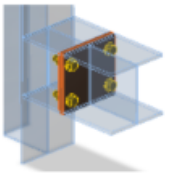
button from the ribbon and set that on member DM4.



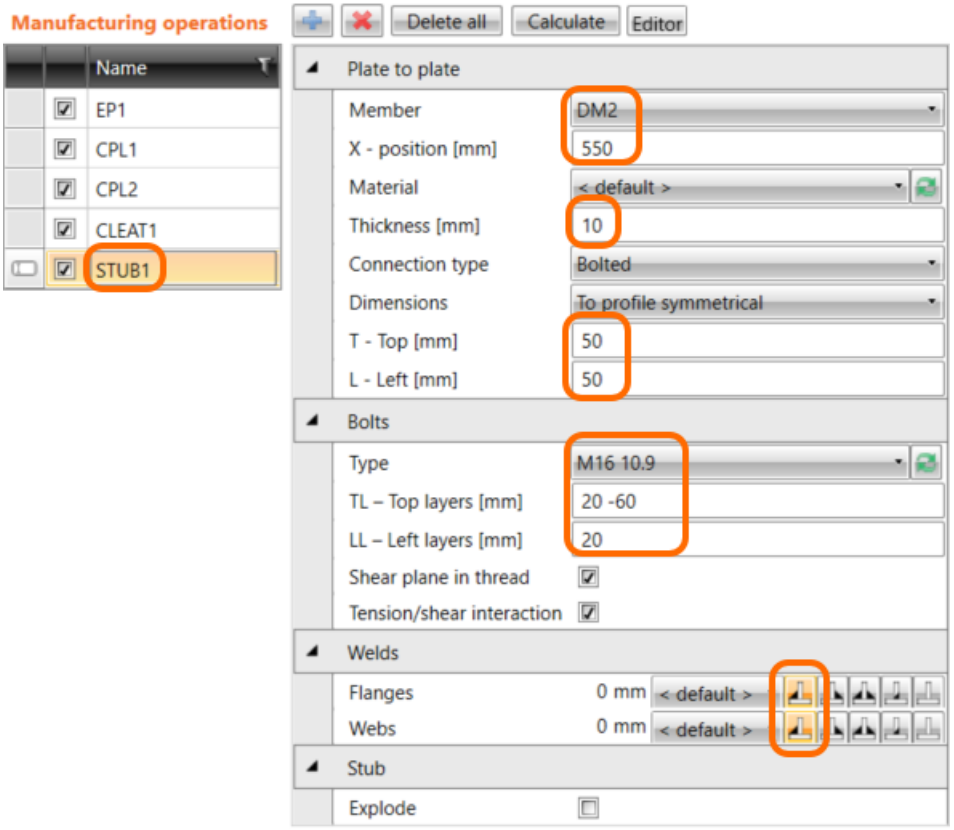


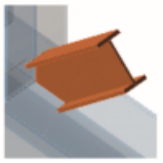
Cleat



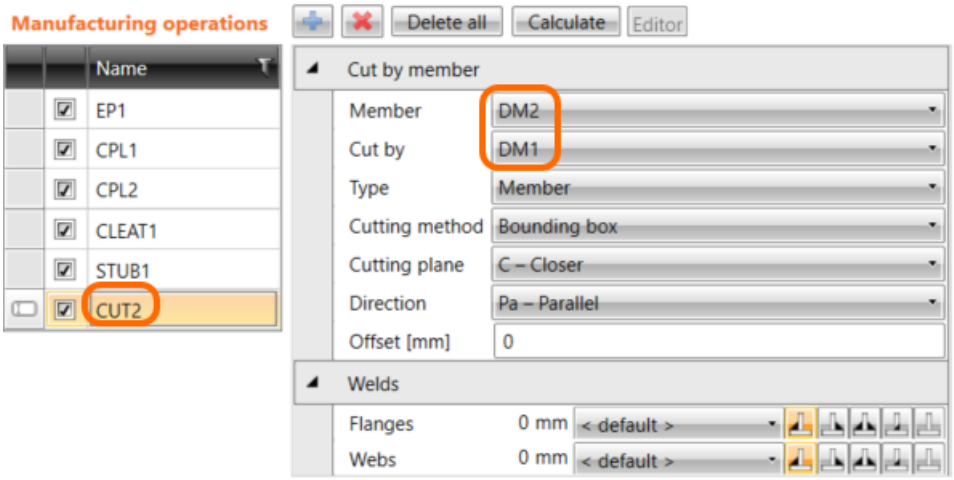


Stub

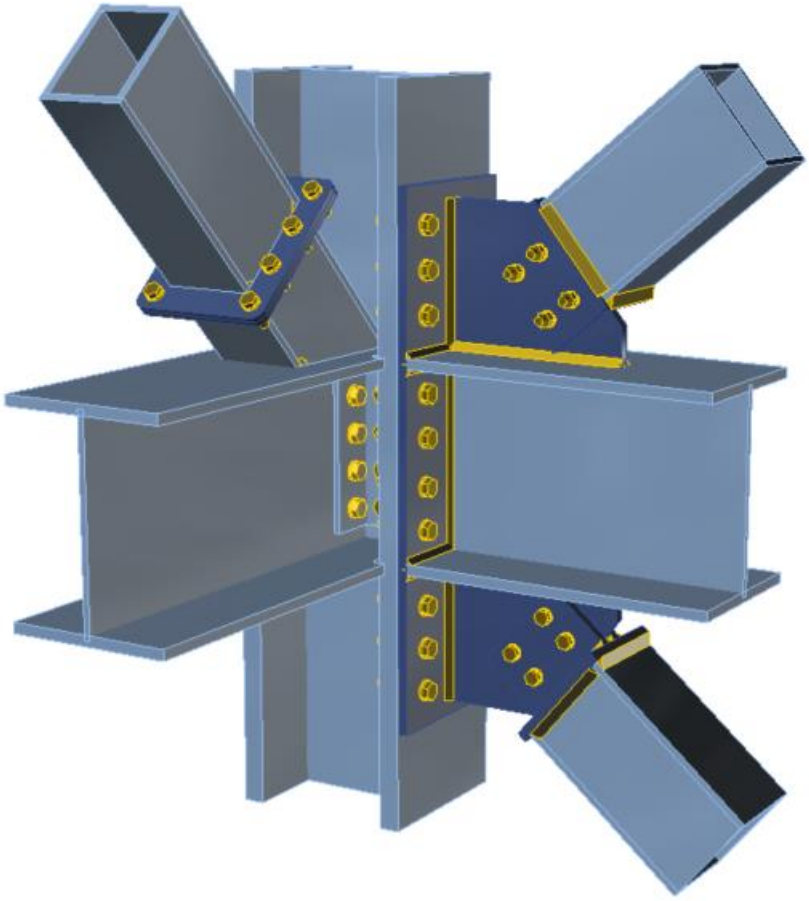

Manufacturing operations
Delete all
Calculate
Editor
Name
EP1
CPL1
CPL2
CLEAT1
STUB1
Plate to plate
Member
DM2
X - position [mm]
550
Material
< default >
Thickness [mm]
10
Connection type
Bolted
Dimensions
To profile symmetrical
T - Top [mm]
50
L - Left [mm]
50
Bolts
Type
M16 10.9
TL – Top layers [mm]
20 -60
LL – Left layers [mm]
20
Shear plane in thread
Tension/shear interaction
Welds
Flanges
0 mm
< default >
Webs
0 mm
< default >
Stub
Explode

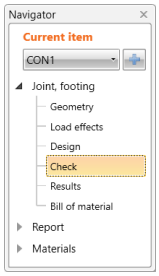


Cut


Manufacturing operations
Delete all
Calculate
Editor
Name
EP1
CPL1
CPL2
CLEAT1
STUB1
CUT2
Cut by member
Member
DM2
Cut by
DM1
Type
Member
Cutting method
Bounding box
Cutting plane
C – Closer
Direction
Pa – Parallel
Offset [mm]
0
Welds
Flanges
0 mm
< default >
Webs
0 mm
< default >

Let's check defined operations of the joint.



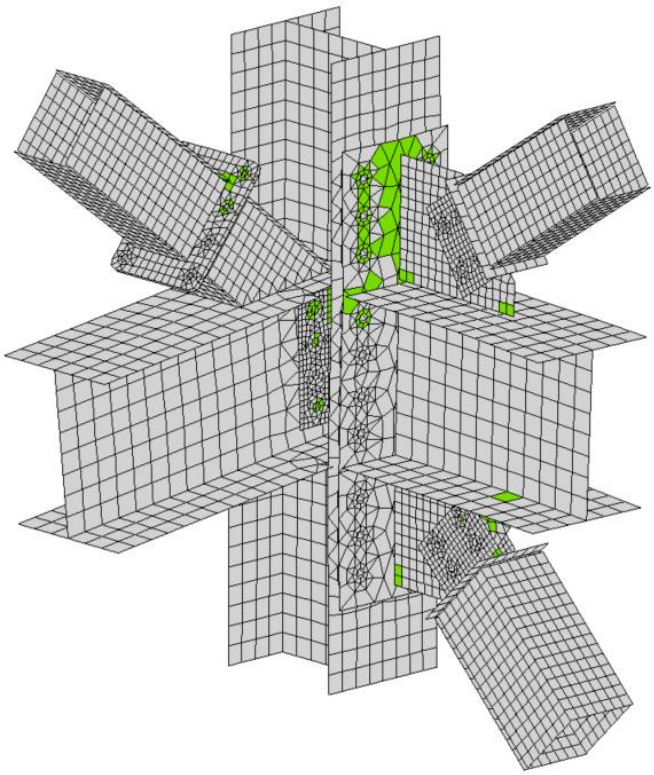


Check of a structural steel joint

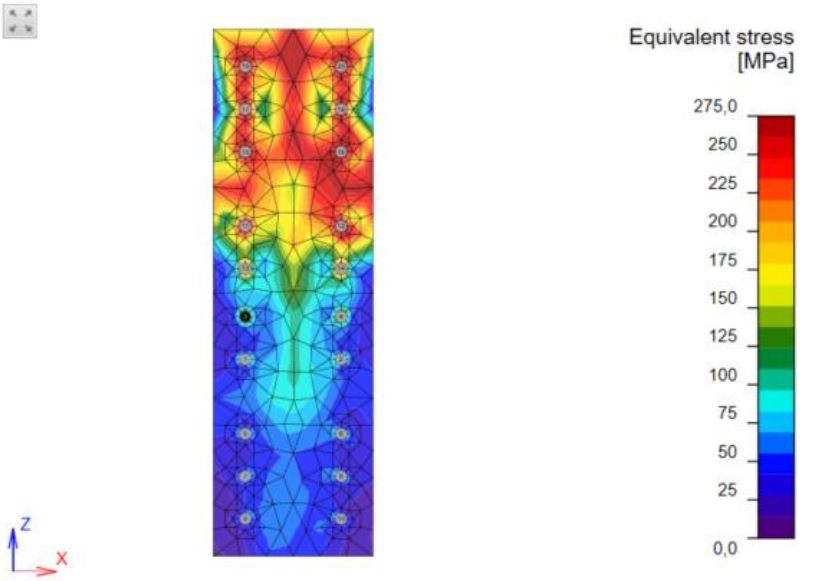
Nonlinear analysis is started by icon 

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.

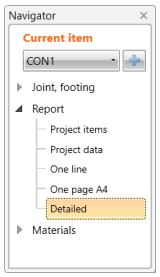


All values can be checked in detailed in the tables and 2D window. For example to display check of bolts we select tab **Bolts/Anchors** tab. We can also activate icon **Equivalent stress** from the ribbon.



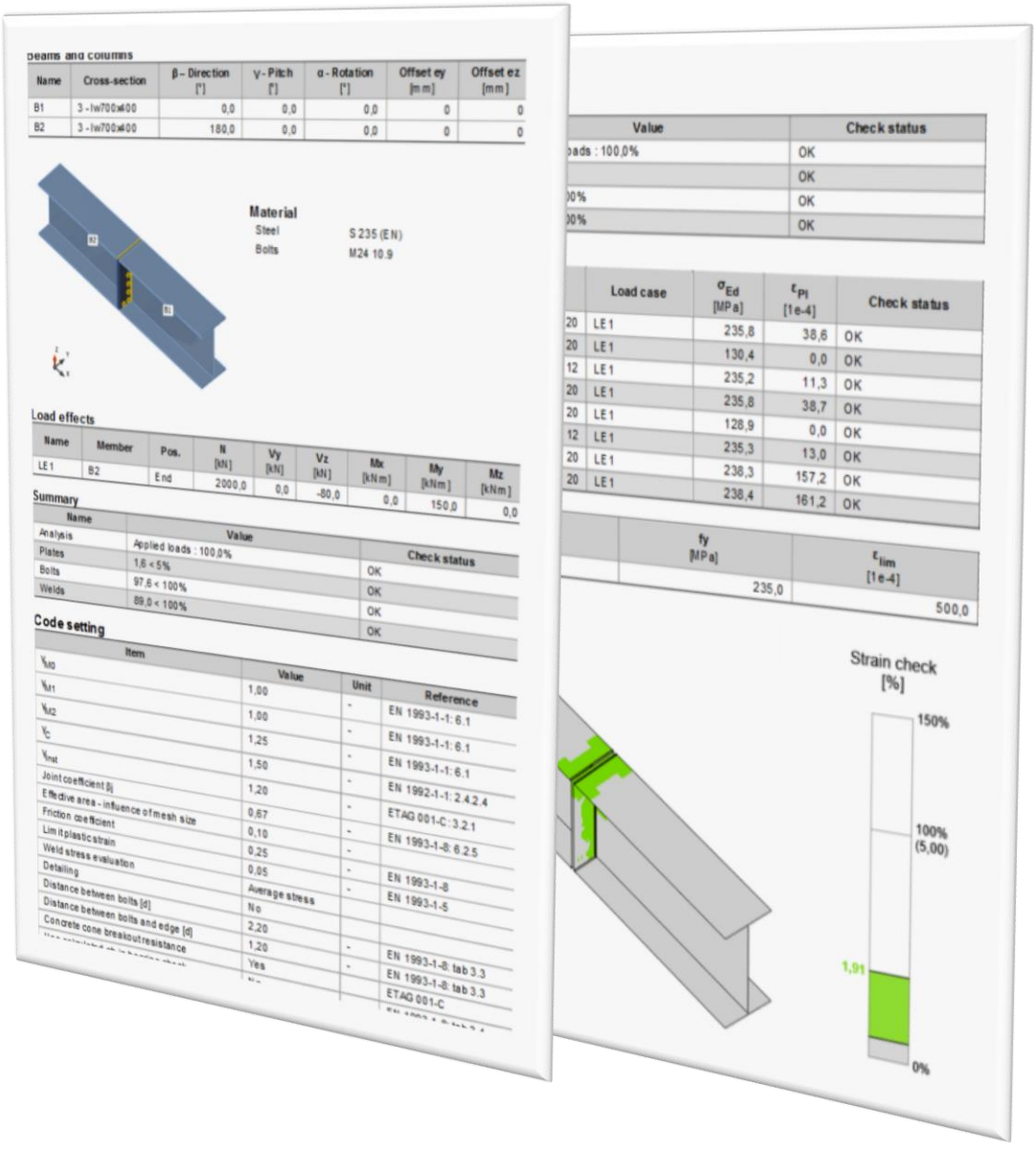
Summary | Analysis | Plates | Bolts/Anchors | Welds

	Item	Material	Loads	Ft [kN]	V [kN]	Utt [%]	Uts [%]	Utts [%]	Status
	B1	M20 10.9 - 1	ULS100_	2,5	7,048	1,4	7,2	8,2	✓
	B2	M20 10.9 - 1	ULS100_	3,0	2,789	1,7	2,8	4,1	✓
>	B3	M20 10.9 - 1	ULS100_	15,2	6,653	8,6	6,8	12,9	✓
	B4	M20 10.9 - 1	ULS100_	17,3	3,55	9,8	3,6	10,6	✓



Report

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



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

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