

Getting Started



Version 7.0
28.01.2013.

CONTENT

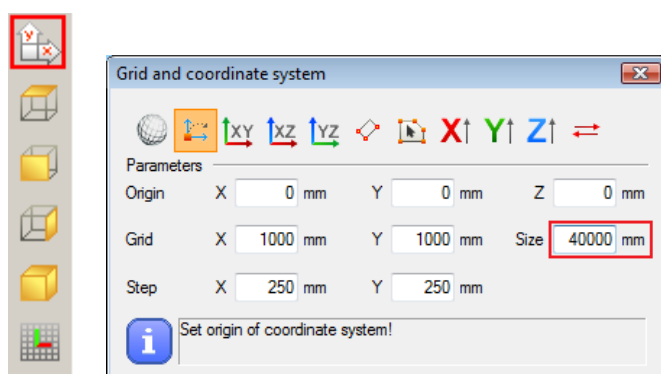
1. Modeling	3
2. Loads and Load Cases	9
3. Load combination	15
5. Standard Design	21
6. Section module – csSection	23
7. Joint Module - csJoint	27
8. Documentation	32
8.1 Creating snapshots	35
8.2. Snapshot manager	35
8.3. Model information	36

GETTING STARTED WITH CONSTEEL

1. MODELING

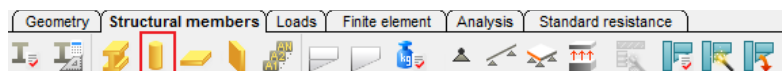
There is a default grid which provides help to place beams and columns. This grid is moveable, rotatable and the size can be adjusted.

4.2.1

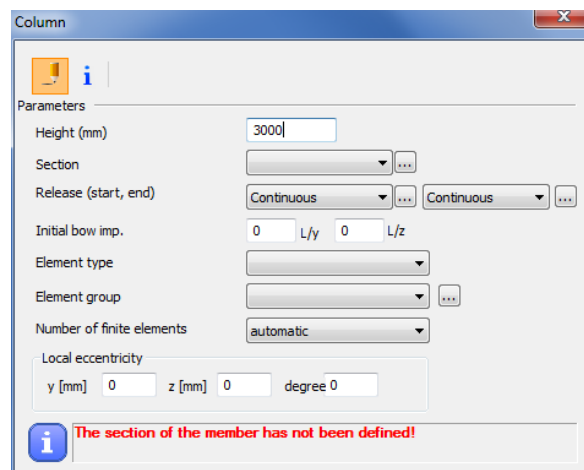


First click on the **COLUMN** icon to place a vertical column.

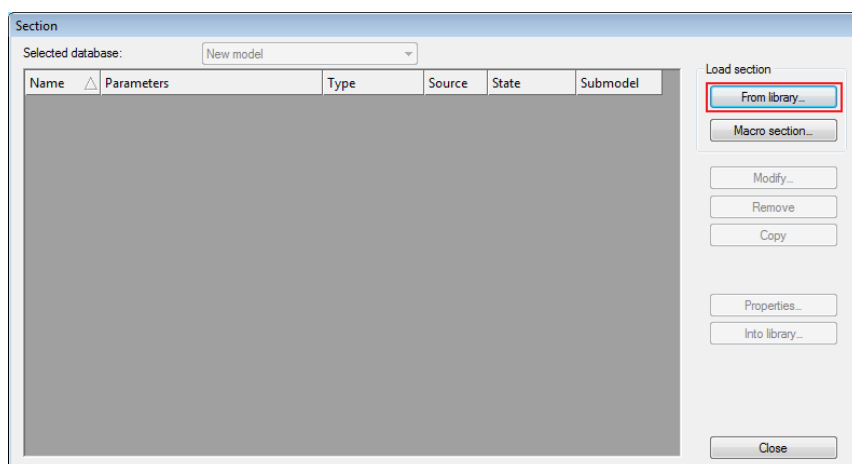
5.2.1



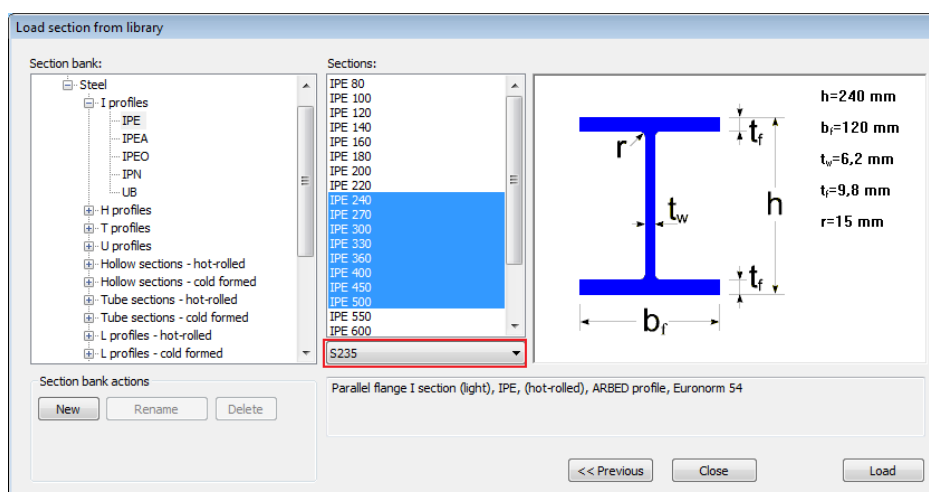
Set the height of the column to 8400 mm. To define the cross section of the column, click the button with the three dots on it.



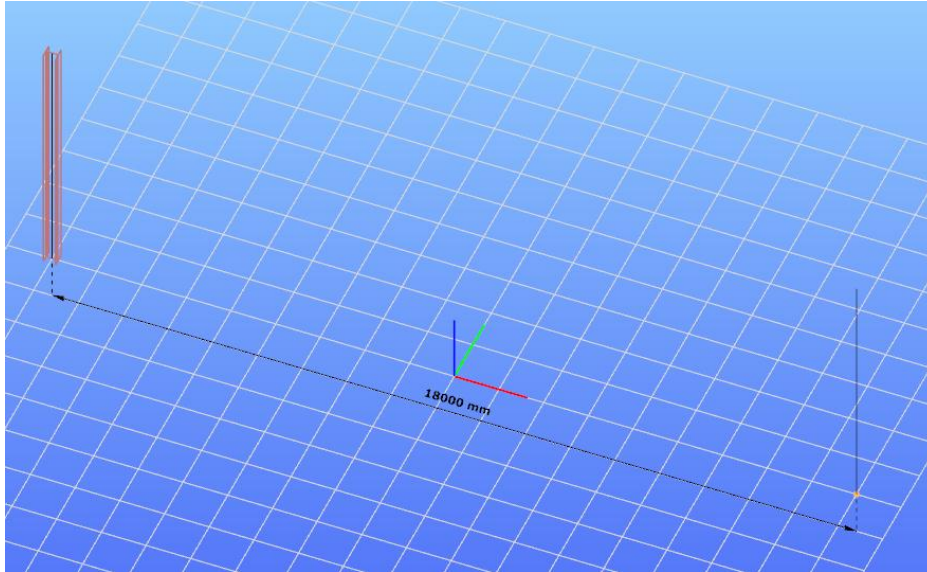
Then click on **FROM LIBRARY** to open the standard section catalog and load some sections. Multiple sections can be loaded at the same time by using the Windows standard functions i.e. Shift or Ctrl button + left mouse click selection.



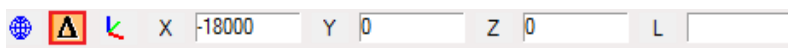
Before clicking on the **LOAD** button set steel material grade to S235. According to the ConSteel philosophy material grade is a property of the section and not the member. Select to following sections to be loaded: IPE 300 - IPE500, HEA 300 – HEA 500. Click on Close and select from the drop down menu section HEA 300 for the column.



Set the section to HEA 300 and height to 8400 mm. Place the column to the grid points like in the image below. The span of the frame is 18.000 mm.



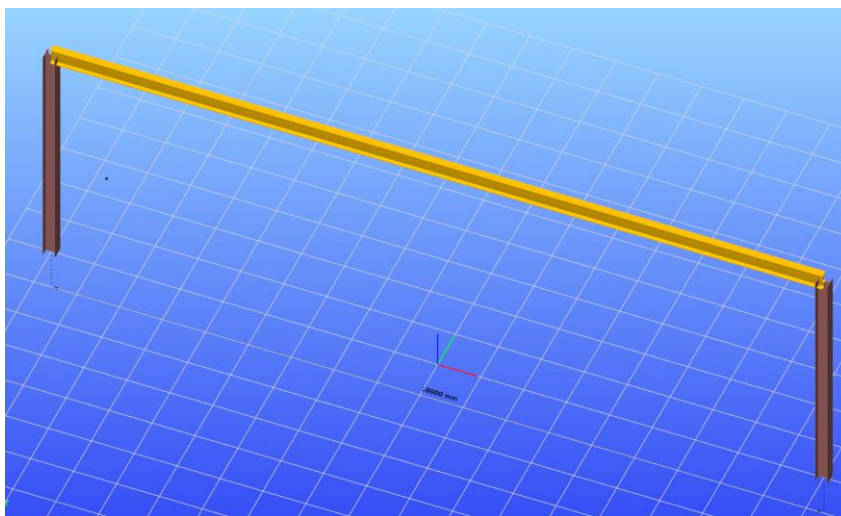
The default grid size is 20.000 mm. In order to make the modeling easier absolute and relative coordinates can be used.



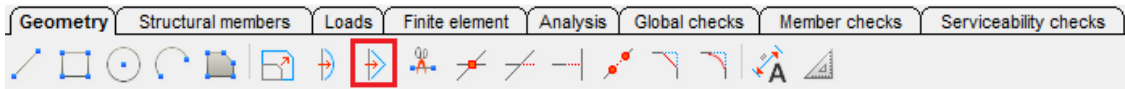
There are two ways to place columns or any object in ConSteel. First is to use the grid points and snaps, second is to use the coordinates. It also good to kept in mind that coordinate system can be changed to local.

To place the beam click on the **BEAM** icon on Structural members bar. Select IPE 400 section size and leave all the other settings as default. Place the beam between the two top points of the columns.

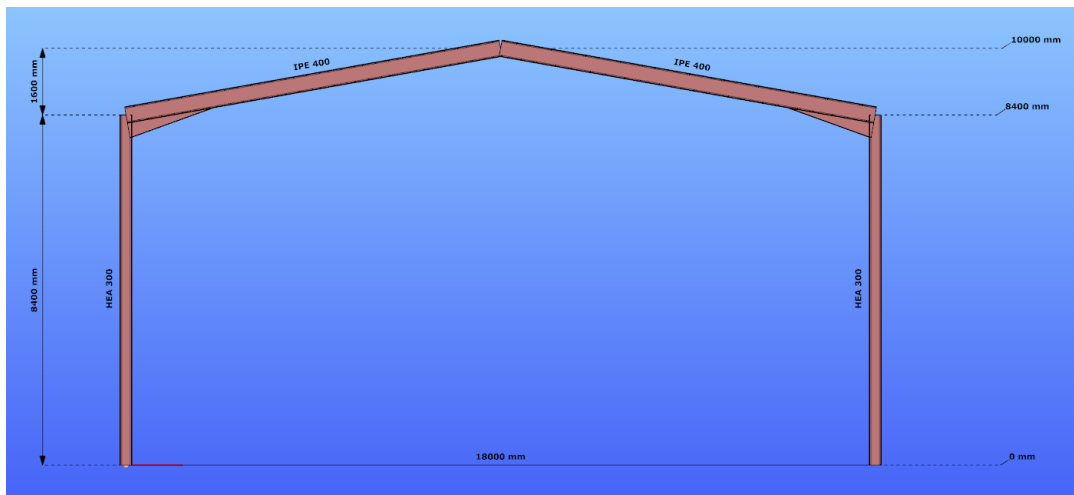
5.2.1



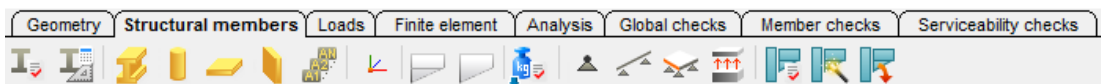
To refract the beam select Geometry tab and click on **REFRACT SELECTED LINE ELEMENTS** icon. Follow the instructions on screen and set the cursor to the middle point of the beam. Press Z button and set the Z coordinate of the point to 10.000 mm if absolute coordinate system is set or 1600mm if relative coordinate system is set. Press **ENTER** to refract the beam.



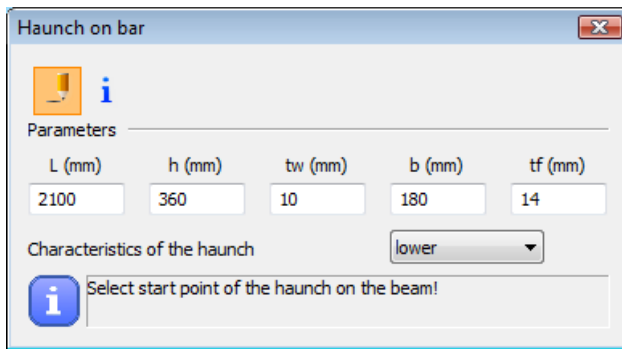
The final geometry looks like as in the image below.



Haunch can be placed on the beam ends by clicking on the dedicated **HAUNCH** icon.

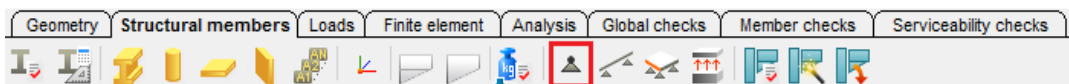


Set the haunch parameters like below and follow the instructions on the dialogue. First pick the end point of the beam then the direction of the haunch.

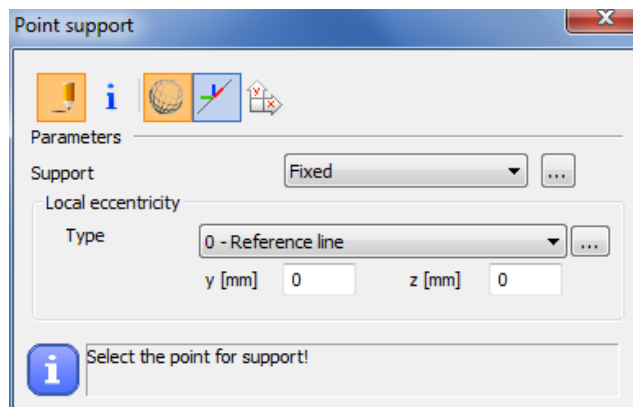


The next step of modelling is to place the supports. Click on the **POINT SUPPORT** icon.

5.5

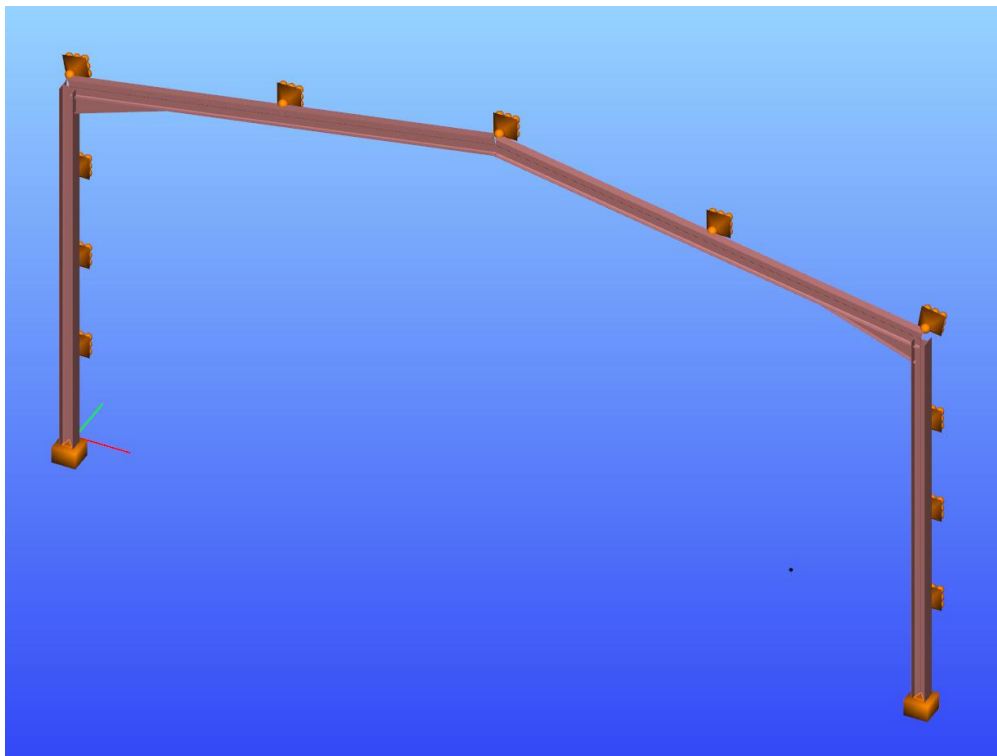
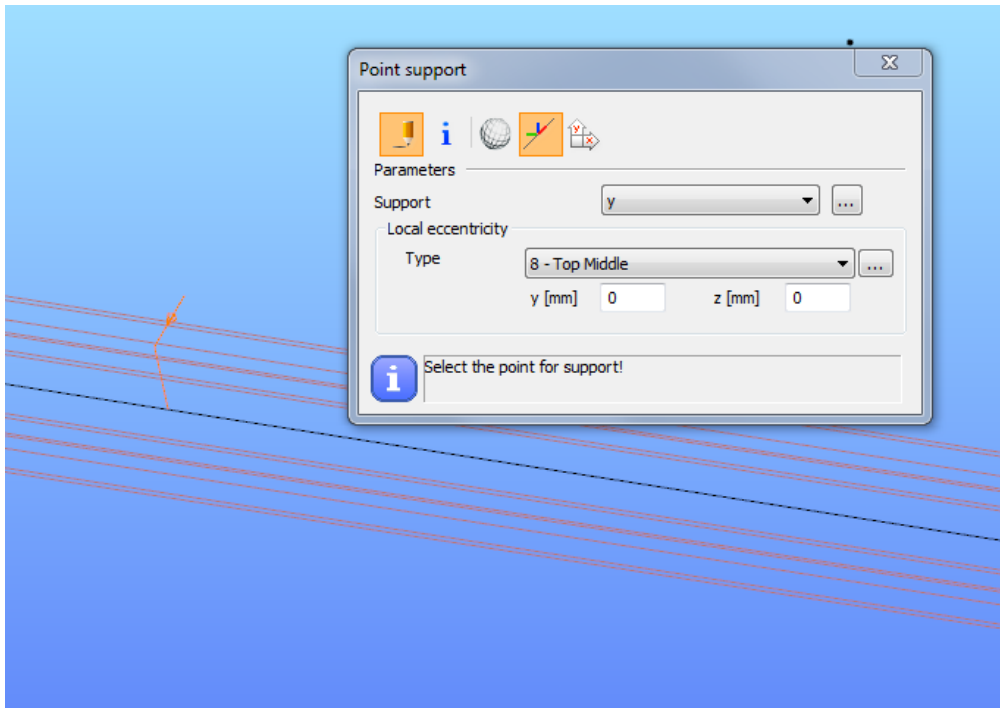


Select Fixed support and select the lower end points of the columns.



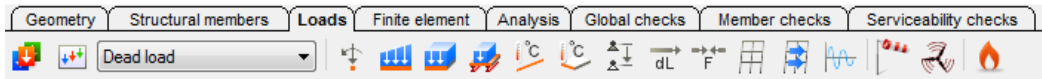
Because ConSteel model is a real 3D model it is also important to place supports perpendicular to the frame to simulate the bracing system.

Select support 'y' and place to the positions like in the image below. The supports on the beam are supporting the upper flange of the section so local eccentricity must be set to 'Top middle'.



2. LOADS AND LOAD CASES

All the loading functionality connecting with the load modeling is placed on the **LOADS** tab.

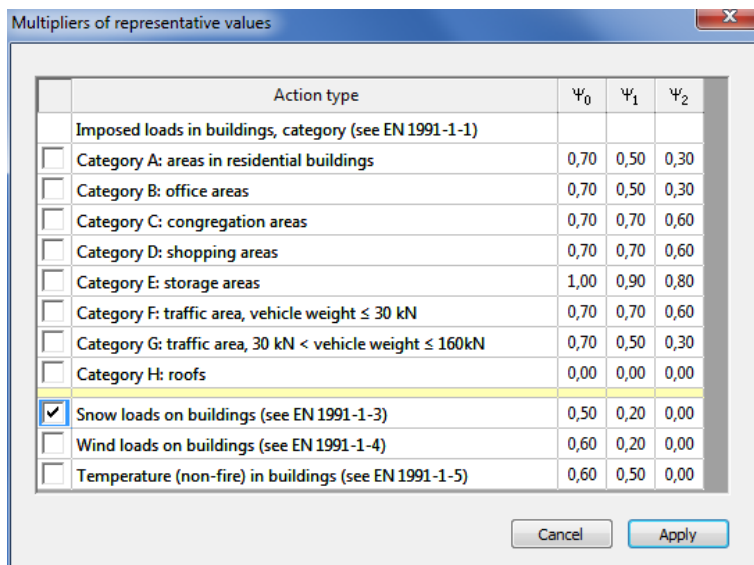
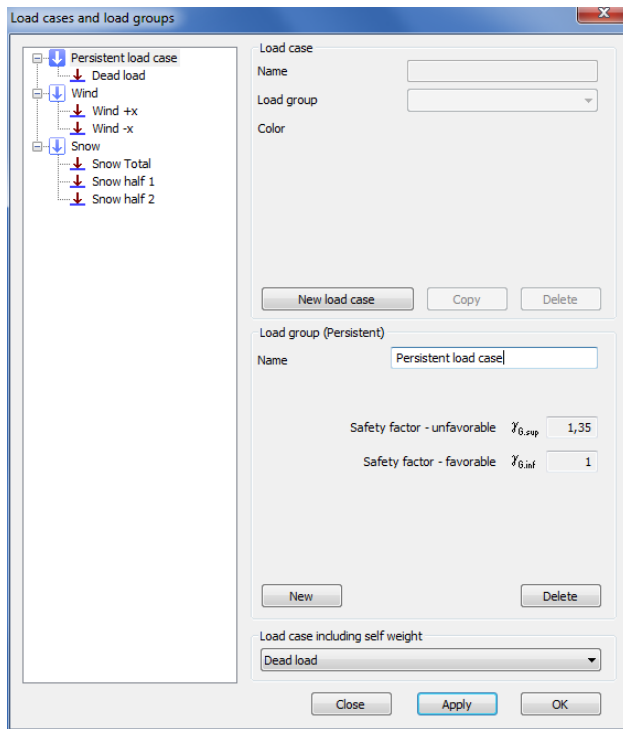


6.2

Before placing any loads in the model load cases and load groups must be defined. In a Load group several load cases can be defined. Precise load groups needed to automatically generate load combinations.

For the portal frame example three load groups would be used: Persistent, Wind and Snow.

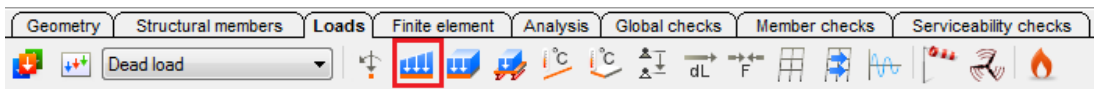
For each load group safety and combination factors must be given. These factors can be edited manually or by clicking the 'Standard values of factors' icon. Here typical cases can be selected.



ConSteel can add the structure's dead load to a selected load case if necessary. In order to do so please select the appropriate load case (Dead load) at the bottom of the window. Create the load groups and load cases like in the image above.

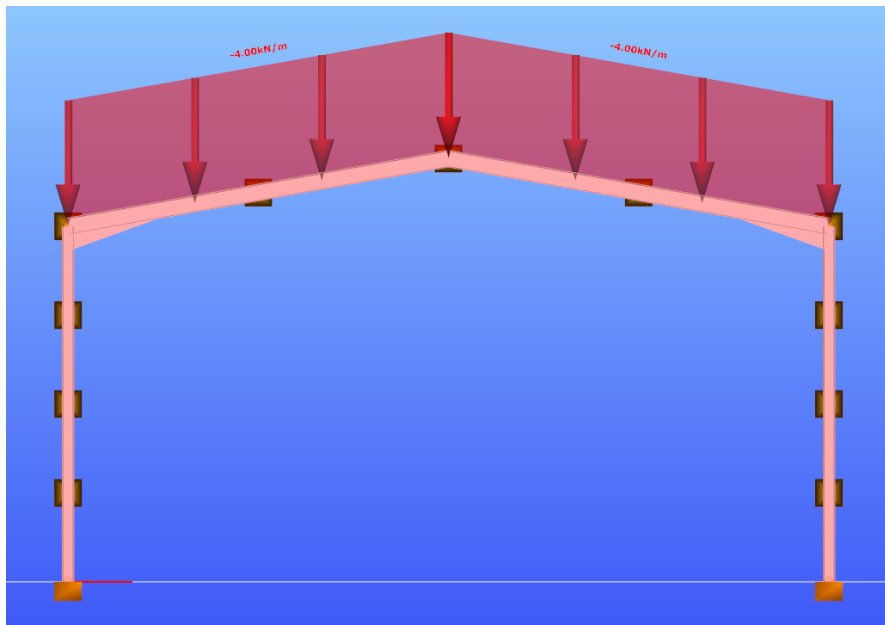
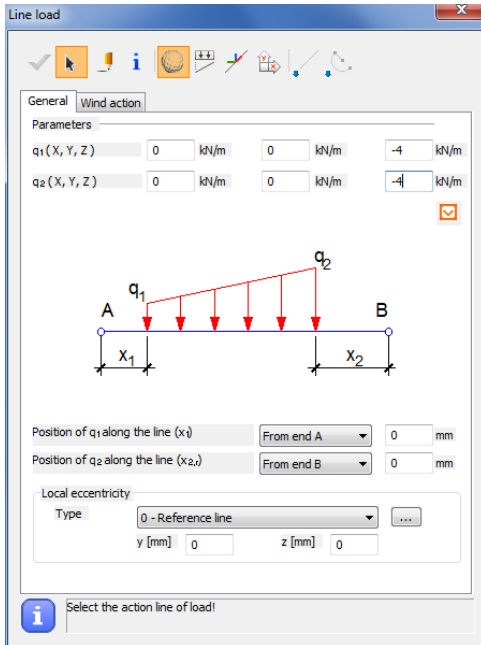
The next step is to define the loads for the load cases. First select from the drop down menu in the **LOAD** tab the **DEAD LOAD** load case. Then select **LINE LOAD** from the **LOADS** tab.

6.4.2



6.4.2

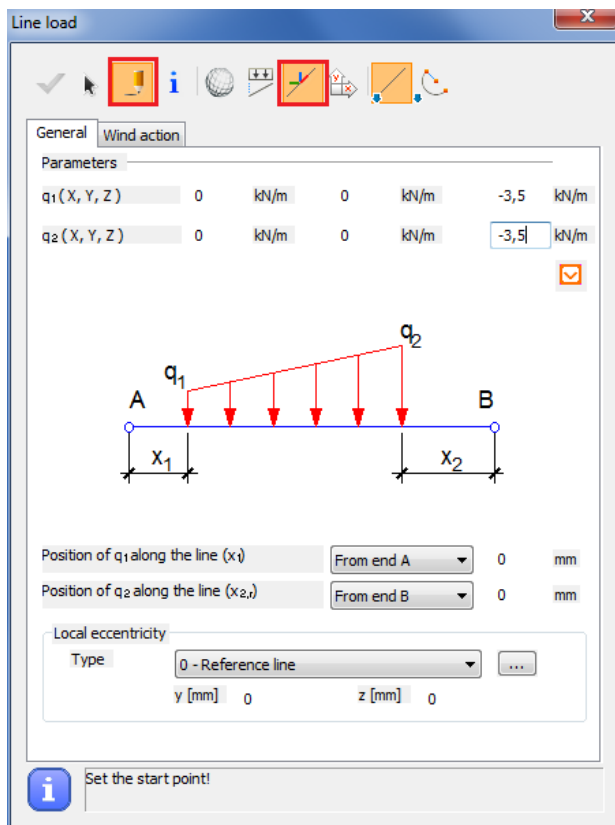
To define line load set the load density to -4 kN/m and click on the beams.



The next step is to define the Wind load from +X direction. It is easy for the columns just the load density and the direction is different the method is the same as previous case. But wind load for the beams is a little bit trickier because load density is not the same along the beam and it is perpendicular to the beam. Therefore set line load dialogue to **DRAW** and **LOCAL COORDINATE SYSTEM**. When the **DRAW**

function is set load can be defined between any picked points along the member including also the start and end point.

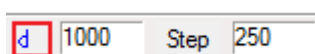
LOCAL COORDINATE SYSTEM option can be used to define load according to the member's local coordinate system.



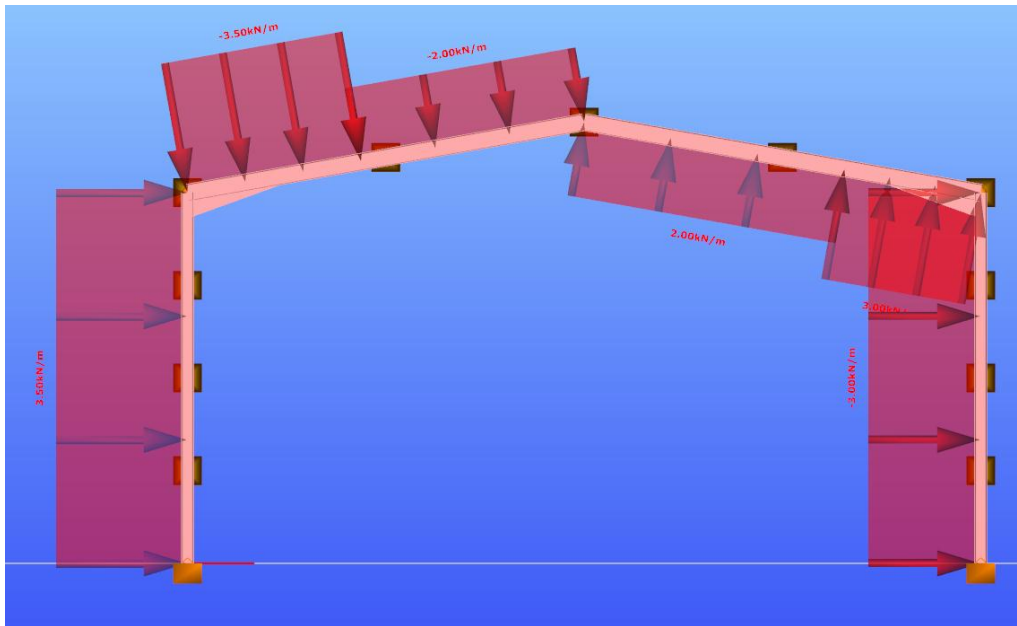
Before drawing the action line of the line load another setting need to be changed. At the bottom right corner of the status bar additional functions can be found in order to help editing and construction of the model. A special line snap can be activated which visualizes snap points on lines (reference line of bar members, line loads, line supports or edge line of surface members, surface loads, surface supports) by red rectangles. These snap points appear only if the mouse approaches the line, and fade away three seconds after stop of movements around the line.

In the first button ('n', '%' or 'd') the type of the division can be set: number of dividing parts ('n'), percentage of length ('%'), and distance from one end ('d'). In the last case the count starts at the line end first approached by the mouse.

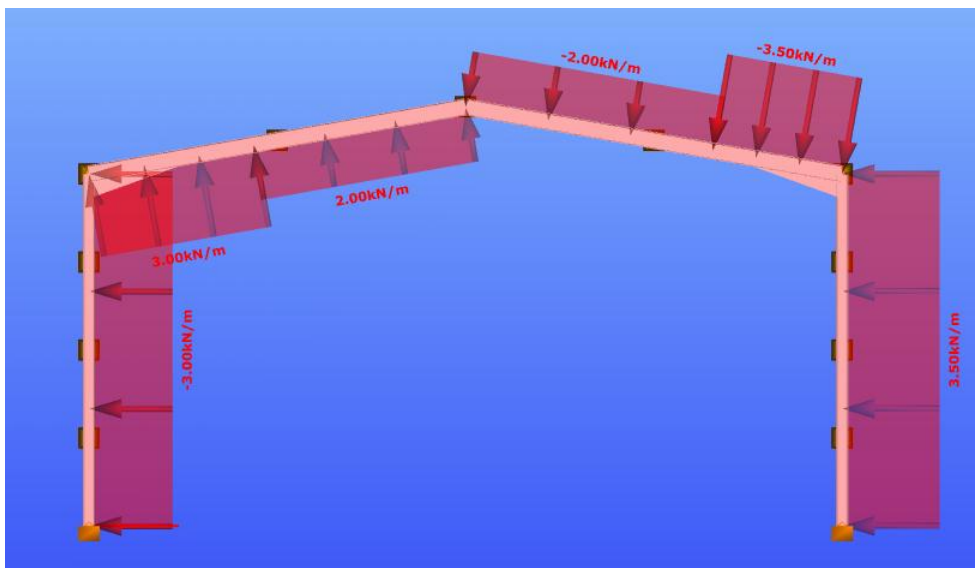
In this case 'd' is needed to set the distance from the end point.



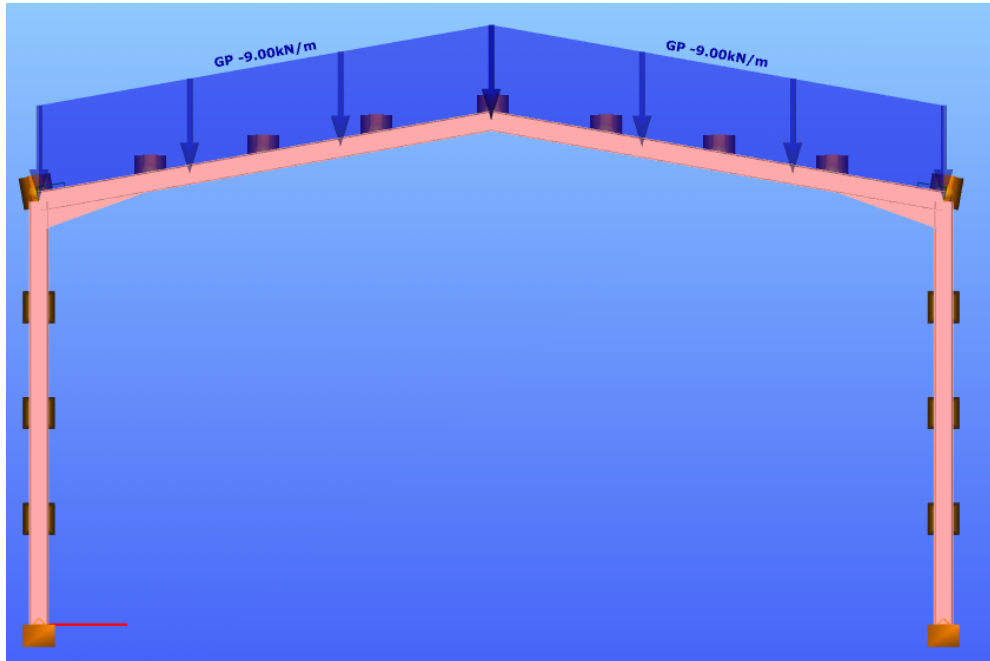
Partial line load starts at the end of the beam and ends 4000 mm (4 red dots).



Using the same method, create the wind load from -X direction.

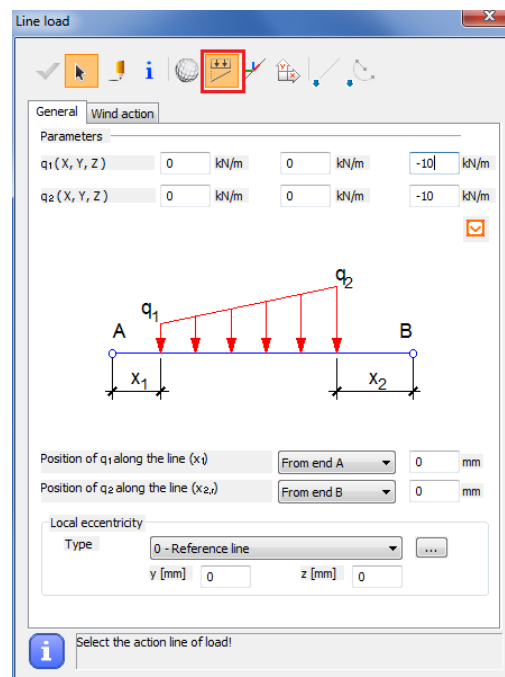


Creating the snow load is much easier: it contains only two line loads.

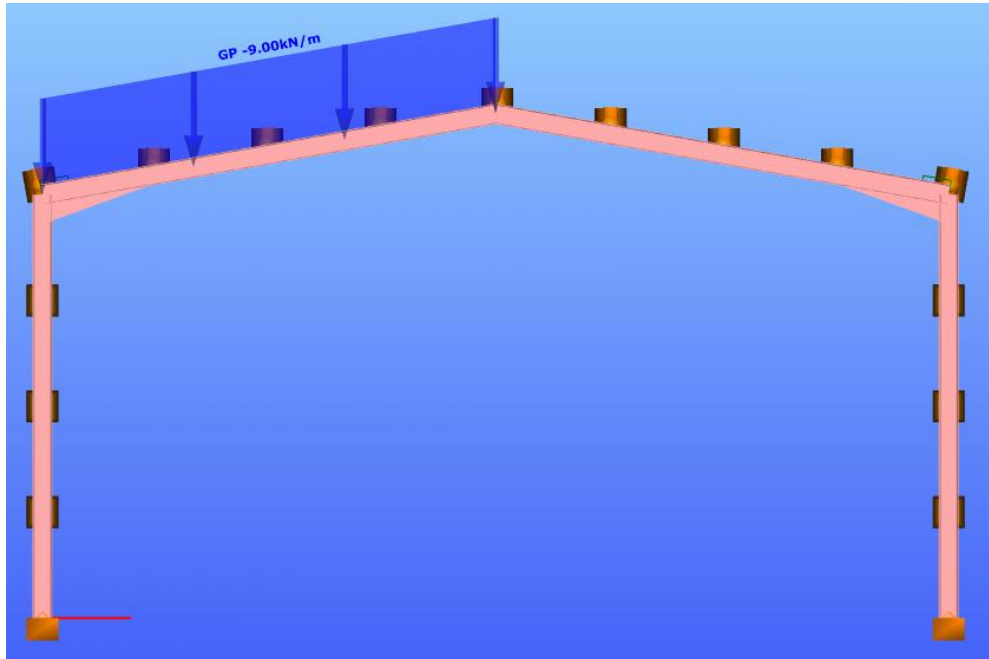


6.4.2

Be sure to turn on the **GLOBAL PROJECTION SYSTEM** setting in the window while defining the snow load.



Half snow load should look like as below.



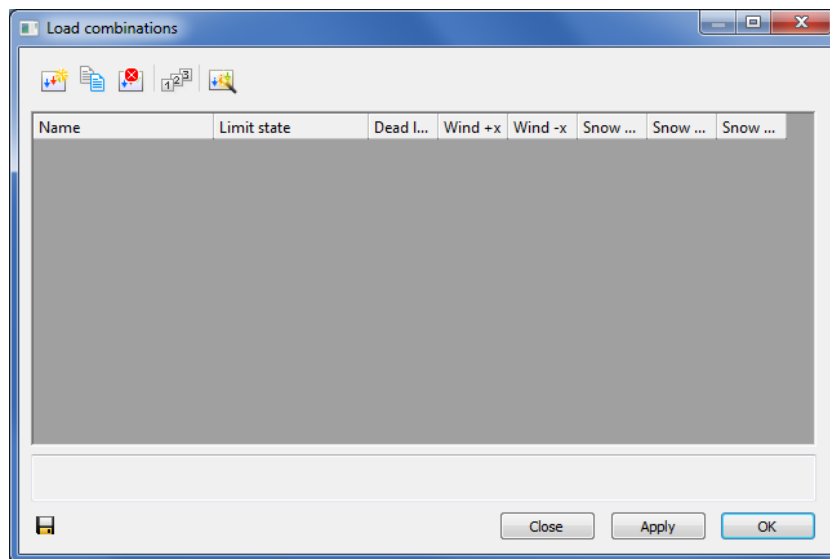
3. LOAD COMBINATION

After all the load cases have been defined take the next step and create the load combinations.

According to the actual structural design code load combinations must be set from load cases. Load combinations can be created manually by giving the safety and combination factors manually for each load case or can be created automatically by using the automatic load combination generation function of ConSteel.

Delete the default load combination first by selecting it and click on the delete icon. Afterwards click on the last icon to generate load combinations automatically.

6.3



To generate the load combinations another set need to be made: the type of the load combinations. The settings were implemented from the EuroCode standard.

Automatic generation of load combinations

Load combinations in ultimate limit states

☒ Load combinations in persistent and transient design situations - (6.10.)

$$\sum_{j \geq 1} \gamma_{G,j} G_{k,j} + \gamma_{Q,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i}$$

☐ Load combinations in persistent and transient design situations - (6.10. a,b)

$$\begin{cases} \sum_{j \geq 1} \gamma_{G,j} G_{k,j} + \gamma_{Q,1} \psi_{0,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \\ \sum_{j \geq 1} \xi_j \gamma_{G,j} G_{k,j} + \gamma_{Q,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \end{cases}$$

☐ Load combinations in accidental design situations - (6.11. b)

$$\sum_{j \geq 1} G_{k,j} + A_d + \psi_{1,1} Q_{k,1} + \sum_{i > 1} \psi_{2,i} Q_{k,i}$$

☐ Load combinations in seismic design situation - (6.12. b)

$$\sum_{j \geq 1} G_{k,j} + A_{Ed} + \sum_{i \geq 1} \varphi \psi_{2,i} Q_{k,i}$$

Load combinations in serviceability limit states

☐ Characteristic combinations (in case of irreversible serviceability limit states) - (6.14. b)

$$\sum_{j \geq 1} G_{k,j} + Q_{k,1} + \sum_{i > 1} \psi_{0,i} Q_{k,i}$$

☐ Frequent combinations (in case of reversible limit states, eg. dislocation, crack) - (6.15. b)

$$\sum_{j \geq 1} G_{k,j} + \psi_{1,1} Q_{k,1} + \sum_{i > 1} \psi_{2,i} Q_{k,i}$$

☐ Quasi permanent combinations (e.g. strains, crack width of concrete) - (6.16. b)

$$\sum_{j \geq 1} G_{k,j} + \sum_{i \geq 1} \psi_{2,i} Q_{k,i}$$

Cancel Apply

After clicking on the **APPLY** button all the possible load combinations are generated. Click on **OK** to proceed.

Load combinations

Name	Limit state	Dead l...	Wind +x	Wind -x	Snow ...	Snow ...	Snow ...
Load combination-1	(ULS) Ultimate	1,35	1,5	0	0	0	0
Load combination-2	(ULS) Ultimate	1	1,5	0	0	0	0
Load combination-3	(ULS) Ultimate	1,35	1,5	0	0,75	0	0
Load combination-4	(ULS) Ultimate	1	1,5	0	0,75	0	0
Load combination-5	(ULS) Ultimate	1,35	1,5	0	0	0,75	0
Load combination-6	(ULS) Ultimate	1	1,5	0	0	0,75	0
Load combination-7	(ULS) Ultimate	1,35	1,5	0	0	0	0,75
Load combination-8	(ULS) Ultimate	1	1,5	0	0	0	0,75
Load combination-9	(ULS) Ultimate	1,35	0	1,5	0	0	0
Load combination-10	(ULS) Ultimate	1	0	1,5	0	0	0
Load combination-11	(ULS) Ultimate	1,35	0	1,5	0,75	0	0

Close Apply OK

It is possible to edit multiple safety factors at a time: select safety factors and press right click over it.

The yellow rows in the load combination table are the automatically generated combinations. The white ones are manually edited.

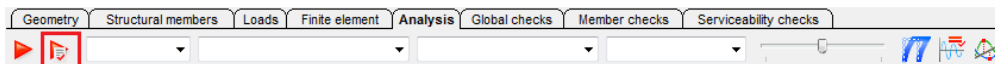
Huge number of load combinations needs long period of time to calculate in the analysis. So it is good to keep the number of the load combinations to the minimum. If the automatic load combination feature is used in ConSteel it is advisable to delete those rows which cannot be relevant. To do so select the load combination (multiple LC-s can be selected with using CTRL or SHIFT buttons) and click on the Delete Load Combination icon.

It is important to set the limit state correctly because design checks will be made to Resistance load combinations and also serviceability check are performed for SLS combinations.

4. ANALYSIS

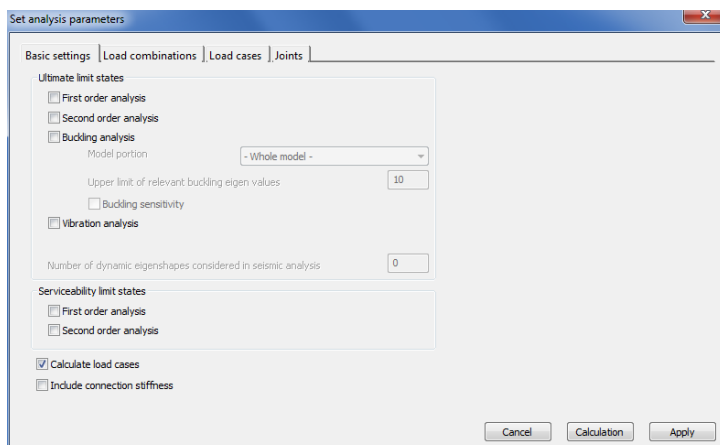
It is strongly recommended to perform only first order analysis for the first time after modeling. In order to do so, click on the **ANALYSIS PARAMETERS** icon.

7.2

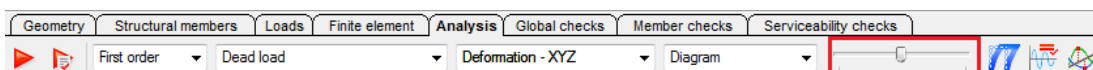


Set 'Calculate load cases' and click on the **CALCULATION** button.

7.4.1



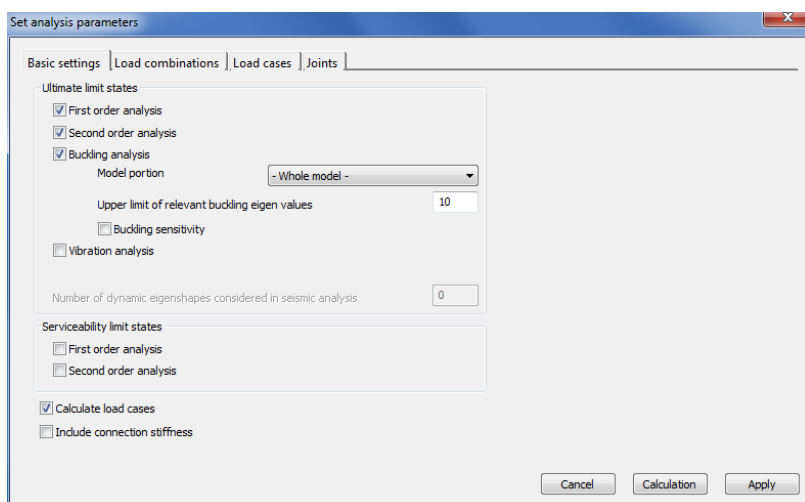
The default result view is the deformation shape. The scale of the deformation shape can be changed by moving the slider. Check the deformation of the model and be sure that all the supports are placed in the right position.



If everything is OK (so there is no modeling errors) set first order, second order and buckling analysis and rerun the calculation.

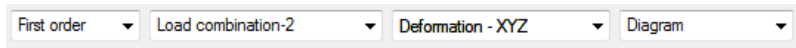
7.4.2

7.4.3

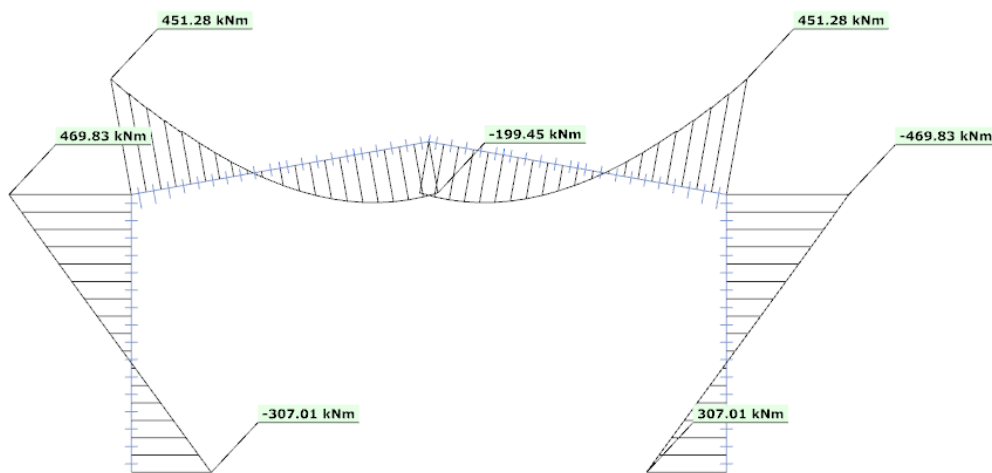


The four combos contain the following selection options (from left to right):

- ▶ selection of analysis type
- ▶ selection of load combination or load case
- ▶ selection of result type
- ▶ selection of type of view



All the model view options are applicable on the result graphics. Moving the slider beside the combos the scaling of the results can be adjusted on the graphics. In case of partial or submodel view the graphics and the tables show only the results of the actual model part.



Three types of result tables can be viewed:



Extreme values by members: select the maximum and minimum values for each structural members



User defined values: show the values belonging to the markers defined by the user



All values

The result tables arrange the values according to the current result type:

- ▶ displacement type results (first or second order deformations, static or dynamic eigenshapes): finite element node number, displacement components

- ▶ force type results (internal forces, forces or stresses on surface): finite element node number, finite element number, force components
- ▶ reaction type results (reactions, equilibrium): finite element node number, reaction components.

On the graphical interpretation of the results markers can be placed in order to show certain significant values. When moving the mouse along the structural elements (or more correctly the finite elements) the result markers continuously appear showing the actual values. These markers can be fixed by clicking on the right mouse button and choosing the **MARKER** option.

When there are fixed markers on the model the **USER DEFINED VALUES** table contains the appropriate (view dependant) values of the marked points. In this table the markers can be switched to disabled by uncheck the proper row.

<input type="checkbox"/>	5	37	-60,130	0,000	4,900	0,000	-56,820	0,000	0,000
<input checked="" type="checkbox"/>	4	56	-71,400	0,000	52,980	0,000	186,740	0,000	0,000
<input checked="" type="checkbox"/>	3	9	-75,880	0,000	-78,680	0,000	221,200	0,000	0,000

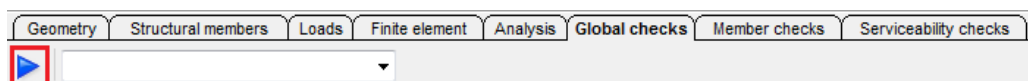
The markers can be deleted by:

- ▶ click on the proper row in the table by the right mouse button
- ▶ click on the marked point on the graphics by the right mouse button and chose 'Remove marker'

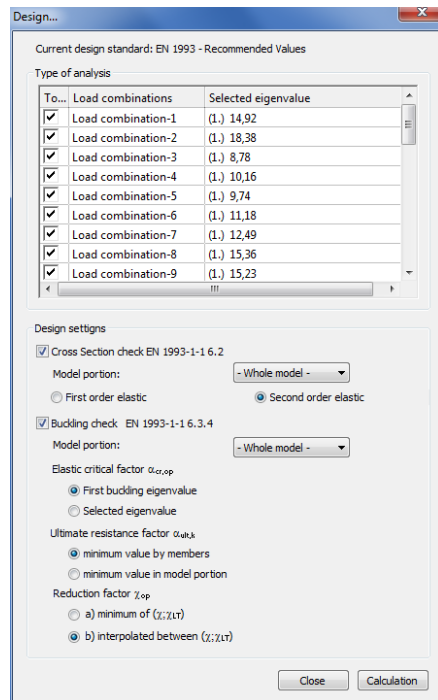
Extreme values can automatically marked by the check boxes on the 'Extreme values' table rows.

5. STANDARD DESIGN

After learning how to use the analysis in ConSteel and check the results move to the standard design. Standard design related function can be found in the three following tabs: Global checks, Member checks and Serviceability checks. In order to run the global check, click on the blue arrow icon.

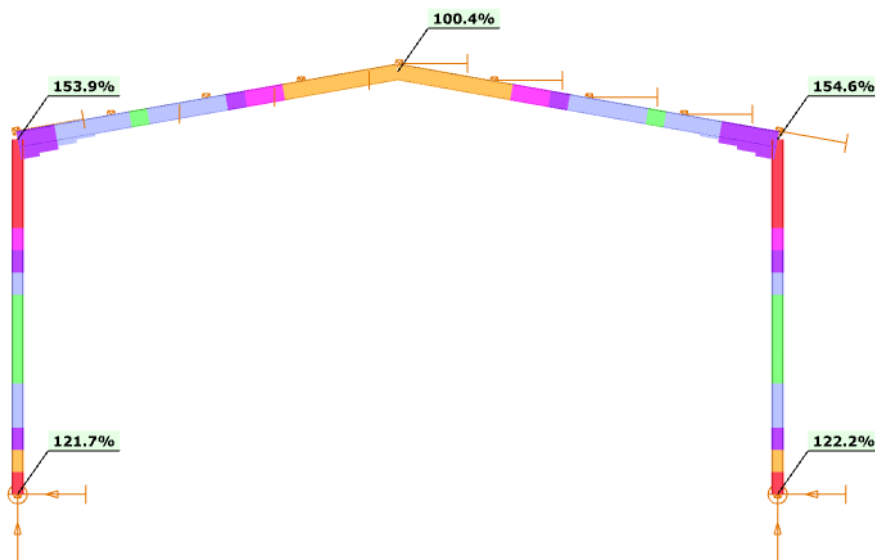


The dialog window as shown below appears and the following settings can be set: type of analysis, load combination, type of design (section resistance or section and stability analysis) and the critical load considered in stability design (first or selected eigenvalue). Click on **GLOBAL RESITANCE** to proceed.



As you can see in the colored image the some results are more than 100% so bigger sections are needed. If the cursor is moved over the model the result is shown. Press right click and select **MARKER** to place a flag.

7.5.3



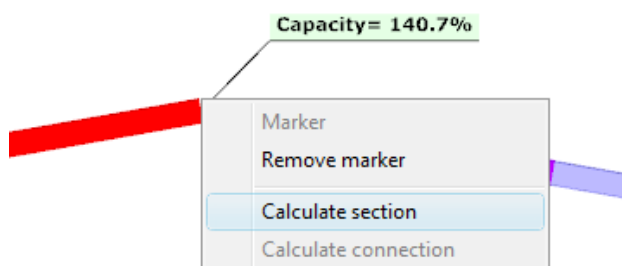
Similar to the analysis results design results can be also checked in sheets at the bottom part of the screen.

element	Joint	Load combination	Examination	Capacity [%]	
32	k	Load combination-19	Global stability resistance	198,2	
16	k	Load combination-21	Global stability resistance	198,0	Calculate section
31	k	Load combination-19	Global stability resistance	180,5	
32	j	Load combination-19	Global stability resistance	180,5	
16	j	Load combination-21	Global stability resistance	180,2	

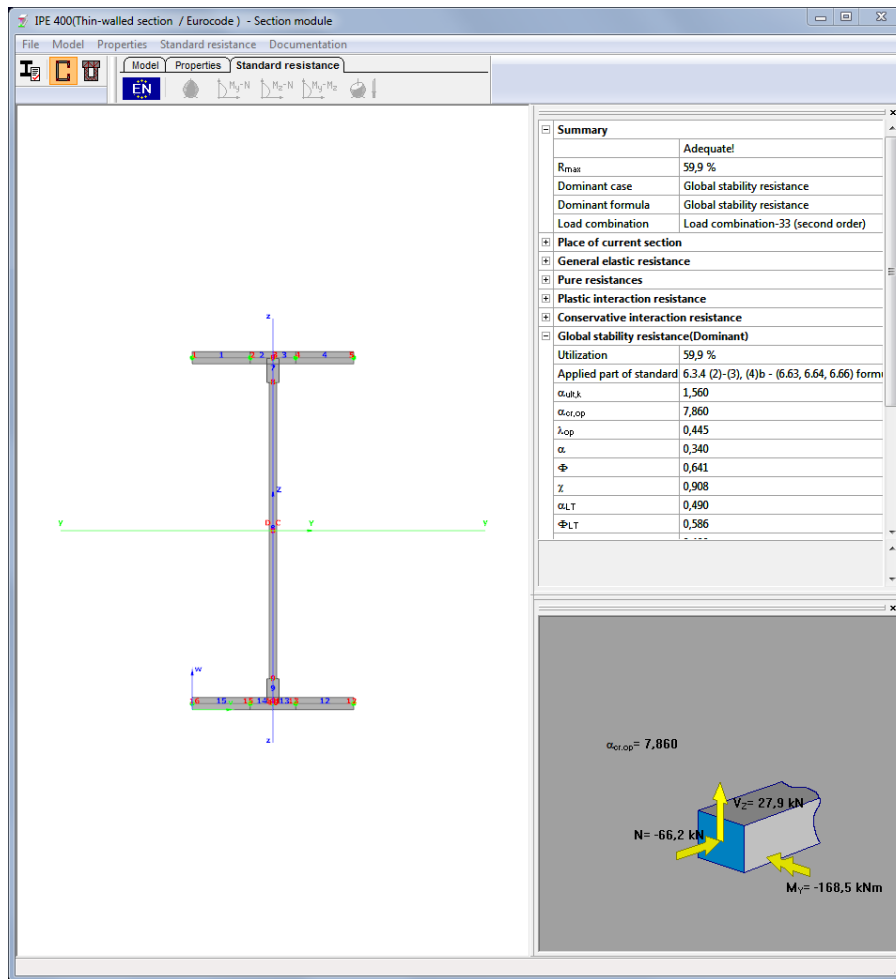
The wide functionality of the ConSteel connected with cross-sections suggested detaching a separate module: Section Module (**CSSECTION**). This module collects all the tools relate to the creation, property calculation, stress analysis and standard design of a cross-section. Two parallel modeling possibilities exist: the *General Solid Section (GSS)* is an accurate finite element based surface model for any kind of cross-sections, the *Elastic Plate Segment (EPS)* is a thin-walled model especially for steel profiles. The steel standard design formulas are easy to evaluate on the *EPS* model including the real effective section generation of class 4 cross-sections based on the sectional forces resulted from the structural analysis.

6. SECTION MODULE – CSSECTION

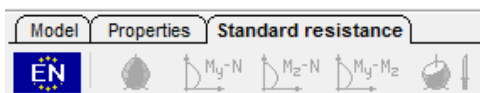
CSSECTION can be launched from the model if the **STANDARD RESISTANCE** tab is chosen and standard design had been run. There are two ways to open **CSSECTION** directly for a selected section. First is to press right click on **STANDARD RESISTANCE** tab over the section which needs to be analyzed and select **CALCULATE SECTION**.



The second way is to press right click over the element in the result sheet which can be found in the bottom of the screen on the **STANDARD RESISTANCE** tab and click on **CALCULATE SECTION**.



Detailed design calculation results can be checked at the right part of the window. The standard design procedures performed on the *EPS* model cover the major part of steel design methods in the EuroCode (the whole EC 3 1-1 and parts of EC 3 1-5) including the calculations of general elastic resistance, examination of pure cases, plastic and conservative interaction resistances of certain section types, web buckling analysis.




For a given loading case the module determines the necessary and possible checks, performs all the standard calculations and evaluates the dominant case.

There is a short summary about the calculations on the right and also the name of the dominant case, formula and load combination can be found there.

If the section module is opened from the model, then all the load combination from the model are automatically imported into the section model. The relevant is chosen but any other can be selected and checked. It is also possible to add load combinations manually by clicking on the New Load Combination icon.

Lcomb.	Current	N _{Ed} [kN]	M _{y,Ed} [kNm]	M _{z,Ed} [kNm]	V _{y,Ed} [kN]	V _{z,Ed} [kN]	B _{Ed} [kNm ²]	T _{Ed} [kNm]	T _{w,Ed} [kNm]	T _{Ed} [kNm]	B _{or.op} [-]	B _{ult} [-]
Load Combination 1 (first order)	<input type="checkbox"/>	0,0	-162,3	0,0	-0,0	93,8	-0,1	-0,1	0,1	0,0	1,370	0,937
Load Combination 1 (second order)	<input checked="" type="checkbox"/>	-0,5	-162,3	-0,0	-0,2	93,8	-0,1	-0,0	0,0	0,0	1,370	0,937

Stresses can be checked by selecting the **PROPERTIES** tab and clicking on the

STRESSES icon . Stresses can be viewed both in the EPS and the GSS model.







10.4.2

The following stresses are calculated by **CSSECTION**:

Stress Symbol	Stress name
σ_N	Normal stress due to axial force
σ_{My}	Normal stress due strong axis bending
σ_{Mz}	Normal stress due to weak axis bending
σ_B	Normal stress due to bimoment
$\Sigma \sigma$	Resultant normal stress
τ_{Vy}	Shear stress due to strong axis shear force
τ_{Vz}	Shear stress due to weak axis shear force
$\Sigma \tau_V$	Resultant shear stress due to shear forces
$\tau_{Tsv,y}$	Strong axis shear stress due to St. Venant torsion
$\tau_{Tsv,z}$	Weak axis shear stress due to St. Venant torsion
$\Sigma \tau_{Tsv}$	Resultant shear stress due to St. Venant torsion
$\tau_{Tw,y}$	Strong axis shear stress due to warping torsion
$\tau_{Tw,z}$	Weak axis shear stress due to warping torsion
$\Sigma \tau_{Tw}$	Resultant shear stress due to warping torsion
$\Sigma \tau_T$	Resultant shear stress due to torsion
$\Sigma \tau_y$	Strong axis resultant shear stress
$\Sigma \tau_z$	Weak axis resultant shear stress

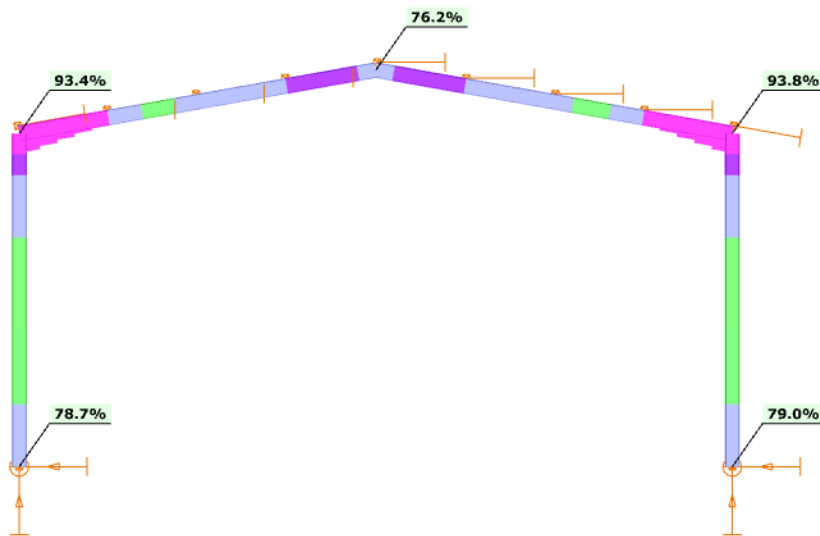
MODIFYING THE SECTION SIZES

The section sizes must be altered. Change the column section size to HEA 400 and the beams to IPE 400. In order to do so click on **STRUCTURAL MEMBERS** tab. Select the columns and change section sizes to HEA 400 in the property sheet at the lower right of the screen.

+ Line (2)		
- Bar member (2)		
Name	different	
Section properties	HEA 400	
Section	HEA 400	
Release start point	Continuous	
Release end point	Continuous	
Eccentricity - y [mm]	0	
Eccentricity - z [mm]	0	
Rotate [deg]	0	
Number of finite elements	Automatic	
Bow imperf. L/v	0	
Bow imperf. L/w	0	
Element type	Beam-column with warping	
Element group	different	
Layer	Layer_1	

The method is the same for the beams.

The next step is to rerun the analysis and design.



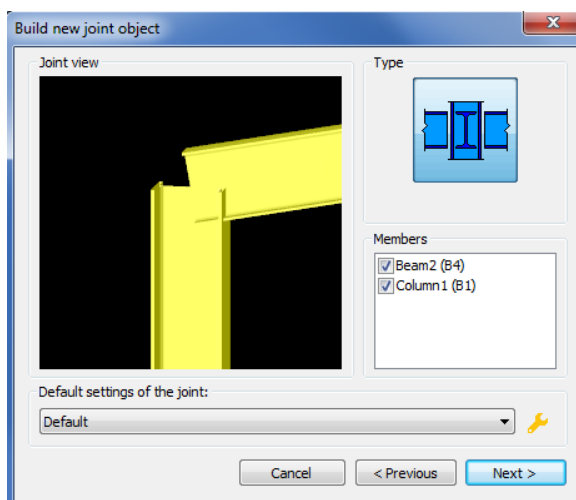
The capacities of the sections are adequate now: all of them are less than 100%.

7. JOINT MODULE - CSJOINT

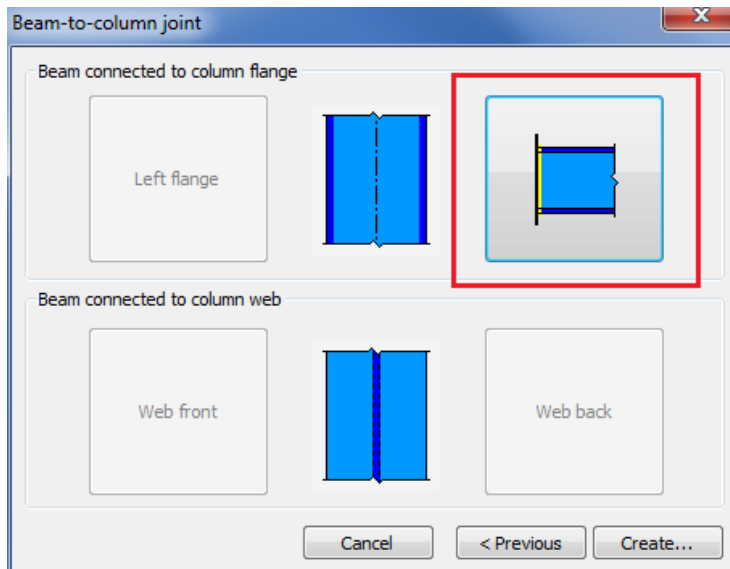
All the joint related commands can be found on the **STRUCTURAL MEMBERS** tab. These are the followings: **EDIT JOINT**, **CREATE JOINT BY MODEL**, **PLACE JOINT**.



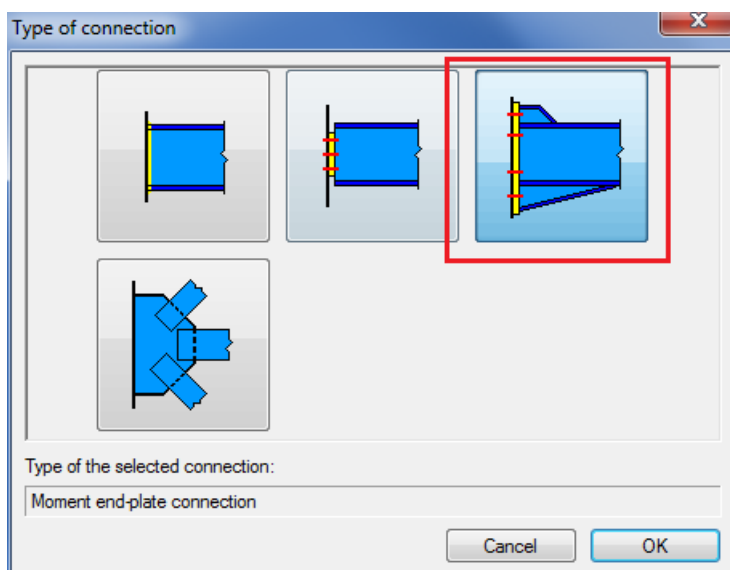
To create joint by model, click on the **CREATE JOINT BY MODEL** icon and select beam to column joint.



Click **NEXT**. Select the Joint image.

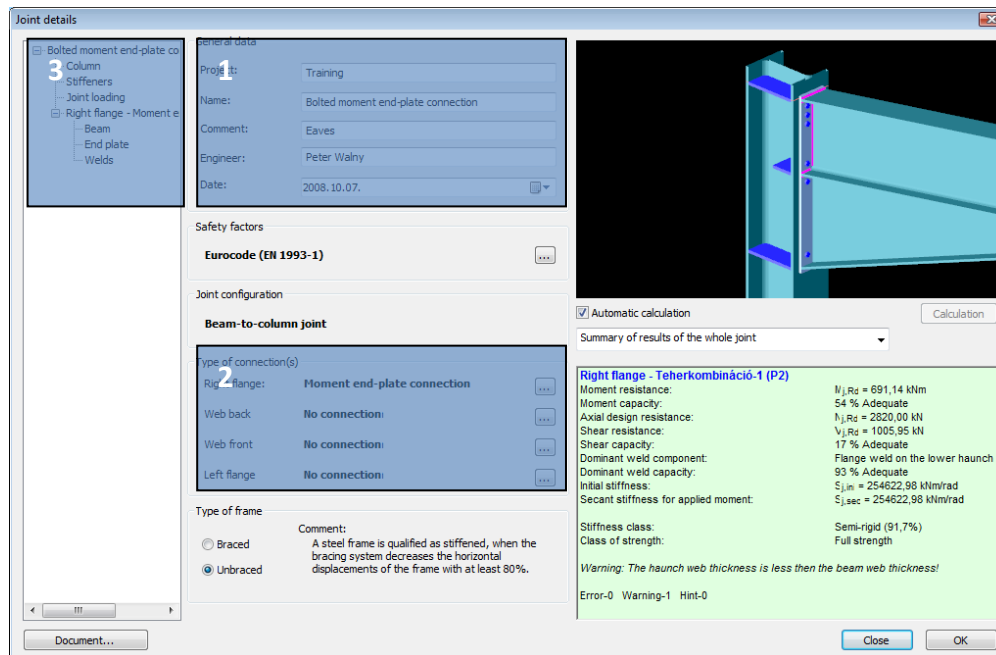


Select the connection type: Moment end plate connection. Click **OK** and **CREATE**.

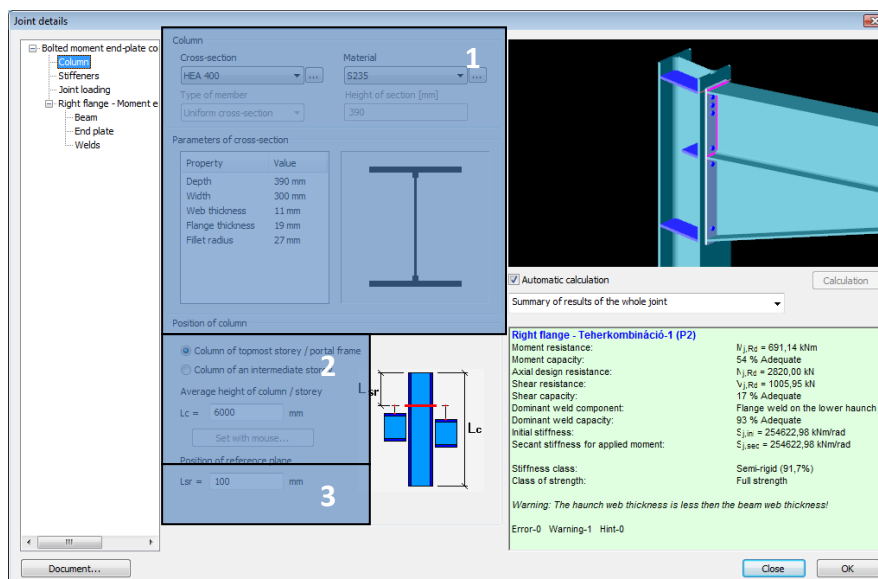


Joint details appear in a new window where numerous settings can be made. In the following pages we would like to give you an overview about the Joint module (**CSJOINT**) using the beam to column connection as an example.

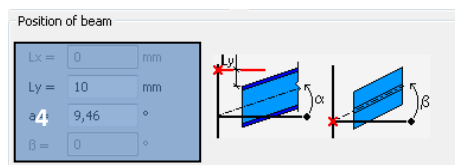
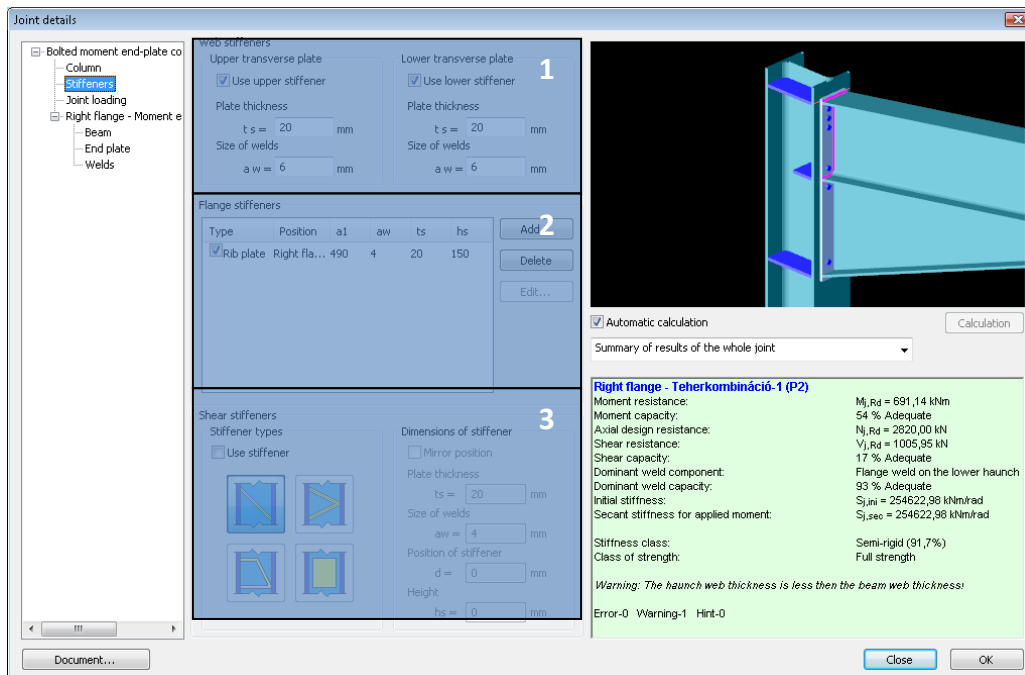
The #1 part in the image below contains general information about the joint: Joint name, project, engineer, date, etc. These properties will appear in the joint documentation and also useful to distinguish joints from each other. No. 2 part in the image below shows the general type of the connection. Additional connections can be added or removed if necessary. No. 3 part shows a tree structure about the joint: all components of the connection can be seen and selected here.



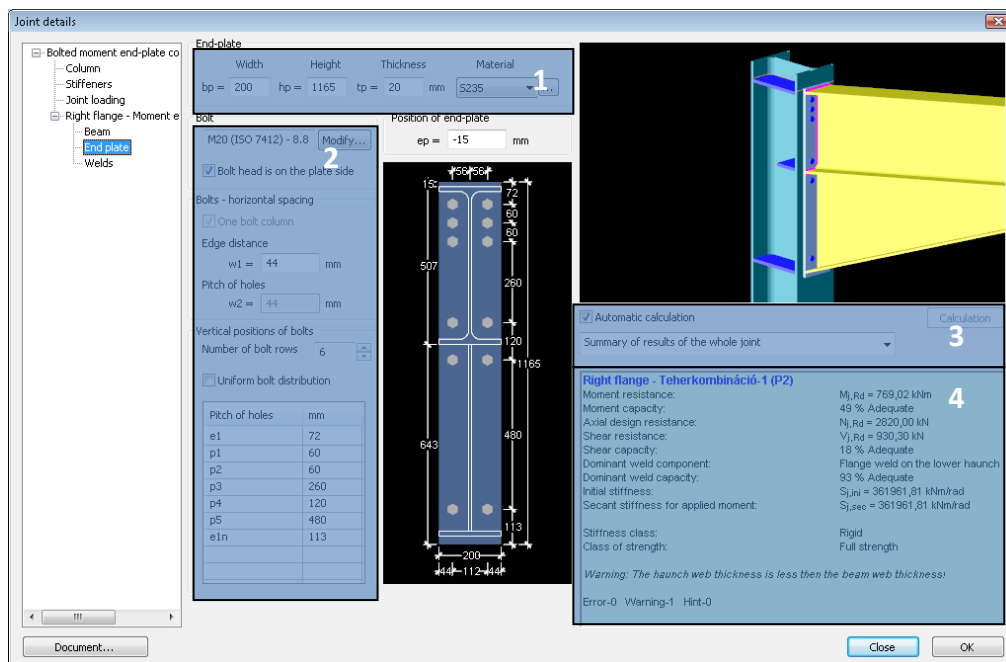
If the column is selected then column related properties of the connection can be set like the section size and material grade (#1). Setting the position of the column is also very important (#2). Furthermore the position of the reference plane is essential for the joint calculation. The image on the right gives a visual help about the meaning of 'Lsr' and 'Lc' variables.



Stiffeners can be added if '**STIFFENERS**' is clicked in the connection tree. Upper and lower stiffener plate properties can be seen in the #1 part of the image below. Additional web and flange stiffeners can also be put in (#2). In order to place shear stiffener in the joint click on the #3 part. Bear in mind that the position of the stiffeners are measured from the reference plane of the beam, which can be set if the 'Beam' is clicked in the joint tree (see #4 image).



End plate properties can be edited in the #1 highlighted part of the image below. Bolt properties can be seen on #2 part. Any changes made on the joint will take place either when pressing Enter on the keyboard or by clicking to another editable field on the window. If 'Automatic calculation' is turned on and any changes take place then the calculation of the joint is automatically rerun. It can be turned off. In that case click on the '**CALCULATION**' button to run the calculation (#3). In the pull down menu of the #3 highlighted part the type of the result visualisation can be set to three different types: Summary for the whole joint, Summary for the selected connection, Detailed results of the selected connection. In the last two cases a connection must be chosen in the connection tree.




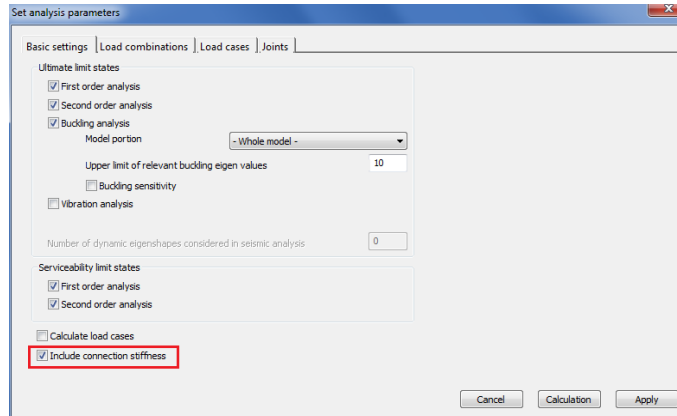
The result of the joint calculation can be seen in the #4 highlighted part of the window. If the detailed result is selected then all the results of the necessary calculations can be seen according to the selected load combination or according to the dominant load case.

After creating the joint it is advisable to place it in the global model. To do so, use the **PLACE JOINT** function of ConSteel. One joint can be placed to several places in the model where the same geometry exists. After the joint is placed ConSteel automatically import the loads from the global model.

7.2. STRUCTURE-JOINT INTERACTION

The most up-to-date structural design procedures take into account the mechanical interaction between the global structural model and its connections (rigid, semi-rigid or pinned) which generally makes the results more economic and realistic. However this approach requires a more complicated relationship between the joints and the structure and accordingly more serious modeling effort from the engineer. In the ConSteel all the joint types can be defined freely or based on the global model geometry using the automatic identification tool, which examines the position of the connected members and the proper cross-sections and offer the possible joint types. After defining the joint it can place back to the global model and the appropriate connection stiffness can be automatically used in the global analysis, and a placed joint is always rechecked based on the current analysis results.

In order to place a joint to the global model click on the **PLACE JOINT** icon.  One joint can be placed to several places in the model if these meet the geometrical requirements. The stiffness of the connection can also be taking into account during the analysis if desired. In order to do so click on the analysis parameters and put a tick to **APPLY CONNECTION STIFFNESS**. Rerun the analysis to the changes take place.

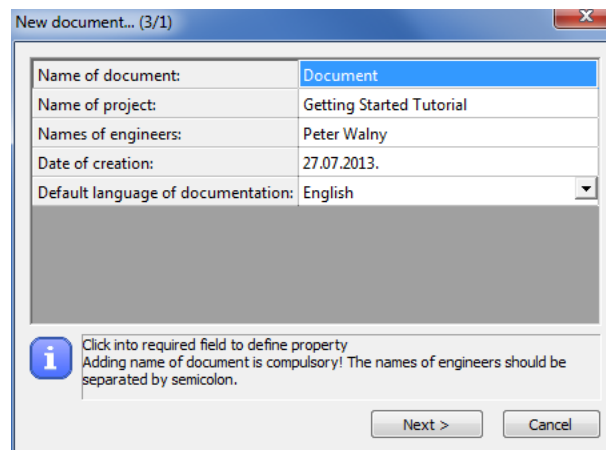


8. DOCUMENTATION

All icons which related to the documentation are placed in the documentation tab. The documentation can be generated if first icon is clicked. The documentation wizard guides you through the documentation generating process.

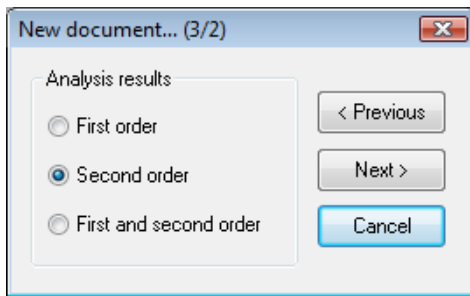


On the first window the name of the documentation can be set. The name appears on every page of the documentation. Other fields like the name of the project, the name of the engineers and the date of the creation can also be set but would appear only on the front page.

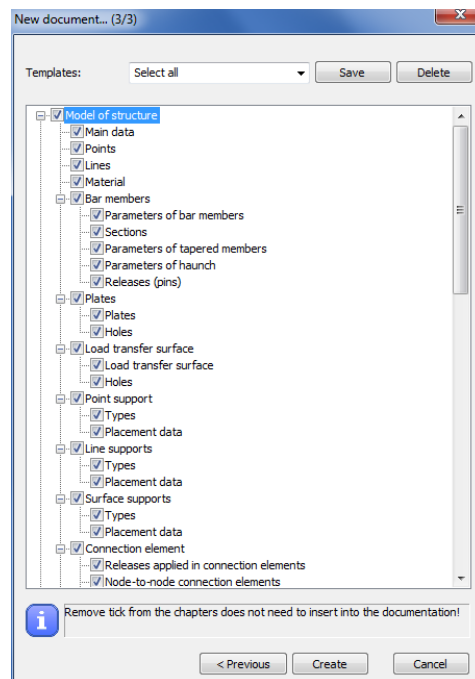


9.

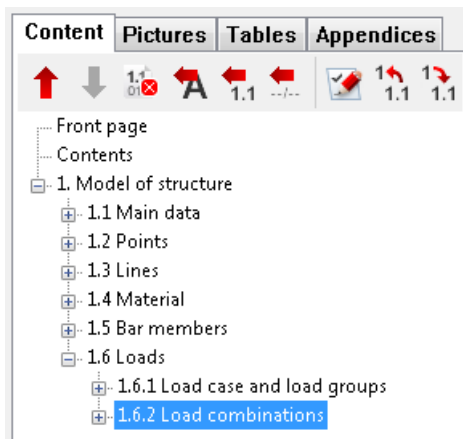
9.2.1.



The second step is to choose which of the analysis results need to be included in the documentation. Any of the analysis results can be chosen if had been calculated previously.



The third window of the wizard includes the name of the chapters which can be added to or can be removed from the documentation. If you click on the Create button the program will generate the documentation.




After creating the documentation it will be opened in a new window. At the top right of the window the chapters can be seen. The chapters can be moved up or down using the arrow icons if a chapter is selected. It is also possible to delete content from document by clicking the third icon.

Text can be inserted anywhere in the document if you click on the fourth icon and type in the desired text into the window. It is also possible to insert headings by clicking on the fifth icon.

You can also insert page break anywhere in the document using the sixth icon. The last three icons can be used for the following purposes: Edit heading and increase or decrease level of heading.

The last two icons on the right are different if you click on a chapter or in a paragraph. If you click on a chapter then the icons are as described above. If a paragraph is clicked where there is a table the last icon can be used to delete any column from the table. If you click on a paragraph which contains text then the last two icons on the right can be used to edit the paragraph content and the alignment.

By clicking the **PICTURES** tab the image handling icons can be seen. Any of the taken snapshots can be inserted into the documentation or can be deleted. The image will be inserted below the selected paragraph.

The third tab is the **TABLES**. There are lot of possibilities in ConSteel where tables can be saved. If you see the  icon then that table is saveable. Any previously saved tables can be inserted to the documentation or can be deleted from the model database. The table will be inserted below the selected paragraph.

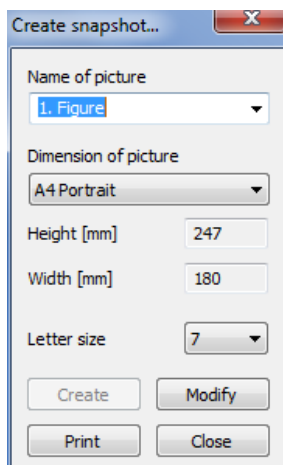
The last tab is the **Appendices**. If any joint or section documentation had been generated previously, these can be found here and can be added to the end of the documentation. The appendices can also be deleted from the database.



IT SHOULD BE BEAR IN MIND THAT THE DOCUMENTATION NEED TO BE REGENERATED TO BE UP-TO-DATE AFTER ANY CHANGES HAS BEEN MADE ON THE MODEL. IF THERE ARE SNAPSHOTS IN THE DOCUMENTATION THESE SNAPSHOTS SHOULD BE TAKEN AGAIN AND NEED TO BE ADDED TO THE DOCUMENTATION.

8.1 CREATING SNAPSHOTS

The second icon on the documentation tab can be used to take a snapshot of the model. The snapshots can be easily added to the desired part of the documentation.

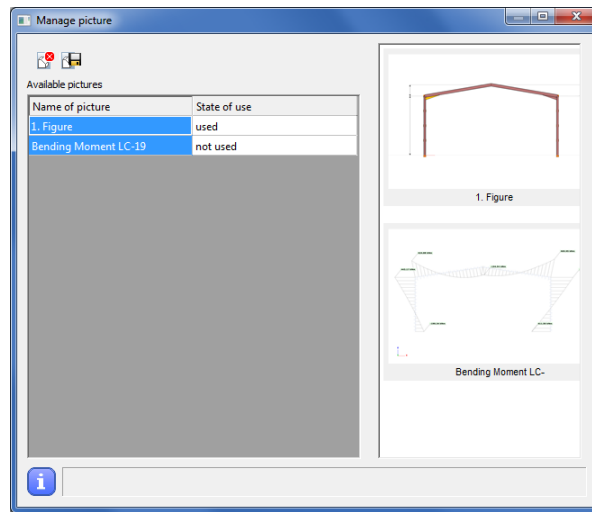


The size of the snapshot can be set and a dashed rectangle shows which part of the model would appear on the picture. The model can be moved or rotated to fit into the dashed rectangle.

8.2. SNAPSHOT MANAGER

The third icon on the documentation tab is the ***SNAPSHOT MANAGER*** where the taken snapshots can be exported to a file or can be deleted from the model in order to reduce model file size. The name of the images can also be edited by double clicking the name of the image.

9.2.3



8.3. MODEL INFORMATION

With using the fourth icon on the documentation tab the model information can be viewed or can be exported to a file. The model information window shows the most important features of the model like Number of bars, Number of supports, Total weight of the structure, etc. The mass, surface and the length can be viewed for each section in the model. The model information can be shown based on the full model or on the current view if the model view shows only a part of the full model using the sub model view.

9.2.4

