

Contents

Beam to column connection with end plate	2
Steel truss joint	18
Web to chord connection	27

Beam to column connection with end plate

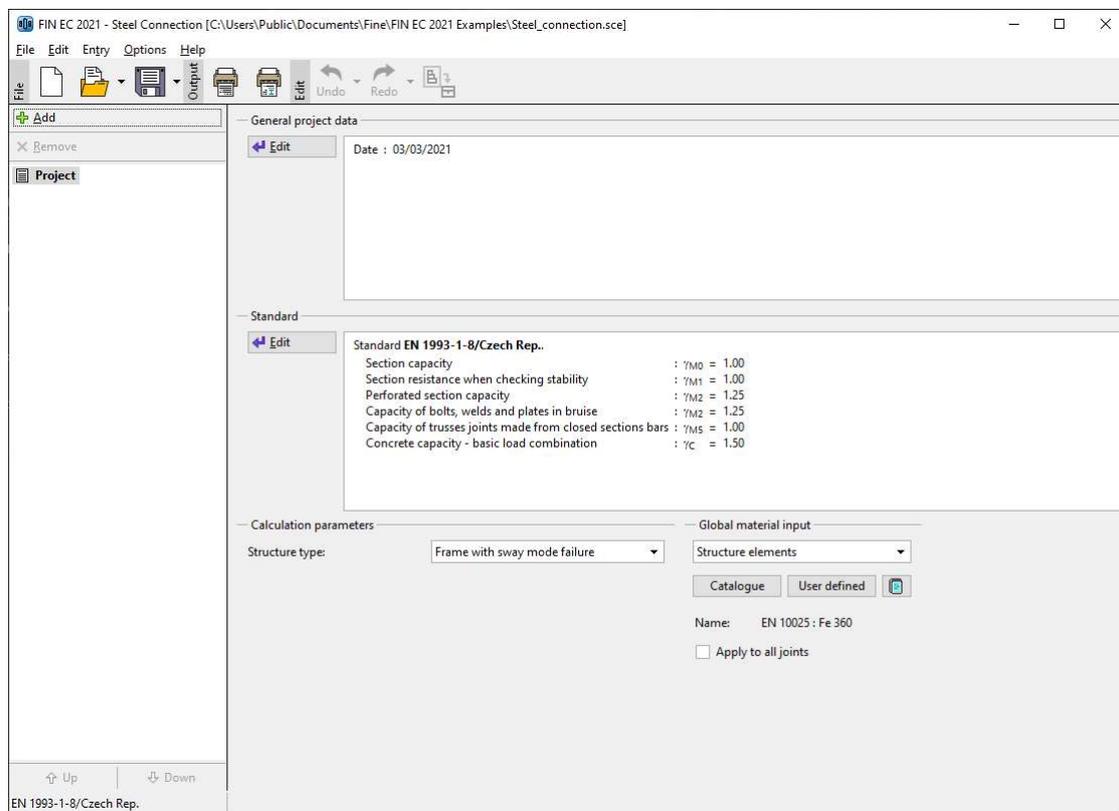
Introduction

This guide shows the input of double-sided connection of beams to a column in the program "Steel Connection". The connection on the left flange of the column is designed using end plate, a beam on the right flange of the column is connected with the help of welding. The joint is a part of a structure without sway mode failure.

Beam forces: $M_{y,Sd} = 30 \text{ kNm}$, $V_{z,Sd} = 100 \text{ kN}$
 Axial force in column: $N_{x,1} = 700 \text{ kN}$, $N_{x,2} = 500 \text{ kN}$
 Column: HE 140B - EN 10025 : Fe360
 Beam: IPE 200 - EN 10025 : Fe360
 Welds: $a_{w,f} = 6 \text{ mm}$; $a_{w,w} = 4 \text{ mm}$
 End plate: $b_p = 120 \text{ mm}$, $h_p = 280 \text{ mm}$, $t_p = 12 \text{ mm}$, $a_1 = -70 \text{ mm}$ - EN 10025: Fe360
 Bolts position: $w_1 = 30 \text{ mm}$, $e = [35, 80, 120] \text{ mm}$
 Bolts: M16 8.8

Starting new project

The main screen of the program contains general project information, properties of design standard and also an option to specify global material for all parts of connections.



Basic screen of the program

First, we select a national annex. It is necessary to click on the button "Edit" in the part "Standard" in the middle of the basic screen. A window "Standard selection" appears. Upper part contains a list box with available national annexes. National annex "User defined" contains an option to specify partial safety factor by the user in the bottom part of the window.

Standard selection

National annex: Czech Rep.

Factors for steel structures:

Section capacity	γ_{M0}	=	1.00	[-]	EN 1993-1-1 - Chap.6.1
Section resistance when checking stability	γ_{M1}	=	1.00	[-]	EN 1993-1-1 - Chap.6.1
Perforated section capacity	γ_{M2}	=	1.25	[-]	EN 1993-1-1 - Chap.6.1
Capacity of bolts, welds and plates in bruise	γ_{M2}	=	1.25	[-]	EN 1993-1-8 - Chap.2.2
Capacity of trusses joints made from closed sections bars	γ_{M5}	=	1.00	[-]	EN 1993-1-8 - Chap.2.2

Factors for concrete structures:

Concrete capacity - basic load combination	γ_C	=	1.50	[-]	EN 1992-1-1 - Chap.2.4.2.4
--------------------------------------------	------------	---	------	-----	----------------------------

Default

OK Cancel

Choice of a national annex

As all parts of the joint have identical material, we can use a tool "Global material input" in the right bottom corner of the window and assign this material to all parts of the connection. We select "Structure elements" in the list of entities. "Structural elements" are all steel parts of the detail (column, beams, end plates etc.) except connectors. This material will be assigned to all structural parts after ticking the check box "Apply to all joints".

Global material input

Structure elements

Catalogue User defined

Name: EN 10025 : Fe 360

Apply to all joints

Input of a global material

To avoid unwanted changes in existing connections, this step has to be confirmed by a confirmation window.

Question

Material for "Structure elements" will be assigned for all joints.
Do you want to continue?

OK Cancel

Assignment of the global material to all connections

The part "Calculation parameters" contains an option to specify a structure type. This setting is important for a calculation of stiffness and classification of the connection (pinned, semi-rigid, rigid).

Calculation parameters

Structure type: Frame with sway mode failure

Frame with sway mode failure
Frame with no sway mode failure

Choice of structure type

After the setting of all these parameters, it is possible to start with the input of joints. The wizard for a new joint can be launched by the button "Add" in the heading of the tree menu.



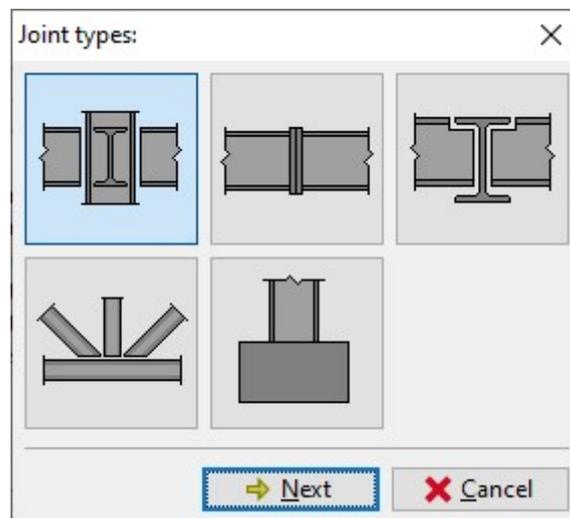
Insertion of a new joint

We specify a name ("Joint 1") in the window "Add joint", that appears after the clicking on that button.



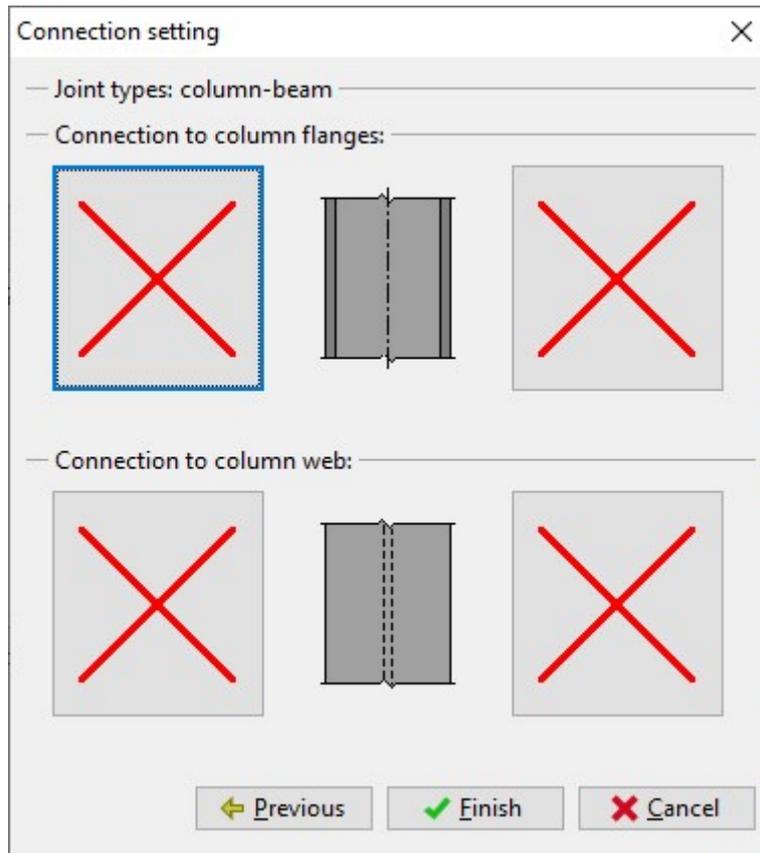
Input of a name

After the confirmation by the button "OK", the window with main joint types appears. We select beams connected to a column (left option in the upper row).



Selection of joint type

Following window contains an option to specify particular connections in the joint. New connection can be assigned to corresponding direction with the help of buttons with red "X". We have to click on the left upper button to assign a connection to the left flange.

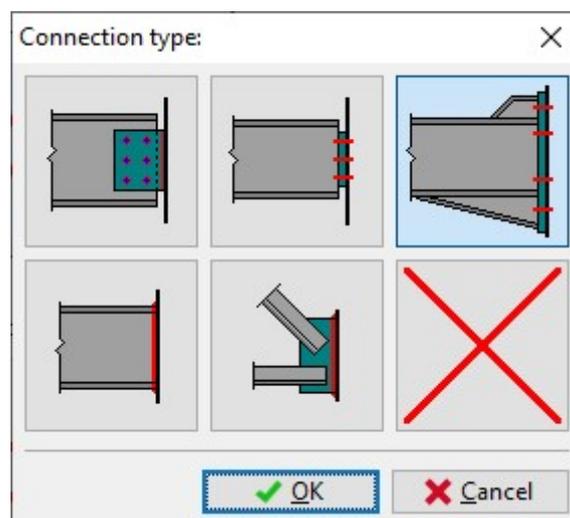


Selection of connection to the left flange

The new window "**Connection type**" appears after clicking on the button. Following connection types are available for this type of joint (first column is the position in the window "**Connection type**"):

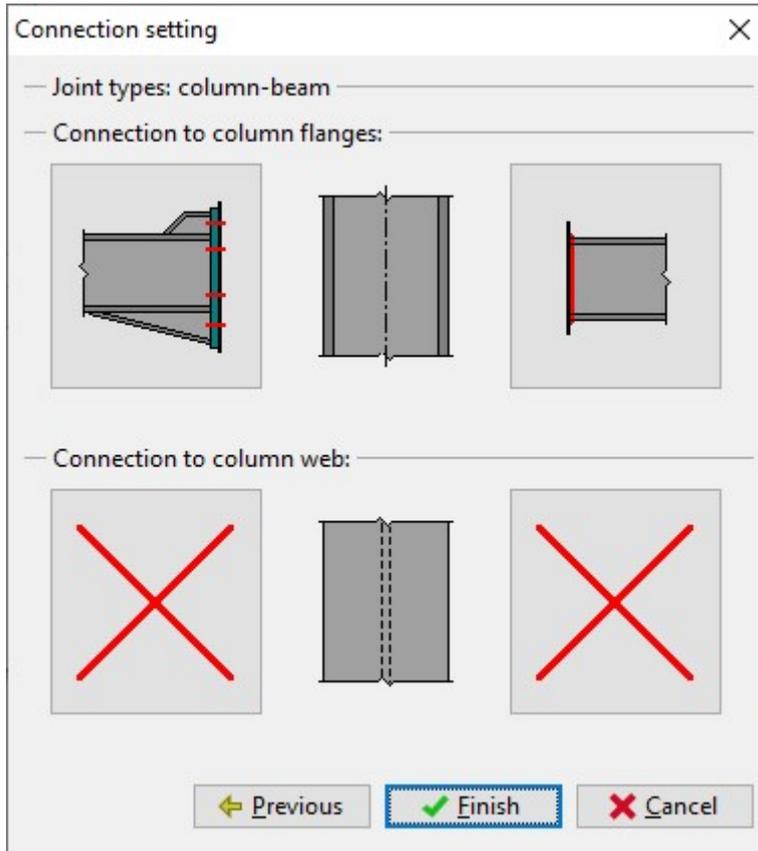
- Upper row, left** •Hinged end plate
- Upper row, middle** •Fin plate
- Upper row, right** •Stiff end plate
- Bottom row, left** •Welded connection
- Bottom row, middle** •Truss connection

We select stiff end plate (the button in the right upper corner) and confirm the input by the button "**OK**"



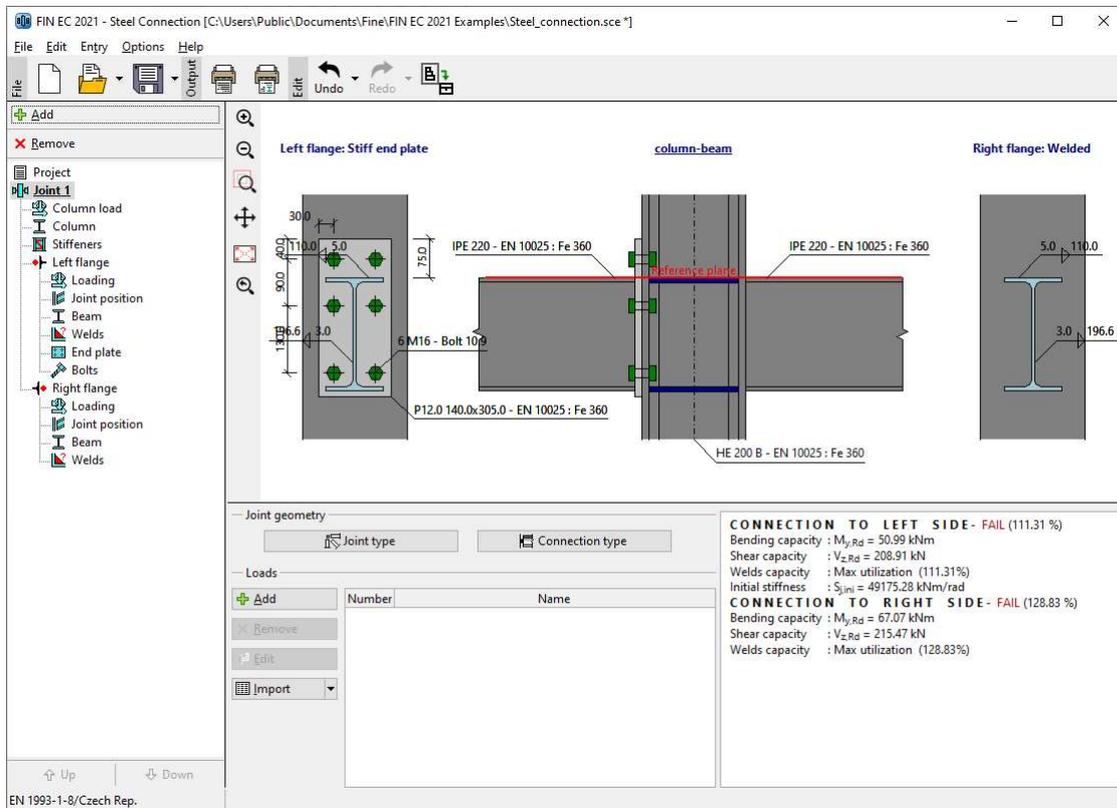
Choice of connection type

We specify also the connection to the right flange. For this connection, we select the type "**Welded connection**". Following figure shows the joint with selected connections:



Joint with specified connections

The initial wizard has to be confirmed by the button "Finish". The fundamental geometry of the joint will be created in the workspace, the structure of the tree menu corresponds to this geometry.

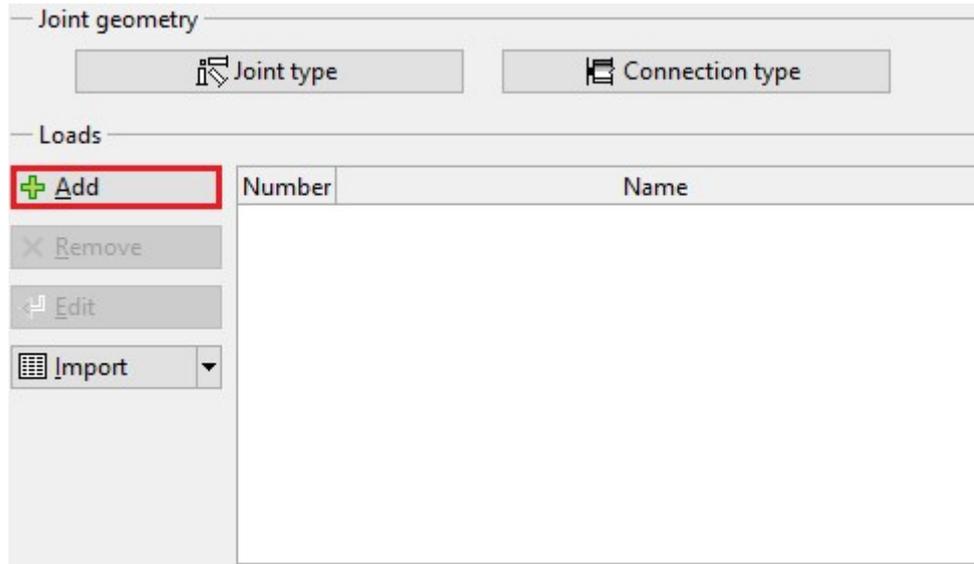


Main screen with defined geometry of the joint

Input of particular components

The following work is done with the help of the tree menu on the left side of the program. The structure of this menu is generated according to the specified joint geometry. We go through all parts from the top to the bottom and modify inputs. The connection to the left flange will be described in detail. The connection to the right flange would be solved in the same way.

On the main screen of the joint, it is possible to change the specified geometry of the joint (buttons "**Joint type**" and "**Connection type**") and specify list of loads. The load represents a set of internal forces that have to be defined for all members in the joint (column, beams). These internal forces should be resulting values of certain load combination. Therefore, they are considered as design values. Number of loads for the joint is not limited. We insert a new load with the help of the button "**Add**" in the toolbar on the left side of the loads table. The toolbar also contains buttons for editing and deletion of loads and also a tool for import of loads including internal forces from *.txt or *.csv file.



Button for input of new load

The new is specified by a name. The input has to be confirmed by the button "**Add**".



Window "New load"

As we want to add only one load, we close the window by the button "**Cancel**" after the input of first load.

Column load

If at least one load is entered, it is possible to switch to the part "**Column load**" in the tree menu and specify internal forces for the column. The axial force above the joint " $N_{x,2}$ " should be $500kN$. As the increment due to connected beams is $2 \times 100kN$, the axial force under the joint " $N_{x,1}$ " is $700kN$.

Load: Load 1

— Load values —

$M_{y,2}$ =	<input type="text" value="0.00"/>	[kNm]
$N_{x,2}$ =	<input style="border: 2px solid red;" type="text" value="500.00"/>	[kN]
$V_{z,2}$ =	<input type="text" value="0.00"/>	[kN]
$M_{y,1}$ =	<input type="text" value="0.00"/>	[kNm]
$N_{x,1}$ =	<input style="border: 2px solid red;" type="text" value="700.00"/>	[kN]
$V_{z,1}$ =	<input type="text" value="0.00"/>	[kN]

Input of forces in the mode "Column load"

The next step is an input of column geometry in the part "**Column**".

Column

This part contains a geometry of the column (cross-section, length, material). Buttons for an input of material are disabled, as the global material specified at the beginning is used. The window for the input of column geometry can be launched by the button "**Edit section**" or by clicking on the cross-section in the bottom frame.

— Section —

Edit section

HE 200 B

— Material —

Catalogue
User defined

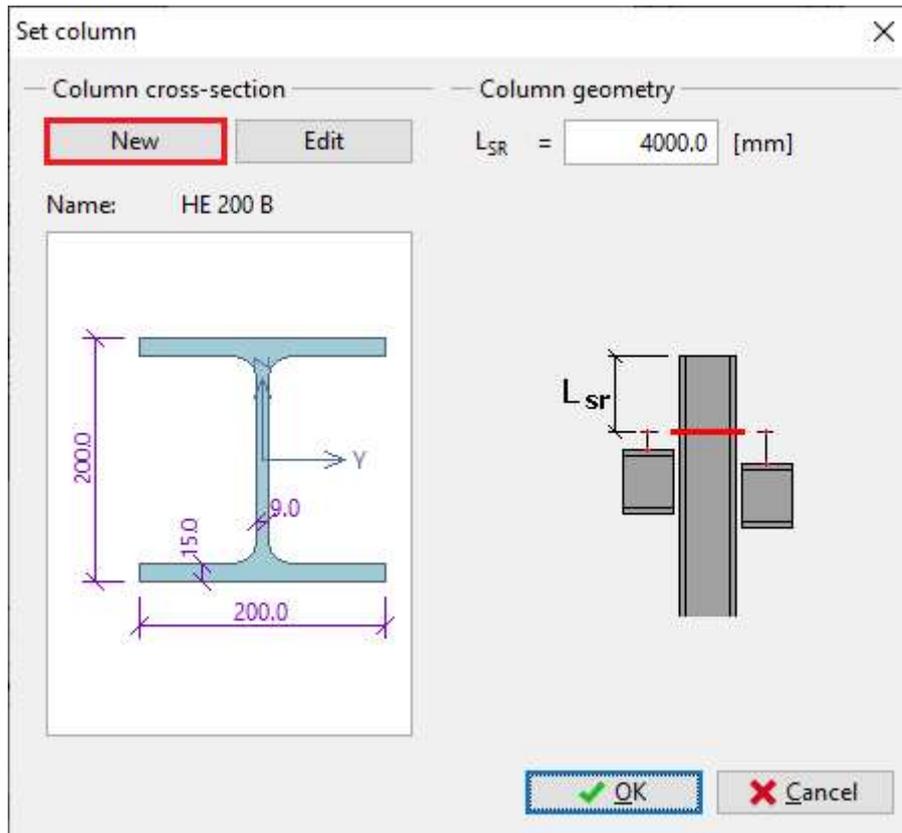
Name: EN 10025 : Fe 360

— Material characteristics —

f_y =		235.0 [MPa]
f_u =		360.0 [MPa]

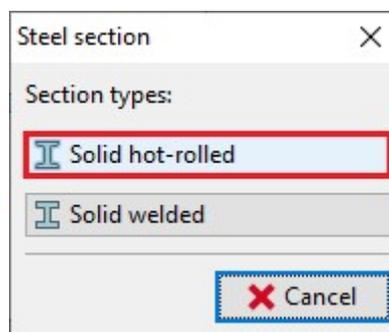
Button for editing the column cross-section

The window "**Set column**" is a general window for input of cross-section. The identical window is also used for input of beams etc. The distance between fundamental level of the joint and column end can be also specified here. This value is important for joints, that are affected by the proximity of column end. We change the cross-section with the help of the button "**New**".



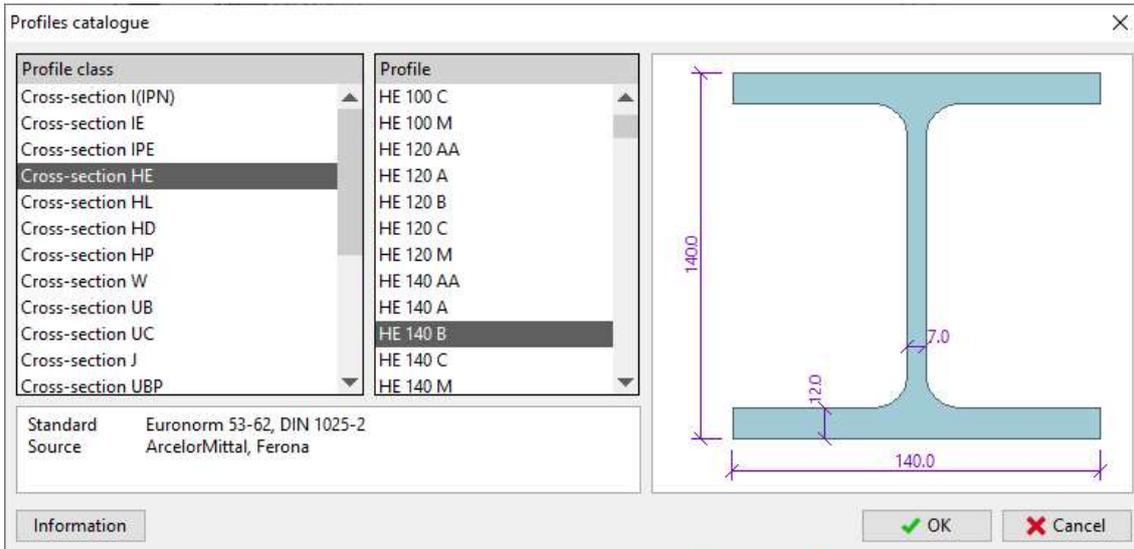
The button for input of cross-section

The window "**Steel section**" that appears after the clicking on the button contains an option to select type of cross-section. We select database of rolled cross-sections (the option "**Solid hot-rolled**") and open the window "**Profiles catalogue**" by pressing the button "**OK**".



Choice of cross-section type

We select the section type "**Cross-section HE**" in the first column of the database and the item "**HE 140B**" in the second column. The choice of cross-section has to be confirmed by the button "**OK**".



The database of predefined cross-sections

The part "Stiffeners" of the tree menu follows.

Stiffeners

The reinforcement of the column can be added in this part. Several types of reinforcement are organized into tabs. We specify web reinforcement on the first tab "Web reinforcement". It is necessary to change the values " a_w " and " t_s " both for upper and bottom stiffeners.



Web stiffeners

This is the last part of column properties. Now, we switch to part "Left flange".

Load

Similarly to column, this part contains the input of internal forces for beam. The bending moment $30kN$ has to be entered into the input line " M_y " and shear force $100kN$ into the input line " V_z ". Also a value for axial force " N_x " can be specified in this part.

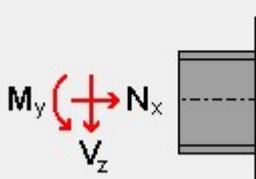
Load: Load 1

— Load values —

N_x = [kN]

V_z = [kN]

M_y = [kNm]



Internal forces for left flange

The part "Joint position" follows.

Joint position

The eccentricity of connection or pitch of beam can be specified in this part. We keep default settings and switch to part "Beam".

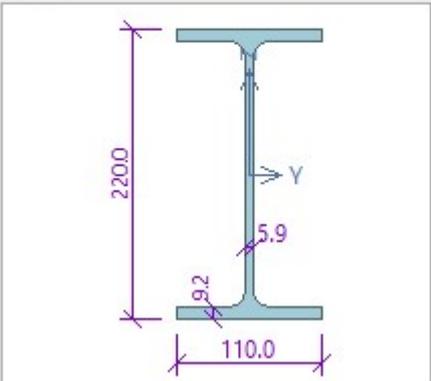
Beam

We specify the beam geometry (cross-section, haunch) with the help of the button "Edit section".

Section

Edit section

IPE 220



Material

Catalogue User defined

Name: EN 10025 : Fe 360

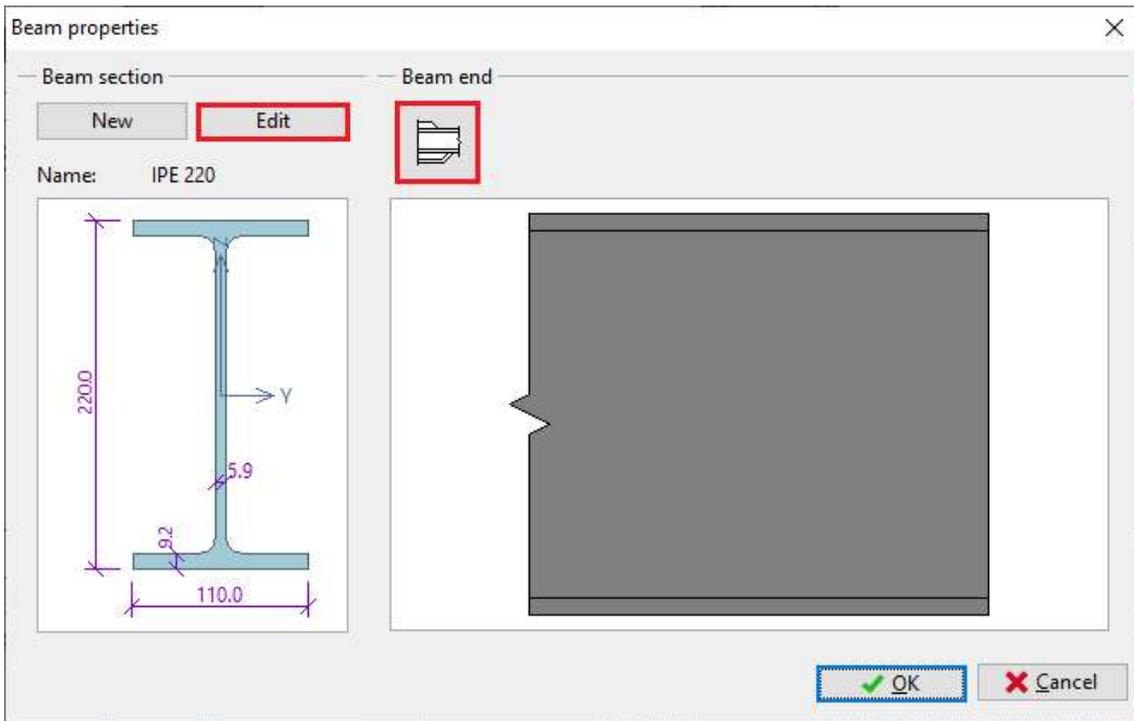
— Material characteristics —

f_y = 235.0 [MPa]

f_u = 360.0 [MPa]

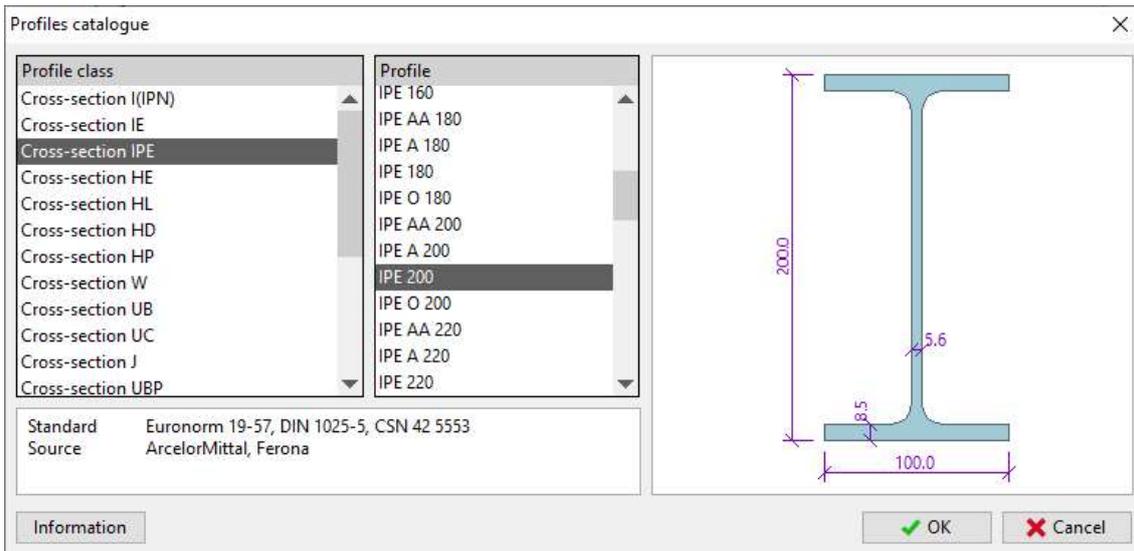
The button for edit of beam geometry

This window is similar to the window with column geometry. Left part of the window contains input of cross-section, right part input of haunches.



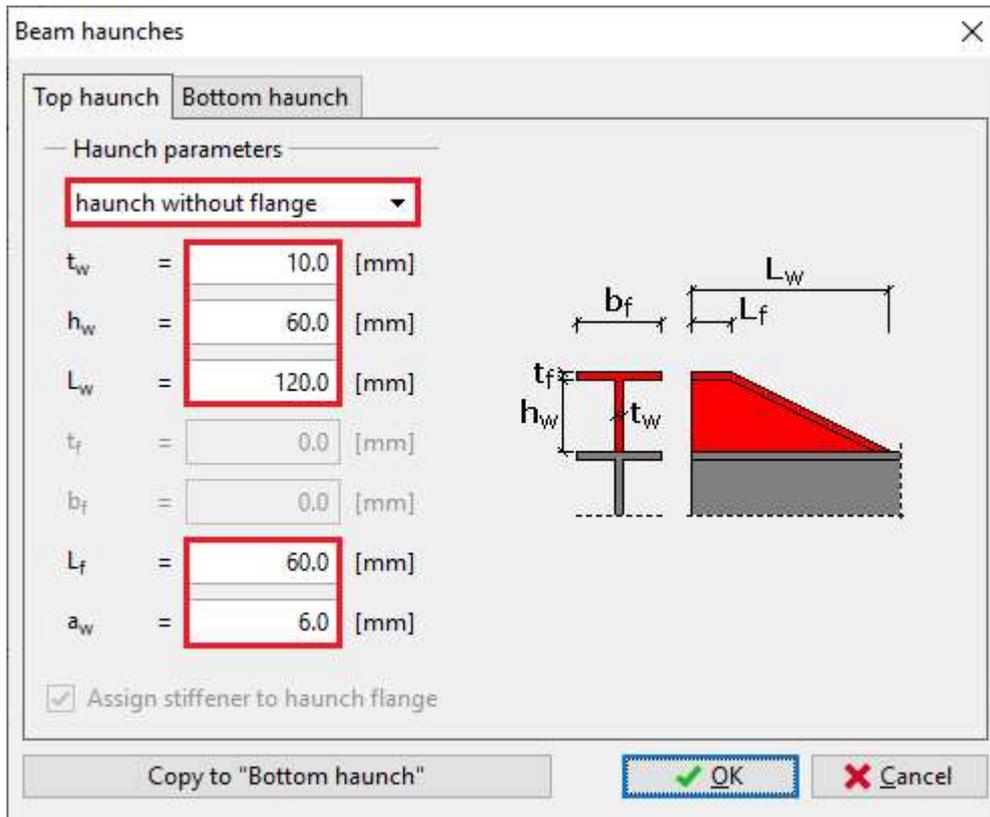
Buttons for edit of cross-section and haunch

The button "**Edit**" opens the database of rolled cross-sections. We change the cross-section to "**IPE 200**".



Selection of beam cross-section

The button in the part "**Beam end**" opens the window "**Beam haunches**". We select an option "**Haunch without flange**" and specify dimensions according to the following figure in the tab "**Top haunch**".

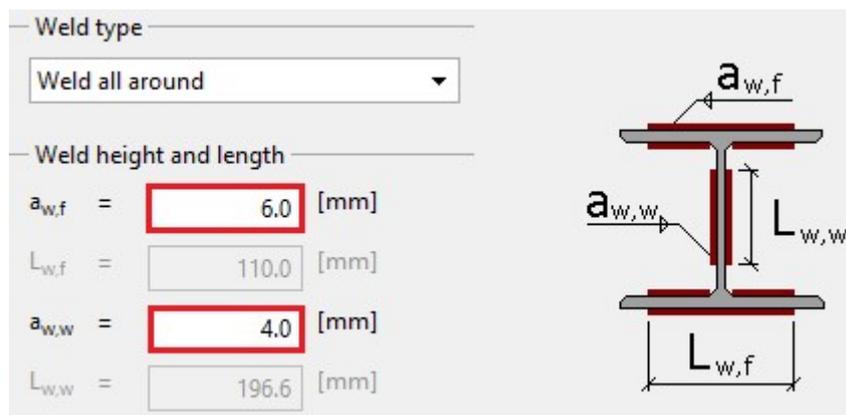


Haunch properties

We close windows "**Beam haunches**" and "**Beam properties**" by buttons "**OK**" and switch the tree menu into the mode "**Welds**".

Welds

Fillet welds all round will be used for the connection of the end plate to the beam. Therefore, we select weld type "**Weld all around**" and enter the throat thickness for flanges " $a_{w,f}$ " and the throat thickness for the web " $a_{w,w}$ ". Lengths are calculated automatically according to the geometry of the cross-section. Arbitrary lengths can be defined using weld type "**User defined weld**".



Weld properties

End plate

The input of end plate geometry follows. We open an appropriate window using the button "**Geometry adjustment**" in the bottom frame.

Geometry		Material	
Geometry adjustment		<input type="button" value="Catalogue"/> <input type="button" value="User defined"/>	
Dimensions		Name:	EN 10025 : Fe 360
Material characteristics			
b_p =	120.0 [mm]	f_y =	235.0 [MPa]
h_p =	280.0 [mm]	f_u =	360.0 [MPa]
t_p =	12.0 [mm]		
Openings - single row			
w_1 =	30.0 [mm]		

Button for input of end plate geometry

We specify end plate dimensions " b_p ", " h_p ", " t_p ", position of end plate relative to the beam edge " a_1 ", horizontal position of bolts " w_1 " and vertical positions of rows. Entered values are shown in the figure below. Only two bolt columns can be specified for rigid end plate. Four columns are permitted for hinged end plate. The input can be done with the help of input lines in the left part of the window or using active dimensions in the end plate figure in the right part of the window. The input has to be confirmed by the button "OK".

Edit end plate

<p>Size</p> <p>b_p = <input type="text" value="120.0"/> [mm]</p> <p>h_p = <input type="text" value="280.0"/> [mm]</p> <p>t_p = <input type="text" value="12.0"/> [mm]</p> <p>End plate position</p> <p>a_1 = <input type="text" value="-70.0"/> [mm]</p> <p>Bolts rows - horizontally</p> <p>w_1 = <input type="text" value="30.0"/> [mm]</p> <p>Bolt head position</p> <p><input type="checkbox"/> Bolt head on end plate side</p>	<p>Bolts rows-vertically</p> <p>Row count <input type="text" value="3"/></p> <table border="1" style="width: 100%; text-align: center;"> <thead> <tr> <th></th> <th>[mm]</th> </tr> </thead> <tbody> <tr> <td>e_1</td> <td>35.0</td> </tr> <tr> <td>$p_{1,1}$</td> <td>80.0</td> </tr> <tr> <td>$p_{1,2}$</td> <td>120.0</td> </tr> </tbody> </table>		[mm]	e_1	35.0	$p_{1,1}$	80.0	$p_{1,2}$	120.0	
	[mm]									
e_1	35.0									
$p_{1,1}$	80.0									
$p_{1,2}$	120.0									

Geometry of end plate

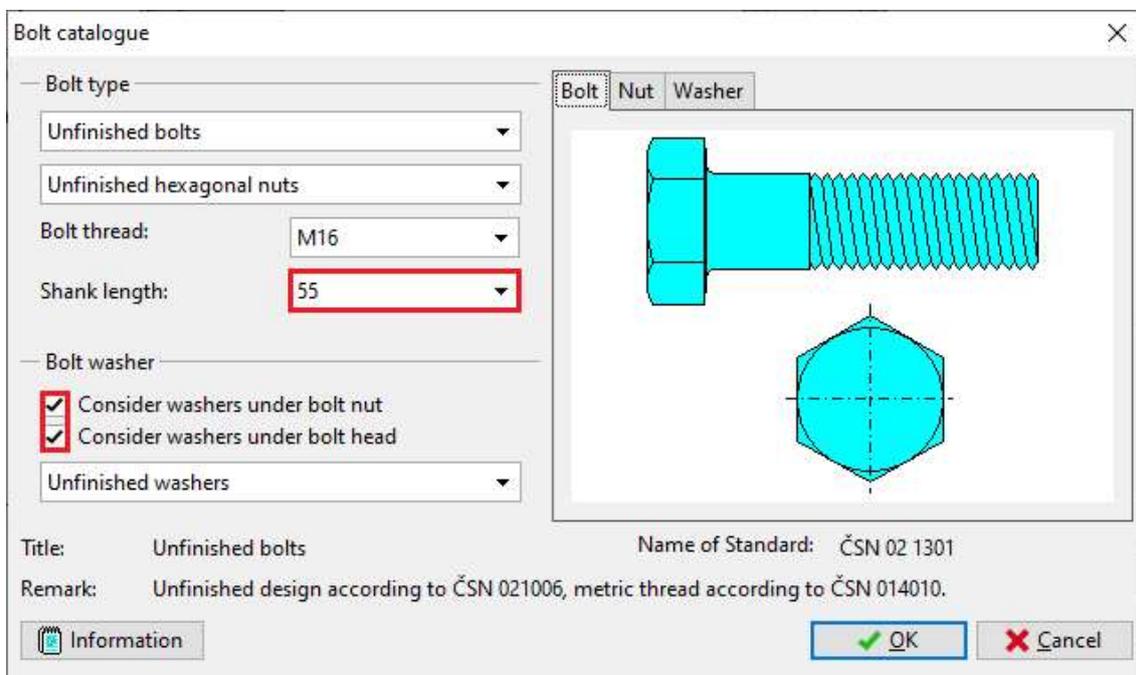
Bolts

The part "**Bolts**" contains the input of bolt type, dimensions and material. The type and dimension can be specified with the help of the button "**Catalogue**" in the part "**Bolt type**" of the bottom frame. The material can be selected using the button "**Catalogue**" in the part "**Bolt material**".

Bolt type		Bolt material	
<input type="button" value="Catalogue"/>		<input type="button" value="Catalogue"/> <input type="button" value="User defined"/>	
Type:	M16	Name:	Bolt 8.8
Standard:	ČSN 02 1301		
Bolts are not prestressed			
Shank characteristics		Material characteristics	
A_b =	201.062 [mm ²]	f_{yb} =	640.0 [MPa]
A_s =	156.668 [mm ²]	f_{ub} =	800.0 [MPa]
d =	16.0 [mm]		
d_0 =	18.0 [mm]		

The bottom frame with buttons for input of bolt properties

We select bolt diameter "**M16**" and shank length *55mm* in the window "**Bolt catalogue**". The shank length is an important input for the determination of shear plane. We tick on also washers under the bolt head and nut. The window has to be closed by the button "**OK**".



Bolt catalogue

— Bolt type

Unfinished bolts

Unfinished hexagonal nuts

Bolt thread: M16

Shank length: **55**

— Bolt washer

Consider washers under bolt nut

Consider washers under bolt head

Unfinished washers

Title: Unfinished bolts

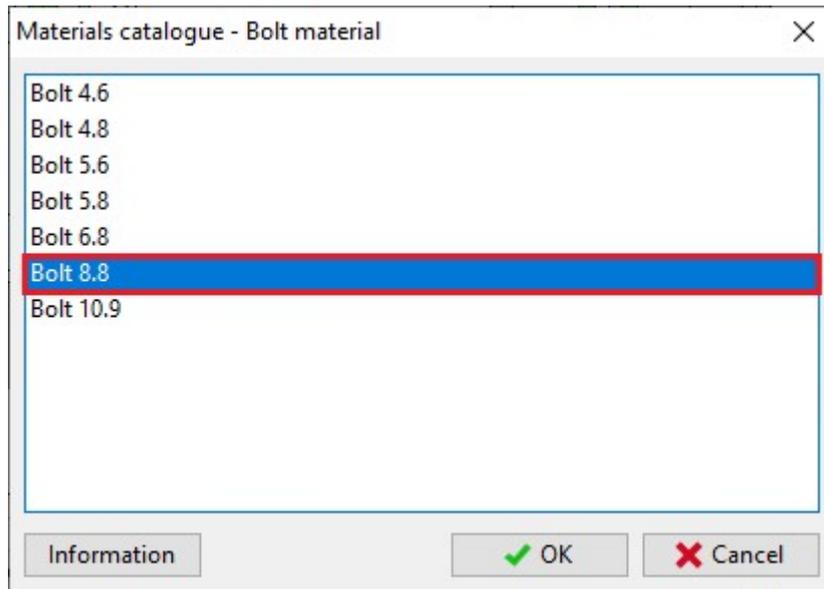
Remark: Unfinished design according to ČSN 021006, metric thread according to ČSN 014010.

Information

OK Cancel

Window "Bolt catalogue"

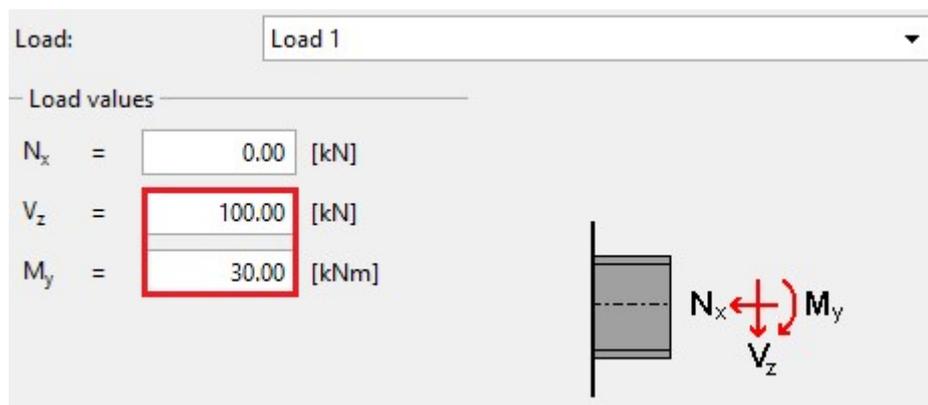
We choose the material "**Bolt 8.8**" in the window "**Materials catalogue**" and confirm the input by the button "**OK**".



Window "Materials catalogue"

Right flange

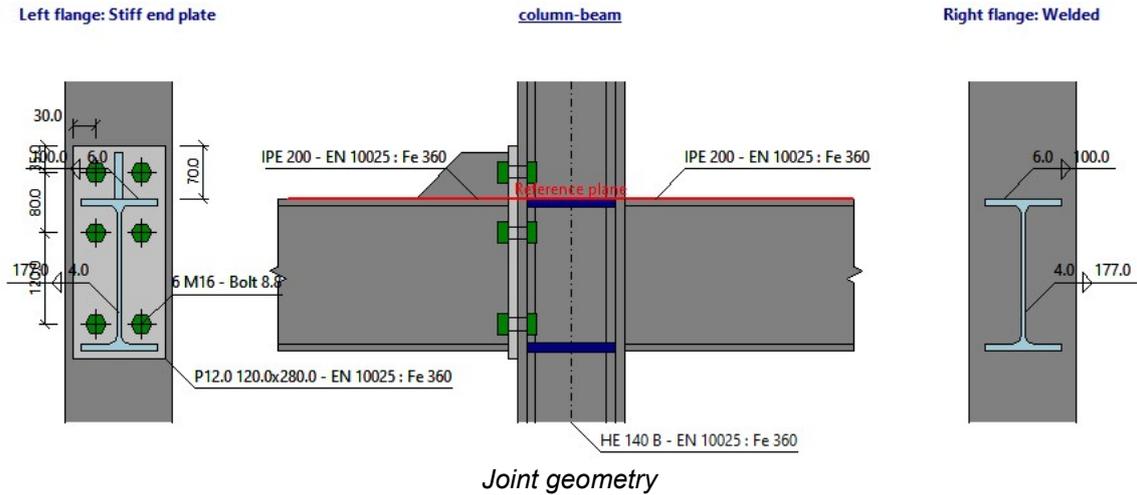
The connection of the second beam to the right flange is similar to the already entered connection. Only properties of the end plate are missing in this case. In the part "Load", we specify the bending moment $M_y = 30\text{kN}$ and shear force $V_z = 100\text{kN}$.



Load for the right connection

Warnings regarding the unbalance of forces in joint should disappear after the input of these forces. No changes are required in the part "Joint position".

We select also the profile *YPE 200* as the beam cross-section in the part "Beam". Finally, the weld type in the part "Welds" should be "Weld all around" and the throat thickness for flanges " $a_{w,f}$ " and for the web " $a_{w,w}$ " should be identical to the welds in the left connection.



Results

The total results are displayed in the right part of the bottom frame. They contain maximum utilization, the decisive load and connection and also brief results for all connections.

Overall check : PASS (57.87%)
Decisive load : L 1 - Load 1
Decisive joint : Connection to right side
CONNECTION TO LEFT SIDE - PASS (57.02 %)
 Bending capacity : $M_{y,Rd} = 52.77 \text{ kNm}$ (56.85%)
 Shear capacity : $V_{z,Rd} = 352.71 \text{ kN}$ (28.35%)
 Welds capacity : Max utilization (57.02%)
 Initial stiffness : $S_{j,ini} = 61443.53 \text{ kNm/rad}$
CONNECTION TO RIGHT SIDE - PASS (57.87 %)
 Bending capacity : $M_{y,Rd} = 51.84 \text{ kNm}$ (57.87%)
 Shear capacity : $V_{z,Rd} = 189.89 \text{ kN}$ (52.66%)
 Welds capacity : Max utilization (53.89%)

Total results

The detailed results for particular components are displayed in the basic mode of corresponding component (nodes "Left flange" and "Right flange" of the tree menu). These results contain detailed bearing capacities and decisive components. The results can be displayed for all entered loads. Available is also an option "Zero load". In this case, maximum bearing capacities for all components are displayed.

Results for loads:

L 1 - Load 1
 Zero load
 L 1 - Load 1

Decisive component
 row no.1 - End plate in bending $F = 149.55 \text{ kN}$
 row no.2 - Beam flange in compression $F = 121.16 \text{ kN}$
 Analysis
 $M_{y,Rd} = 52.77 \text{ kNm} > M_{y,Ed} = 30.00 \text{ kNm}$ PASS
Shear capacity
 Decisive component : Beam wall in shear
 Analysis : $V_{z,Rd} = 352.71 \text{ kN} > V_{z,Ed} = 100.00 \text{ kN}$ PASS
Welds capacity
 Critical point : web t. h.
 Max utilization : (57.02%)
Bend stiffness
 Initial stiffness : $S_{j,ini} = 61443.53 \text{ kNm/rad}$

Choice of loads

Steel truss joint

Introduction

This guide shows the input of truss joint in the program "Steel Connection". A web made of two L-profiles is bolted to a joint plate.

Chord: $2x L 120x12$
 Web: $2x L 90x8$ ($X = -20mm$, $Y = 0$, $D = 100mm$)
 Plate: $b_p = 230mm$, $h_p = 140mm$, $t_p = 12mm$, $h_{p1} = h_{p2} = 15mm$, $a_w = 3mm$
 Material: $EN 10025: Fe360$
 Bolt type: $M20 4.6$
 Position of bolts: $single\ row$, $e = [40, 70, 70] mm$
 Welds: $a_{w,1} = 3 mm$, $a_{w,2} = 3 mm$, $a_{w,3} = 3 mm$

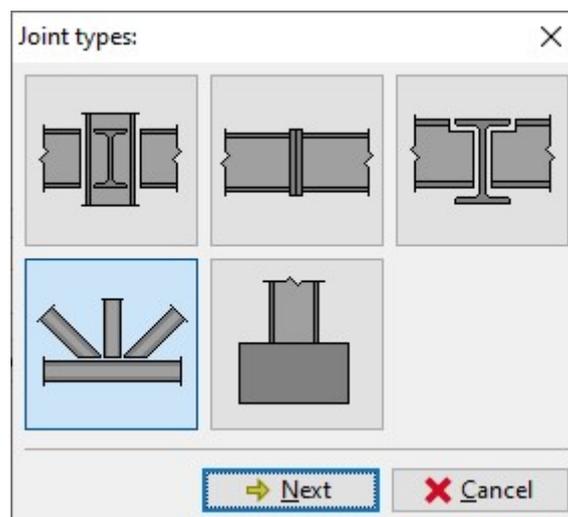
Creation of a new joint

As general characteristics (standard, global material etc.) were already specified in the previous guide, We start directly with addition of a new joint into the existing project. We use the button "Add" at the top of the tree menu on the left side of the window. The window "Add joint" appears, it is necessary to specify a name there. The input has to be confirmed by the button "OK".



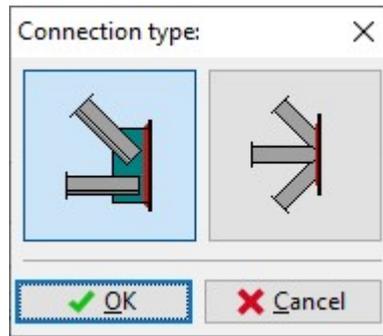
Input of joint name

The window "Joint types" appears. We select a truss joint (left option in the bottom row) and continue with the help of the button "Next".



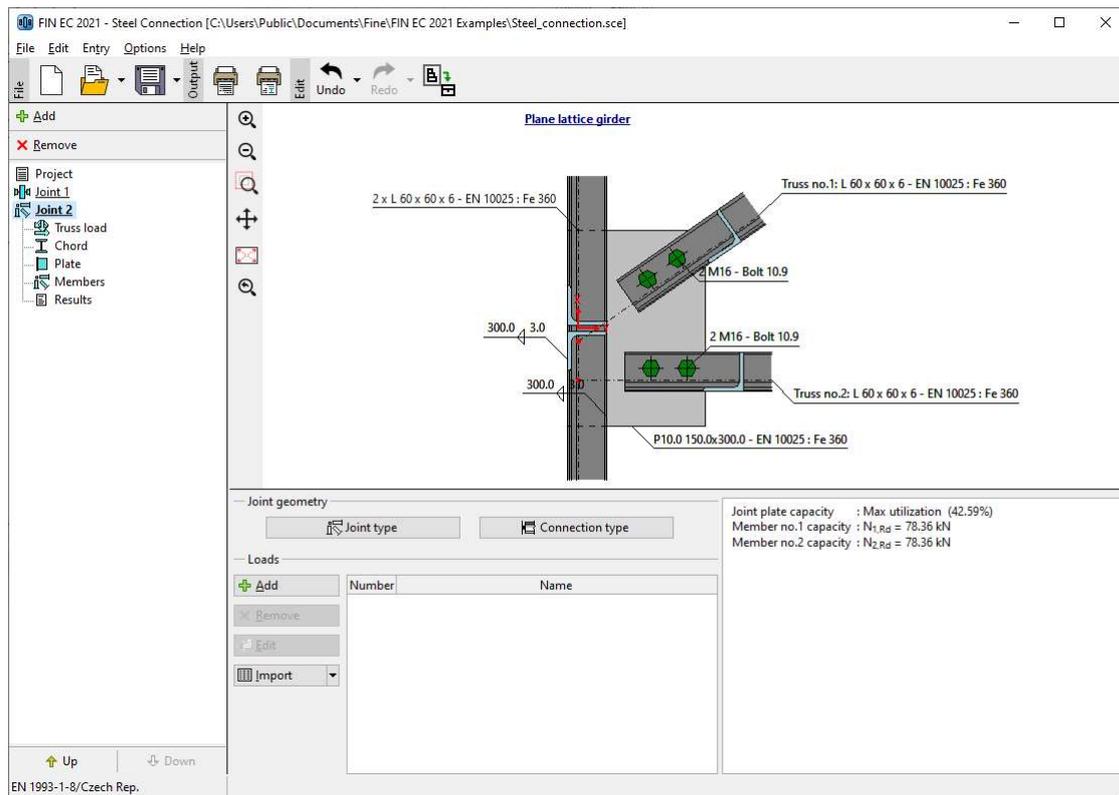
Joint type selection

The choice of connection follows. We select connection with fin plate and confirm the choice by the button "OK".



Choice of connection type

The initial geometry of the joint appears in the workspace.

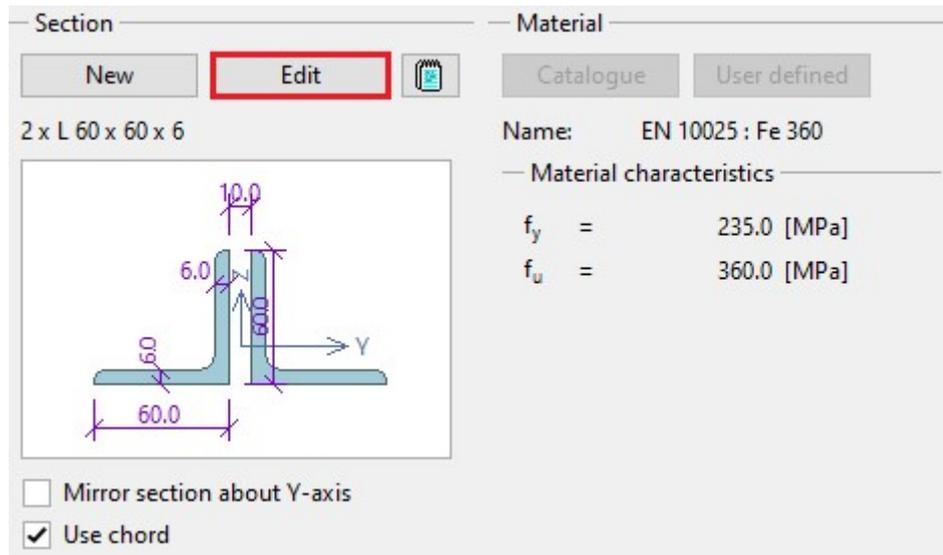


Initial geometry of the joint

The inputs for this joint are also organized into few parts, that are listed in the tree menu. As we are searching for the maximum resistance of the joint, we will skip parts **"Joint 2"** (list of loads) and **"Truss load"** (internal forces for connected webs).

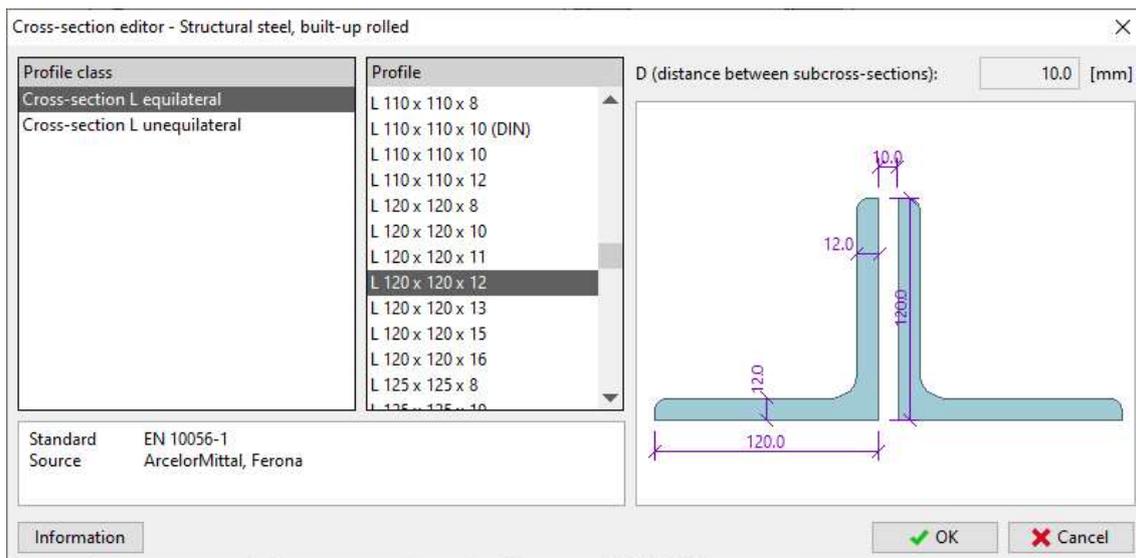
Chord

The part **"Chord"** contains the properties of continuous member of the joint. The input and consideration of the chord can be switched off with the help of the setting **"Use chord"**. Unsymmetrical L-profiles can be mirrored by using the setting **"Mirror section about Y-axis"**. We use the button **"Edit"** to modify the cross-section.



Chord properties

We select the profile class "**Cross-section L equilateral**" in the left column of the database in the window "**Cross-section editor**". The cross-section "**L 120x120x12**" should be selected in the second column of the database. The distance "**D**" between partial cross-sections cannot be edited, as it is selected automatically according to the plate thickness. The choice has to be confirmed by the button "**OK**".



Database of cross-sections

Plate

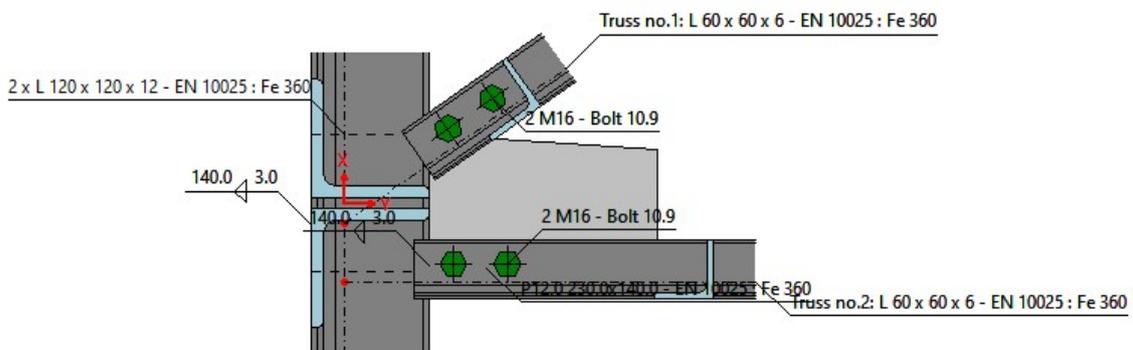
The geometry of the plate and weld thickness can be changed in this part. We use values according to the following figure. Buttons for an input of material are disabled, as the global material specified at the beginning is used.

Geometry		Material	
b_p	= 230.0 [mm]	Catalogue	User defined
h_p	= 140.0 [mm]	Name:	EN 10025 : Fe 360
t_p	= 12.0 [mm]		
h_{p1}	= 15.0 [mm]		
h_{p2}	= 15.0 [mm]		
Welding			
a_w	= 3.0 [mm]		

Plate properties

Workspace shows following geometry of the joint:

Plane lattice girder



Joint geometry

Members

This part contains a list of connected webs. As we need a joint with one connected web, we have to delete one of existing members. This member has to be selected in the table (highlighted by a bold font) and after that it can be deleted with the help of the button "Remove".

+ Add		← Edit		✖ Remove	
Number	Truss position				Truss section
	X[mm]	Y[mm]	D[mm]	α [°]	
1	-20.0	0.0	100.0	35.00	L 60 x 60 x 6
2	-80.0	0.0	70.0	0.00	L 60 x 60 x 6

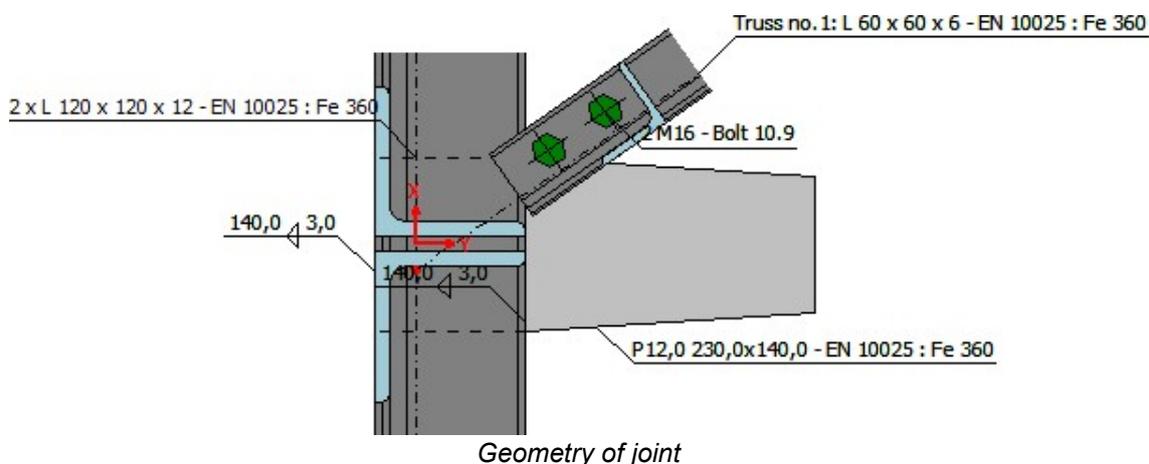
Deletion of the member number 2

The confirmation window appears, we use the button "OK".

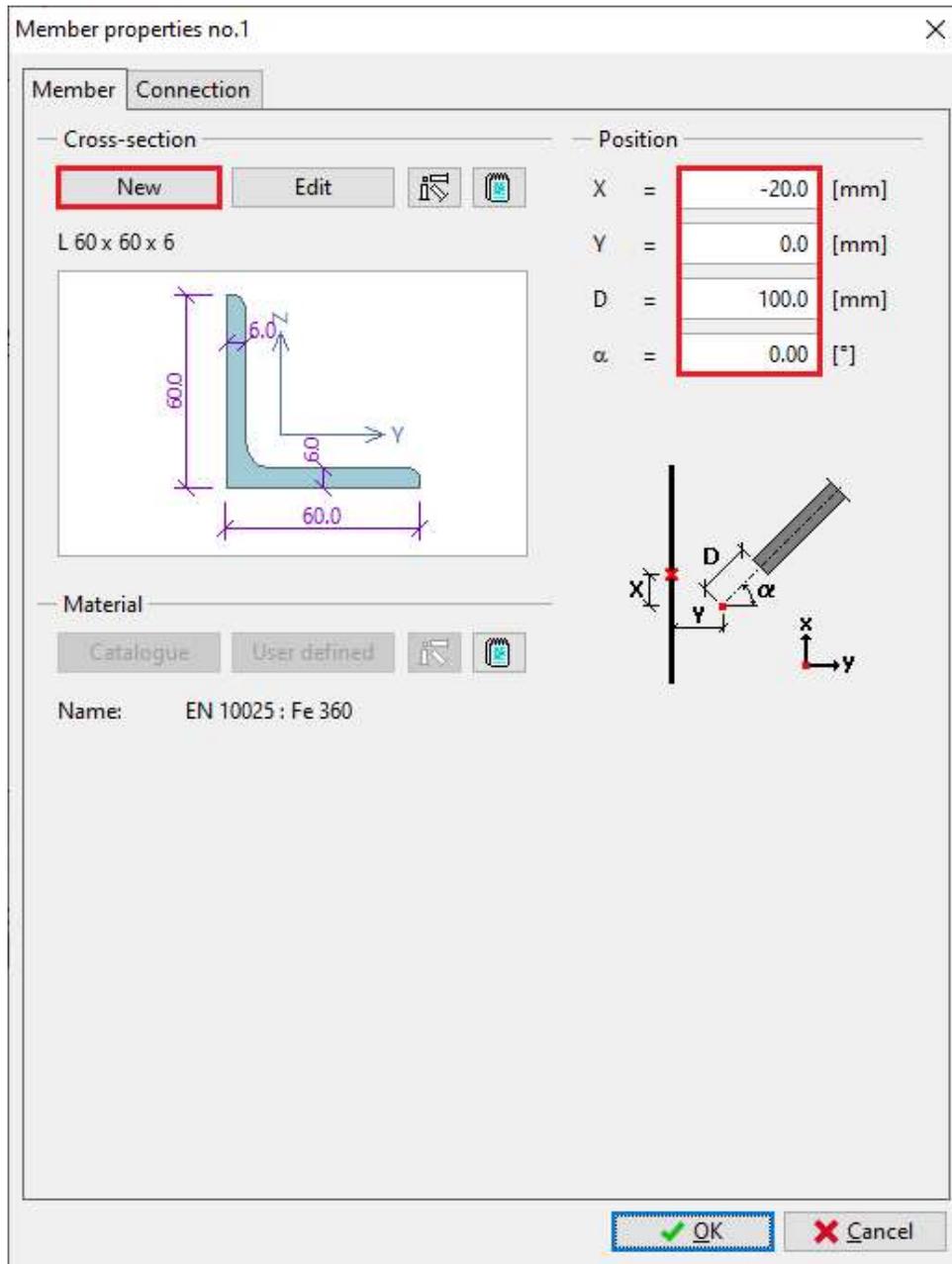


Confirmation window

Only one connected member remains in the joint. This member has a wrong position. Therefore, we have to change its position.

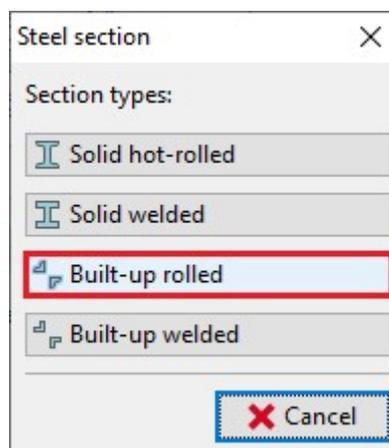


For editing the member, we can use the button "**Edit**" above the table of members or double-click on the corresponding row of the table. The window "**Member properties**" contains two tabs: "**Member**" and "**Connection**". We change the position of the member with the help of input fields on the right side of the tab "**Member**". We also have to change the cross-section. We use the button "**New**" in the left upper corner of this tab.

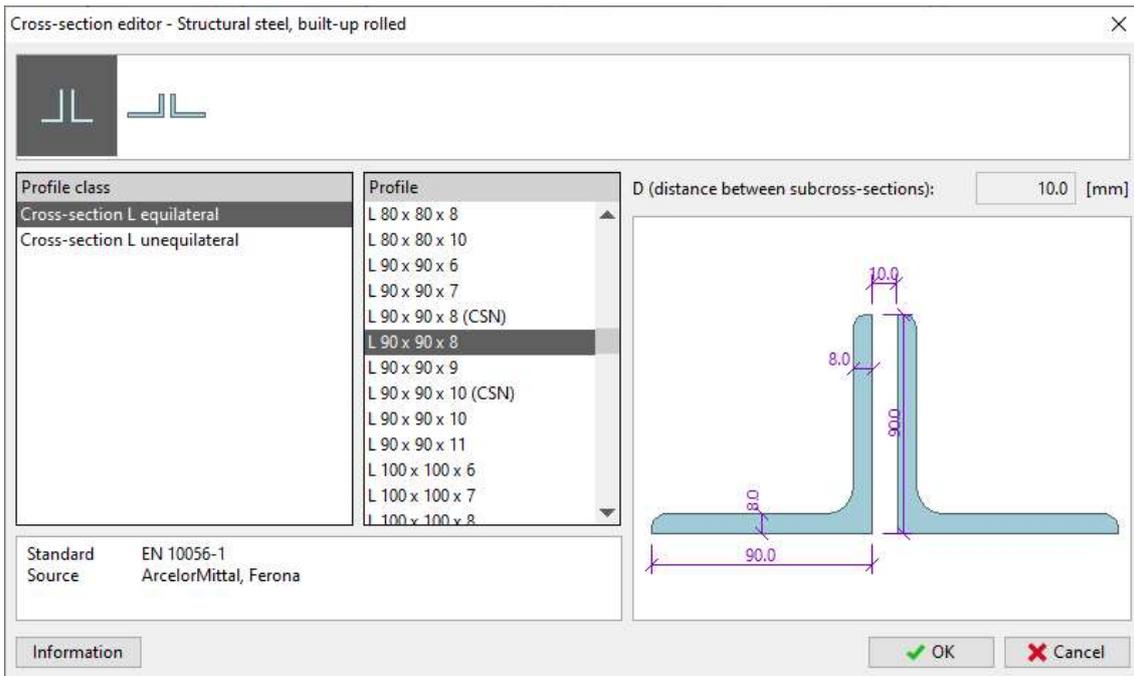


Change of member's position and cross-section

We choose type "**Built-up rolled**" in the next window.



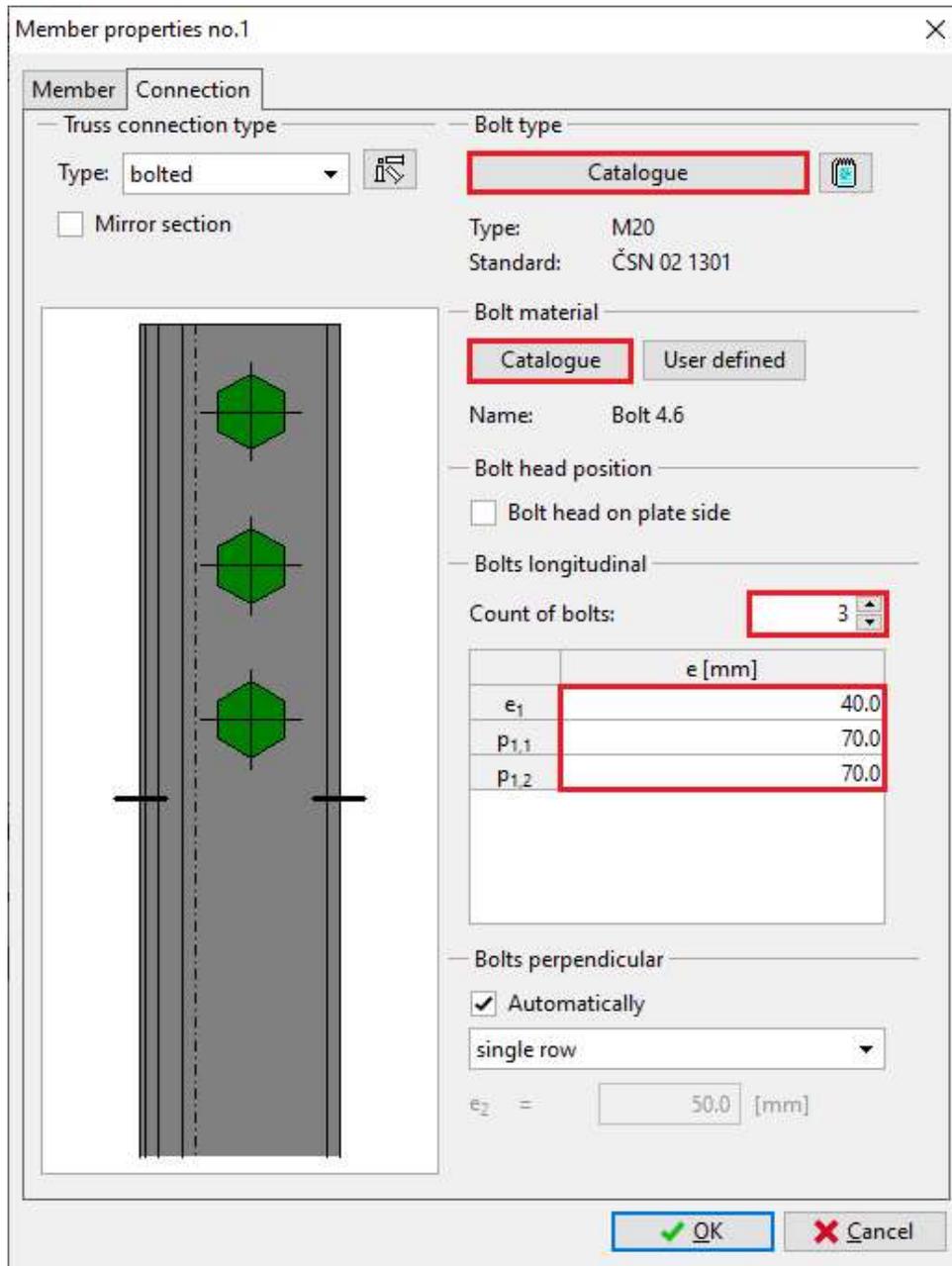
The database of cross-sections opens. The type "**L 90x90x8**" has to be selected here.



Database of cross-sections

We close the window using the button "OK".

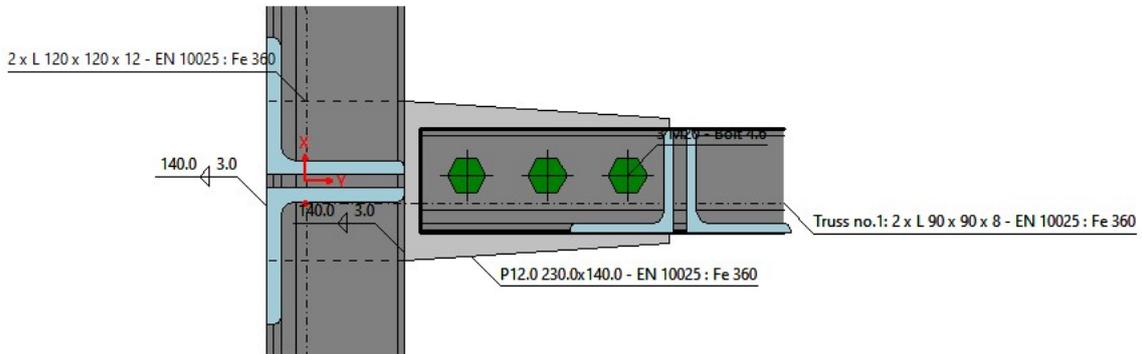
We switch to the tab "Connection" in the window "Member properties". It is necessary to increase the number of bolts to 3 and specify the distances between bolts (40,70,70).



Position of bolts

We also change the material of bolts. In the same tab "**Connection**" of "**Member properties no.1**" we use the button "**Catalogue**" for selection of the material and select the material "**Bolt 4.6**". We close the window "**Member properties no.1**" by the button "**OK**". Final geometry of the joint can be seen on the following picture:

Plane lattice girder



Geometry of joint

Results

The results show that the maximum resistance of the connection is **202,49 kN**. The plate would have an utilization **60,17%** in this case. Detailed results are available only for **"Zero load"**, as we have not specified any load. The detailed analysis can be printed easily using the button "  ".

Results for loads:

Zero load



Resistance - Truss no.1
 Decisive component : Bolt group pullout incl. section part without perp. wall
 Resistance : $N_{1,Rd} = 202.49 \text{ kN}$

Resistance - Joint plate (60.17%)
 Bending capacity : $M_{y,Rd} = 6.73 \text{ kNm}$ (60.17%)
 Shear capacity : $V_{z,Rd} = 216.49 \text{ kN}$ (0.00%)
 Axial capacity : $N_{x,Rd} = 374.97 \text{ kN}$ (54.00%)
 Welds capacity : Max utilization (51.14%)

Joint plate capacity : Max utilization (60.17%)
 Member no.1 capacity : $N_{1,Rd} = 202.49 \text{ kN}$

The button for printing of detailed results

Web to chord connection

Introduction

This guide shows how to create a new joint using an existing one. The joint created in the previous guide "**Steel truss joint**" will be copied and will be converted into welded joint.

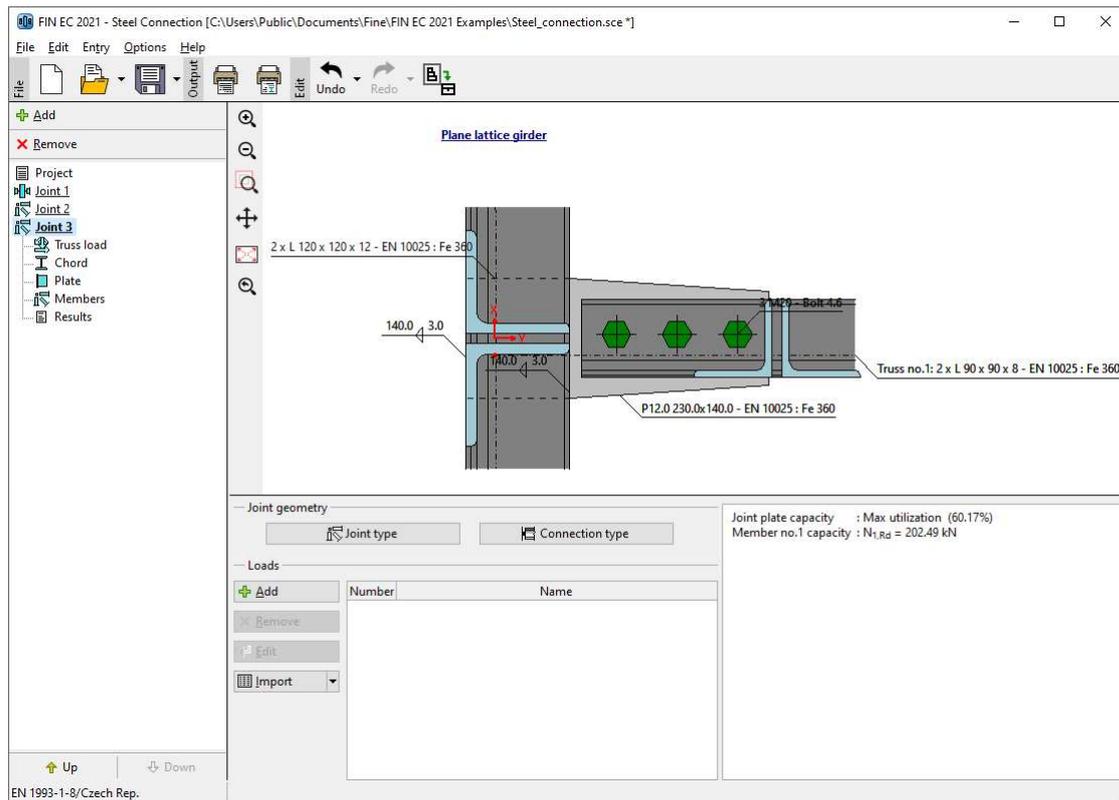
Creation of a new joint

Similarly to previous joints, we start with insertion of a new joint with the help of the button "**Add**" in the tree menu. We select an option "**Copy existing joint**". The list of existing joints appears on the right side, we select "**Joint 2**" there. Additionally, we change the name and close the window by the button "**OK**".



Choice of sample joint

After the closing of the window, the new joint including appropriate data structure appears in the tree menu.



Basic screen for the new joint

Changes of topology

We only need to change the connection style. Therefore, we switch directly to the part "**Members**" of the tree menu and edit the connected member. We can use the button "**Edit**" above the table or double-click on the corresponding row of the table.

Number	Truss position				Truss section
	X[mm]	Y[mm]	D[mm]	α [°]	
1	-20.0	0.0	100.0	0.00	2 x L 90 x 90 x 8

Editing of connected member

We switch to the tab "**Connection**" and change the connection type in the right upper corner to "**Welded**". The parameters of welds appears, we keep default settings. The change has to be confirmed by the button "**OK**".

Member properties no.1

Member Connection

Truss connection type

Type: **welded**

Mirror section

Weld parameters

fillet weld around

$a_{w,p1}$ = 3.0 [mm]

$L_{w,p1}$ = 216.1 [mm]

$a_{w,p2}$ = 3.0 [mm]

$L_{w,p2}$ = 216.1 [mm]

Use end weld

$a_{w,v}$ = 3.0 [mm]

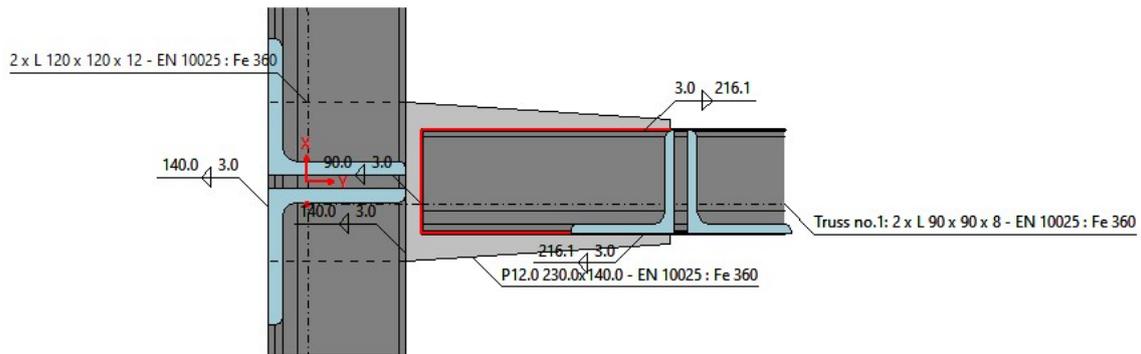
$L_{w,v}$ = 90.0 [mm]

OK Cancel

Change of connection type

The final joint is shown on the following figure.

Plane lattice girder



Final geometry of joint

Results

The results show that the maximum resistance of the connection is **152.4 kN**. The plate would have a utilization **38.6%** in this case. The decisive component is welded connection.

Results for loads:	
Zero load	Joint plate capacity : Max utilization (38.60%) Member no.1 capacity : $N_{1,Rd} = 152.40$ kN
Resistance - Truss no.1 Decisive component : Weld at protruding arm Resistance : $N_{1,Rd} = 152.40$ kN	
Resistance - Joint plate (38.60%) Bending capacity : $M_{y,Rd} = 8.48$ kNm (35.93%) Shear capacity : $V_{z,Rd} = 227.94$ kN (0.00%) Axial capacity : $N_{x,Rd} = 394.80$ kN (38.60%) Welds capacity : Max utilization (38.49%)	

Results