AXISVM

STEP BY STEP TUTORIAL

Edited by: Inter-CAD Kft.

Intentionally blank page

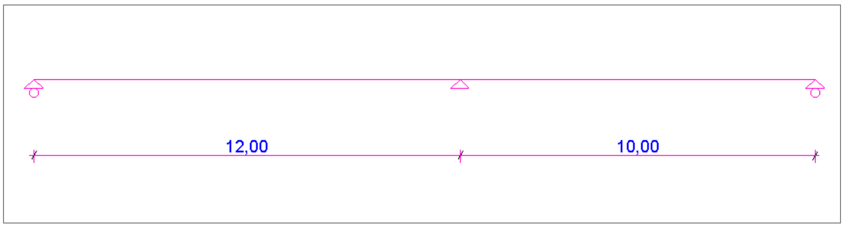
TABLE OF CONTENTS

Intentionally blank page

1. BEAM MODEL

Objective

The objective of this design example is to determine the internal forces, longitudinal reinforcement and shear links of the two-span continuous reinforced concrete beam illustrated below. The loads will be presented subsequently.

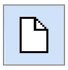


The cross-section is uniform along the beam: 400 mm*720 mm rectangle shape. The beam is analysed according to Eurocode 2 standard.

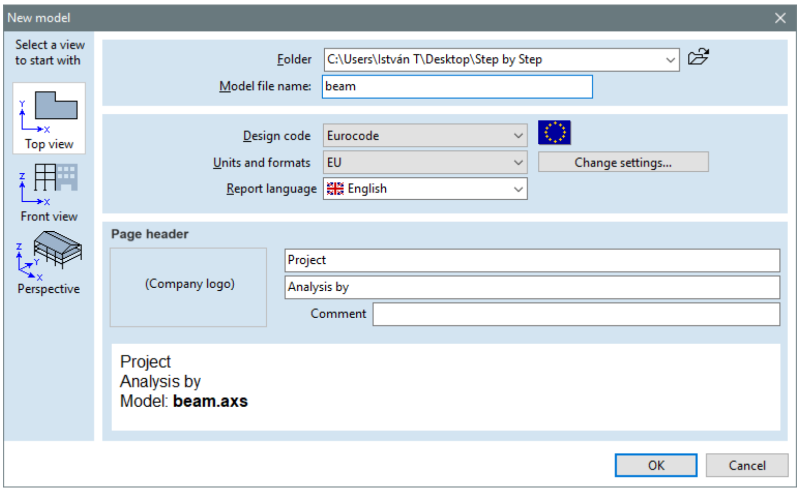
Start

Start ***AxisVMX4*** by double-clicking on ***AxisVMX4*** icon in its installation folder, found in ***Start*** – ***Programs*** menu.

New



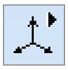
Create a new model by clicking on ***New*** icon. In the dialogue window replace ***Model file name*** with '***beam***', select ***Eurocode*** from ***Design code***s and set ***Unit and formats*** to ***EU***.



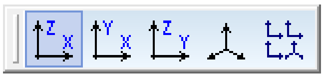
If necessary, enter the name of ***Project*** and designer (***Analysis by***) at ***Page header. Company logo*** also can be uploaded. This set page header will appear on the print image and documentation.

Click ***OK*** to close the dialog window.

Views



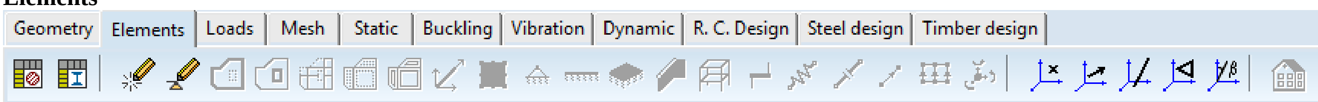
Check the view (workplane) of the model when starting a new model. On the left side of the main window find ***Views*** icon, open it with moving the cursor over the icon and select ***X-Z*** view. The actual view is presented by the global coordinate system sign at the left bottom corner of the main window.



The global coordinate system can be changed during modelling, several local coordinate systems or workplanes can be set. The directions of global coordinate system are marked with capital letters (***X***, ***Y*** or ***Z***). Please note that gravity force acts in -***Z*** direction according to the default settings, but it can be modified by the user.

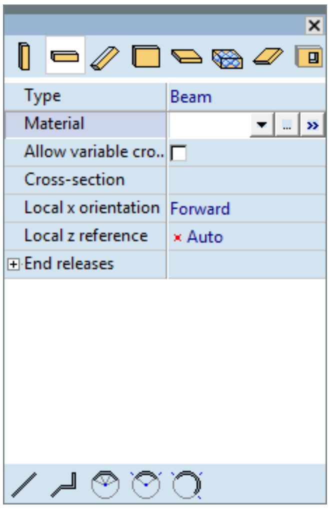
Define of geometry - Elements

Select ***Elements*** tab to define geometry and structural properties of the beam. On the tab the icons of available functions are displayed.

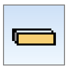


Draw objects directly

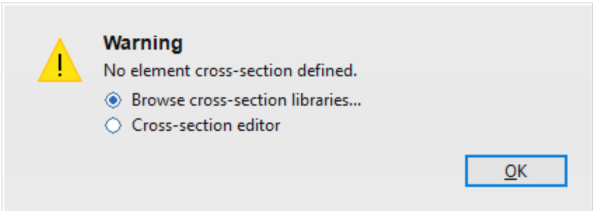
By clicking on ***Draw objects directly*** icon, brings up the following window:



Horizontal beam

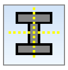


Click on ***Horizontal beam*** icon (the second one on the left among the top row icons). Choose it even if this is the active one because of the sequences of steps. The following dialogue panel shows after clicking:

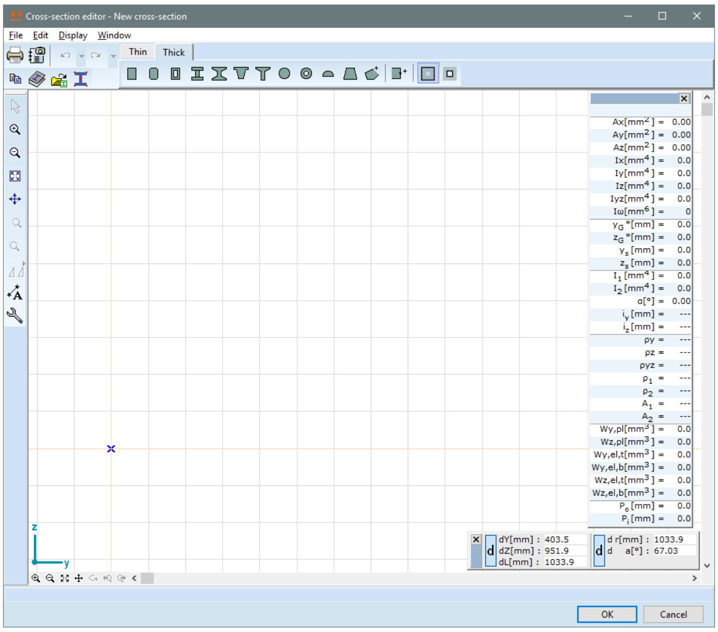


Select ***Browse cross-section libraries*** and click on ***OK***.

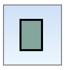
Cross-section editor



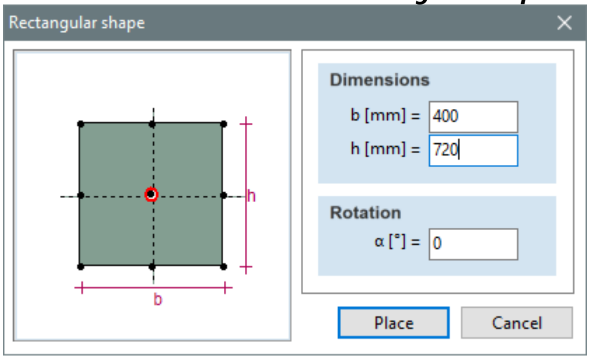
The following, ***Cross-section editor*** window shows after clicking:



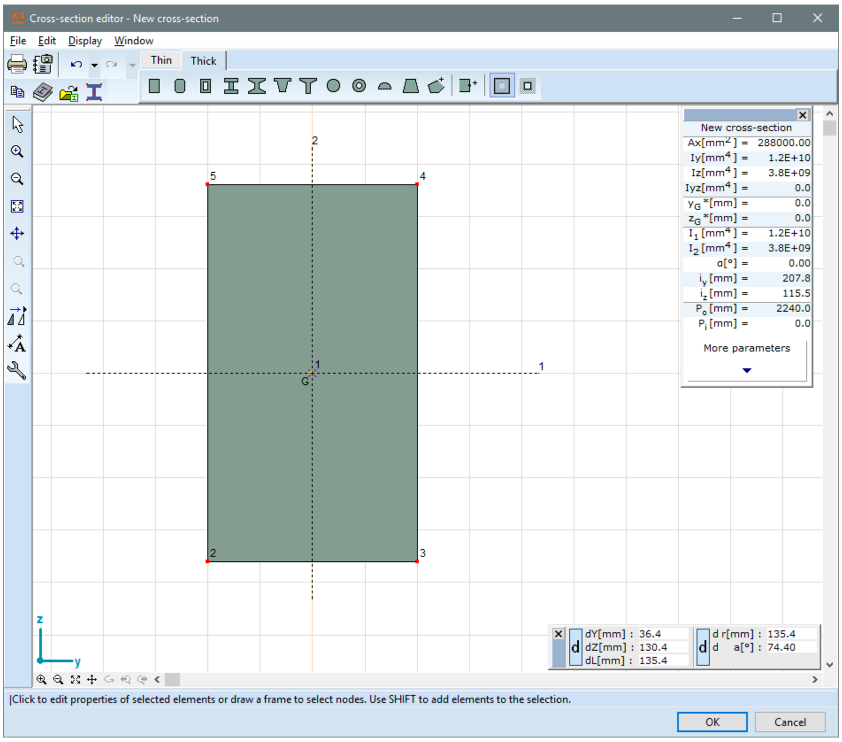
Rectangular shape



In the window find and click on ***Rectangular shape*** icon:

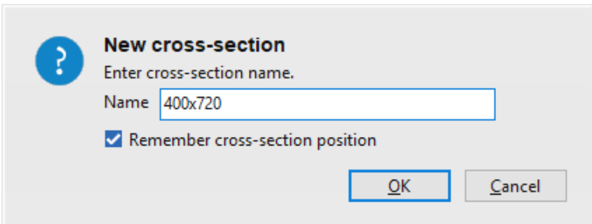


Enter ***720*** for height (***h[mm]=***) and click on ***Place*** button. (The ***b*** size of the rectangle shape is equal to the default value, so it does not have to be overwritten now.) Click anywhere in the workspace to place the cross-section. The following result can be seen:



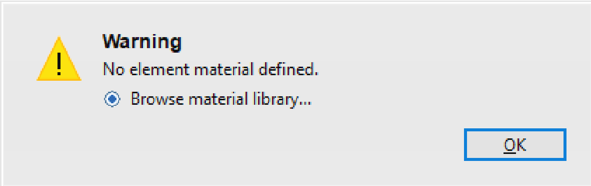
Notes***: Stress calculation points*** are automatically placed at the corners and at the centre (please see numbering 1..5). Stress is calculated only in these predefined points. These points can be modified or cleared or more points can be added. In our analysis there is no need for these points because the longitudinal reinforcement is calculated directly from internal forces.

By clicking on ***OK*** shows a dialogue panel asking for the name of the defined cross-section:

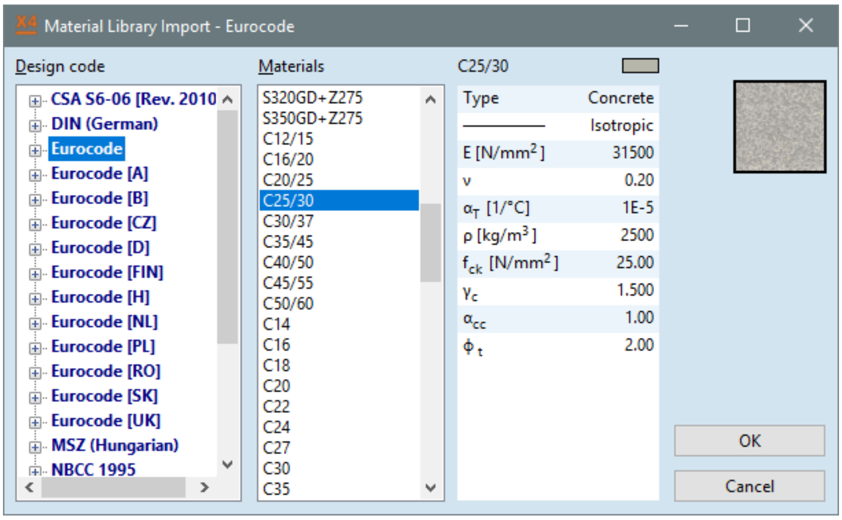


Click on ***OK*** to close the panel.

The following message shows up:



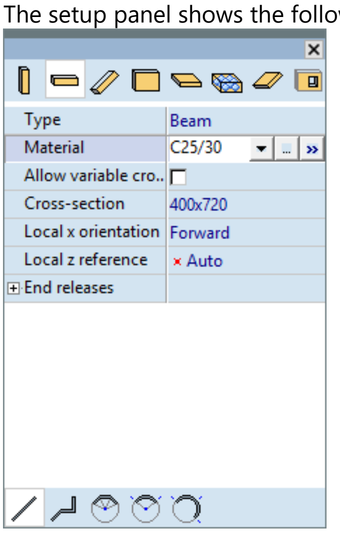
By clicking on ***OK*** shows the following window:



Roll down in the list of ***Materials*** by using the vertical sliding bar (or simply roll down the mouse wheel) and select ***C25/30***, then click on ***OK*** to close the window.

Reference

Leaving the references on auto option, the local ***x*** axis of the beam will be pointed in the direction of the beam and the local ***z*** axis will be in the vertical plane.
The setup panel shows the following:



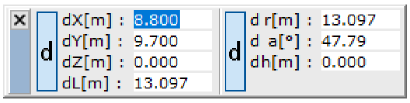
Beam polyline

On the panel, select ***Beam polyline*** function and draw the centre line of the beam.
To define the axis line, there are several options: it can be drawn graphically in the workspace or can be directly defined by specifying coordinates.

Geometry definition using coordinates:
In our example the axis of the beam is defined by drawing two lines between the three supports. The coordinates can be set in the ***Coordinates*** panel which can be found in the bottom right corner of the screen. The coordinates can be specified relative to global and relative origins. The relative origin is symbolized by a blue cross (rotated 45 degrees) in the workspace and it stands in the endpoint of the previously defined element. It can be relocated by moving the cursor to the desired point and pressing ***Insert*** button. When starting a new model, the relative origin always stands in the origin of the global coordinate system.

If ***d*** button is pressed on the ***Coordinates*** panel the values can be given relative to local origin (***dX***, ***dY***, etc…). If ***d*** button is not activated one could specify coordinates in the global system.

In our example relative coordinates are used. The local origin is now in the origin of the global system. To determine the starting point of the line (***X=0***, ***Y=0***, ***Z=0***), press ***x*** button, then the cursor jumps to the field of ***x*** coordinate on the ***Coordinates*** panel. Overwrite the highlighted actual value with ***0***.



After press ***y*** button and enter ***0***. Similarly, specify ***z*** value, finally close the input with ***Enter*** key.

Nodes in Relative coordinate system

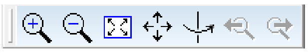
To specify the other two points, hit the buttons in the following order:

x	12	y	0	z	0	<**Enter**>
x	10	y	0	z	0	<**Enter**>,

after press mouse right button and from the quick menu select ***Cancel*** to finish the drawing process. Press ***Esc*** twice to exit from the command of object drawing (***Draw objects directly***).

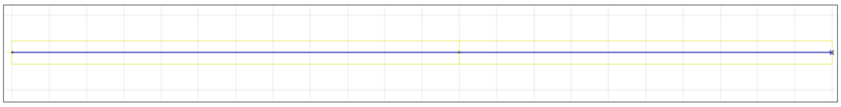
Zoom

To bring up the ***Zoom*** icon bar, move the mouse on ***Zoom*** icon (in the icon bar on the left side of the main window). It contains six icons. Select the third icon (***Zoom to fit***). This function scales the drawing of the model to fit the graphic area. This function can be activated even if pressing ***Ctrl-W*** or double clicking on mouse wheel.

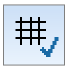


An alternative way of zooming is to press + and - on numerical keypad.

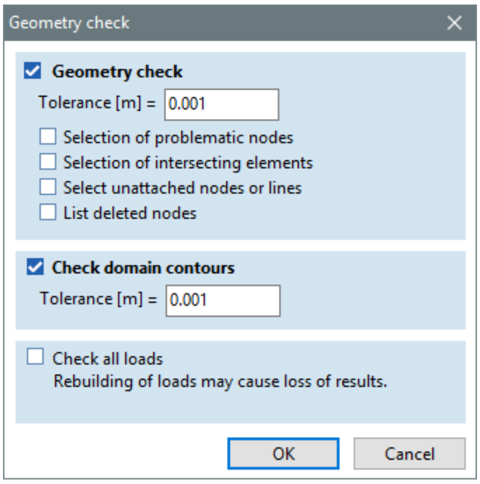
As a result, on the following screenshot the axis of the beam element (blue line) and the contour view of the cross section (yellow lines) can be seen:



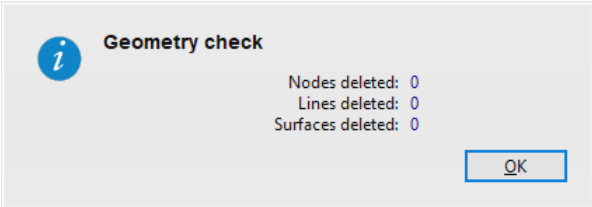
Geometry check



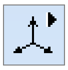
Click ***Geometry check*** icon on the ***Geometry*** tab to filter the errors in geometry. In ***Geometry check*** panel, the maximum tolerance (distance) can be specified to merge nodes.



After the geometry check a summary of actions shows:



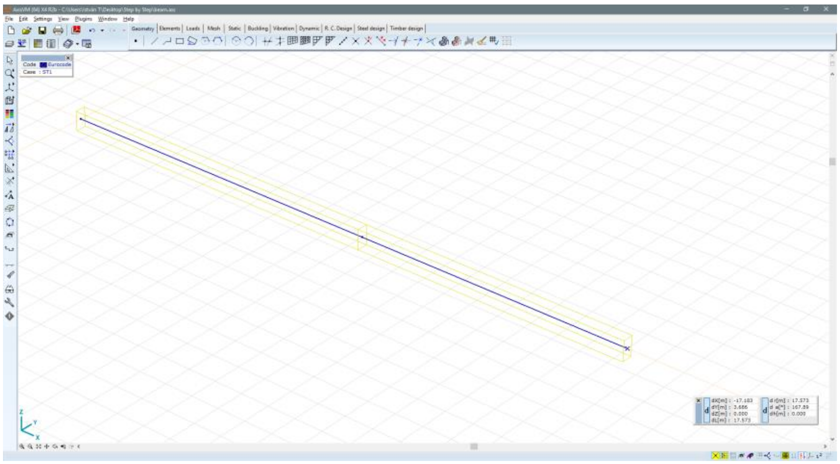
Setting perspective view



Check the perspective view of the beam. Click on ***Perspective*** icon on the left icon bar

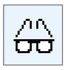
In ***Window*** menu, ***Perspective settings*** panel can be activated, if it is checked. On the panel the parameters of perspective view can be modified and the model can be rotated.

The actual view can be rotated with moving of mouse when ***Alt*** + mouse wheel pressed.



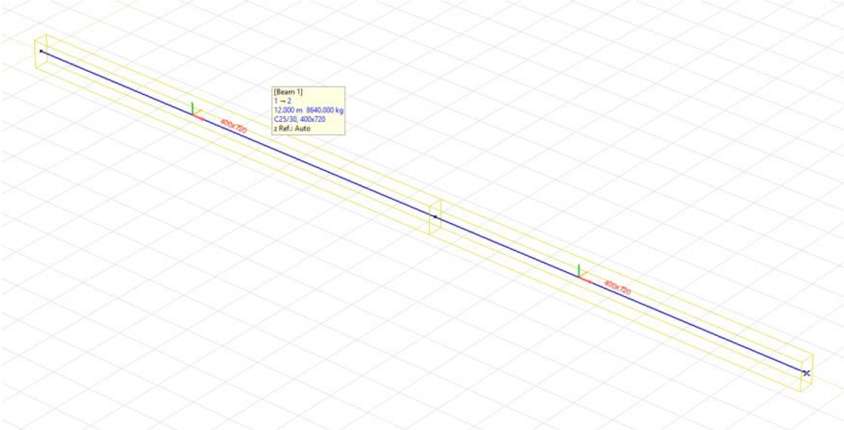
By closing the perspective panel, the view mode is preserved.

Display options



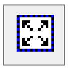
Click on ***Display Options*** icon for showing local coordinate system of the elements, node numbering, graphical symbols, etc… ***Display options*** menu can be activated more quickly by pressing right mouse button and selecting the last menu row on the panel. ***On Symbols*** tab, in ***Local Systems*** group check ***Beam*** box. On ***Labels*** tab check ***Cross-section name*** in ***Properties*** group.

By closing window with ***OK*** button, the sign of local coordinate system of the beam and the name of the cross-section is shown on the element. Moving the cursor near the reference line of an element, some properties of the element are also visible in the hint: number of the beam, number of start and end point, the length of the beam, material, the name of the cross-section and the ***z*** reference (auto or user defined)



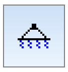
Change view from perspective to ***X-Z*** plane.

Zoom to fit

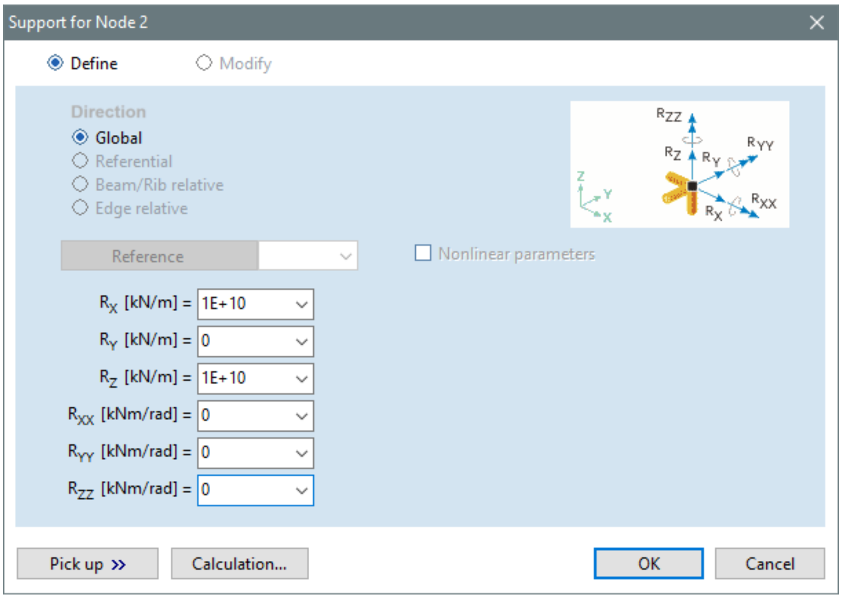


Click on ***Zoom to fit*** icon for better visibility.

Nodal support

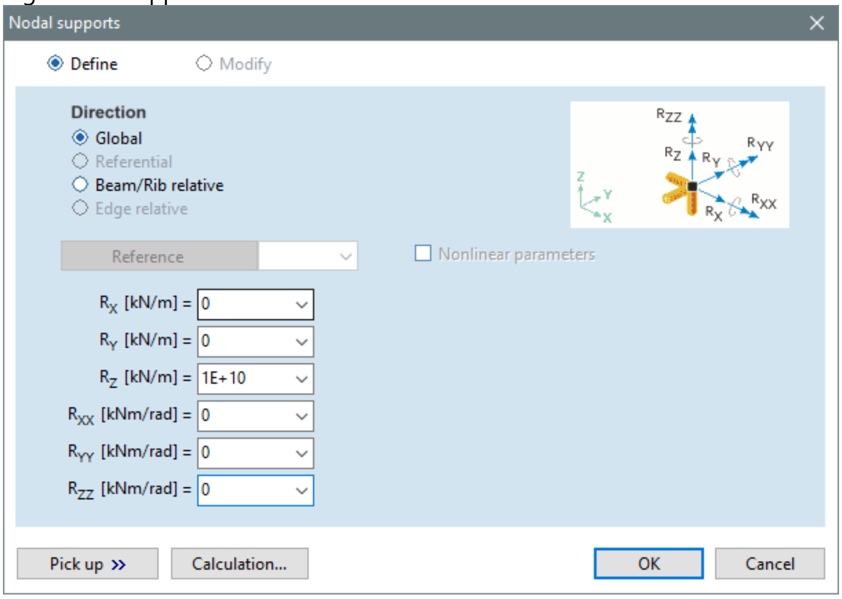


Staying on ***Elements*** tab, click on ***Nodal support*** icon and select the inner node, then click ***OK***. In the window that appears, stiffness of the support in each direction can be set in different ways: the individual components can be defined in the global system, reference direction can be adapted or can be set relative to a beam/rib or edge. Select global direction and set support stiffness. ***Rx***, ***Ry*** and ***Rz*** are the components of translation stiffness. The default stiffness value is 1E+10 [kN/m] for fixed and 0 value for free to move in a given direction.
Rxx, ***Ryy*** and ***Rzz*** are the components of rotational stiffness. The default stiffness value is 1E+10 [kNm/m] for fixed and 0 value for free to rotate around a given direction. Set rotational stiffnesses and ***Ry*** component to ***0*** and leave default values (***1E+10 [kN/m]***) for ***Rx*** and ***Rz*** components.



Click on ***OK*** to apply settings.

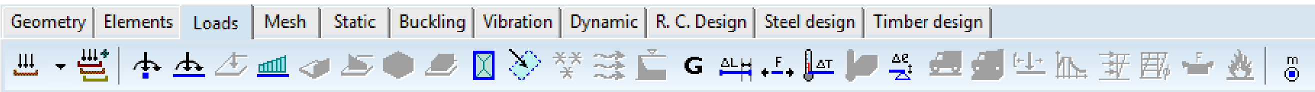
Select the other two nodes and set the stiffness components of the support as above, but create a rolling vertical support as follow:



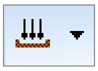
If the support conditions are set correctly, the given non-zero components of the support are indicated by brown symbol on the screen.

Loads

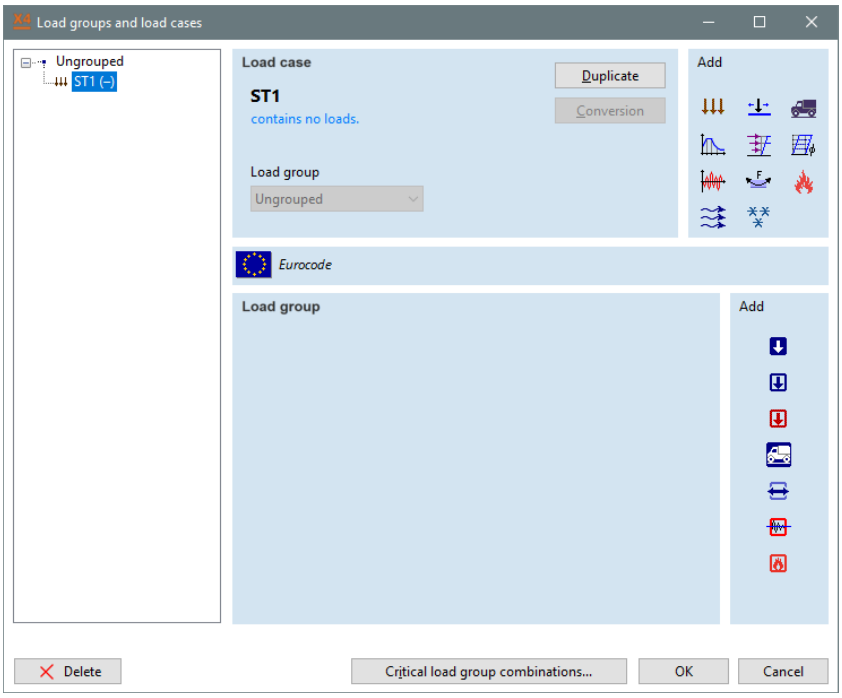
To apply loads and support movements select ***Loads*** tab.



Load cases and load groups

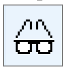


Various loads and support movements should be separated into load cases. Click on ***Load cases*** and ***load groups*** icon. The following window shows:



The software automatically generates load case ***ST1*** which can be seen in the list on the left. Click on the name of ***ST1*** and rename it to ***SELF WEIGHT***. Then click on OK button to go back to model space. The last edited load case (named ***SELF WEIGHT***) will be the active case, this can be checked on ***Status*** panel in the top left corner of the screen.

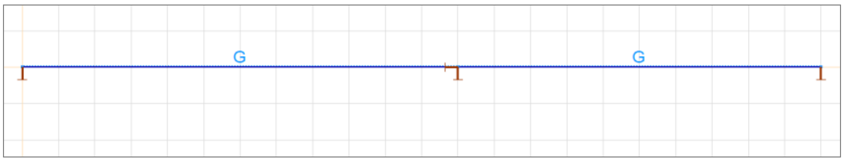
Display options



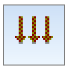
Click on ***Display options*** icon. On ***Symbols*** tab, unselect the followings: ***Object contours in 3D***, Cross-section shape in ***Graphics symbols*** group, ***Beam*** box in ***Local Systems*** group. On ***Labels*** tab, unselect ***Cross-Section Name*** in ***Properties*** group.

Self weight

Click on ***Self weight*** icon then select all elements with ***All*** (*) button. By clicking on ***OK,*** a blue dashed line can be seen above the beam element which indicates the self weight of the element. Gravitational acceleration can be set in ***Gravitation*** option (***Settings***/***Gravitation***). As a default setting gravity acceleration is 9.81 m/s^2 and acts in ***-Z*** direction.

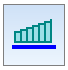


Static load case

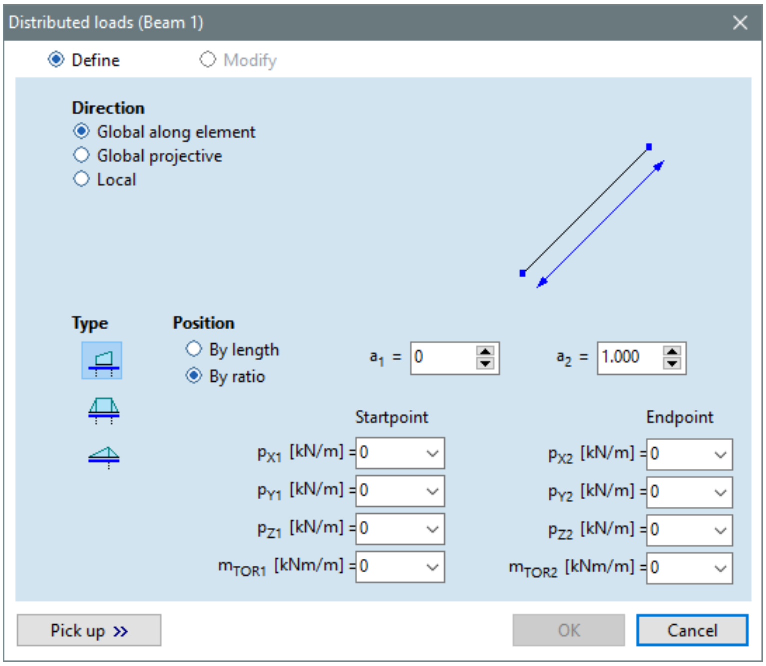


Open ***Load cases and load groups*** window again and click on ***Static*** icon in ***Add*** group. Create load cases with names ***LIVE LOAD 1***, ***LIVE LOAD 2*** and ***SUPPORT MOVEMENT***. Click on ***LIVE LOAD 1*** then close window with ***OK***.

Load along line elements

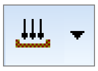


Click on ***Load along line elements*** icon and select beam on the left side then click ***OK***. The following window shows up:

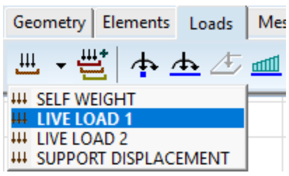


Uniformly distributed load will be defined on the beam relative to the global system. Type in ***-17.5*** into fields p_{Z1} and p_{Z2} then confirm with ***OK***. The negative value means that the load acts downward, in negative direction.

Load cases and load groups



After clicking on down arrow next to ***Load cases and load groups*** icon, the following list shows up.

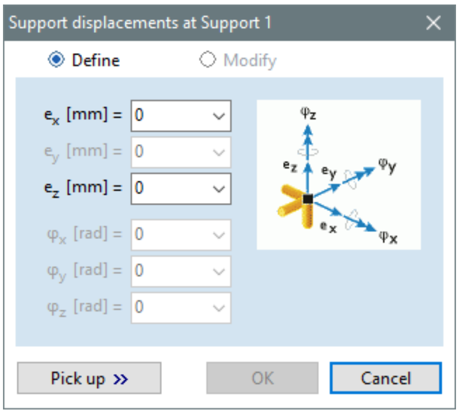


The list contains all the load cases defined and the actual one is highlighted with blue background. Select load case ***LIVE LOAD 2*** with mouse or arrow keys.

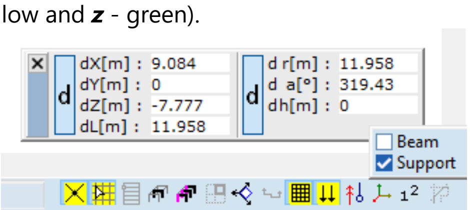
With the same method, apply ***-17,5*** kN/m (in ***Z*** direction) line load on both beams.

Support displacement

Change load case to ***SUPPORT DISPLACEMENT*** then click on ***Support displacement*** icon. Select the inner support and click ***OK***. The following shows up:

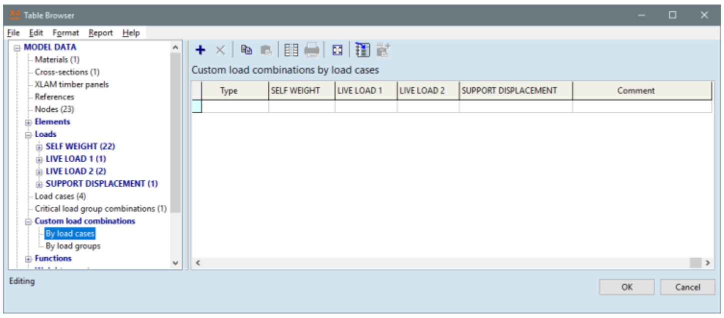


Type in value ***20*** in field $\boldsymbol{e_z}$. During data input always observe signs and directions. If necessary, switch on the local coordinate system of the support (***Display Options*** icon/ ***Symbols*** tab/ ***Local systems*** group/***Support*** checkbox). The color coordinate system show the ***x***, ***y*** and ***z*** directions (***x*** - red, ***y*** - yellow and ***z*** - green).

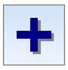


Load combinations

Activate ***Load Combinations*** by clicking on its icon, then ***Table browser*** shows up:



New row



Create new ***ULS (ultimate limit state)*** load combination by clicking on ***New row*** icon. Load factors for each load cases must be set. Set load factors as follows:

1. Load combination (***Co.#1***):

SELF WEIGHT	1.35	<Tab>
LIVE LOAD 1	1.50	<Tab>
LIVE LOAD 2	0	<Tab>
SUPPORT DISPLACEMENT	1.00	<Tab>

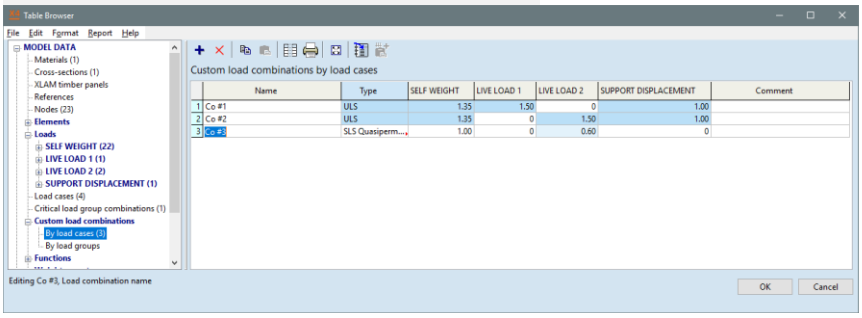
Type in all these values to cells then create another ***ULS*** load combination with name ***Co.#2***, the factors are as follows:

SELF WEIGHT	1.35	<Tab>
LIVE LOAD 1	0	<Tab>
HASZNOS 2	1.50	<Tab>
SUPPORT DISPLACEMENT	1.00	<Tab>

Create the third load combination setting ***SLS Quasipermanent*** one with the following factors:

SELF WEIGHT	1.0 0	<**Tab**>
LIVE LOAD 1	0	<**Tab**>
LIVE LOAD 2	0.6 0	<**Tab**>
SUPPORT DISPLACEMENT	0	<**Tab**>

The following table shows after data input:

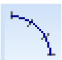


	Name	Type	SELF WEIGHT	LIVE LOAD 1	LIVE LOAD 2	SUPPORT DISPLACEMENT	Comment
1	Co #1	ULS	1.35	1.50	0	1.00	
2	Co #2	ULS	1.35	0	1.50	1.00	
3	Co #3	SLS Quasiperm...	1.00	0	0.60	0	

Remark: the cells are colored by different tone. These tones indicate differences between the specified values for better visibility. For more information please see ***User's manual***.

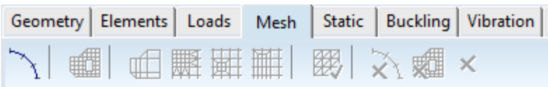
Click on ***OK*** to finish.

Mesh

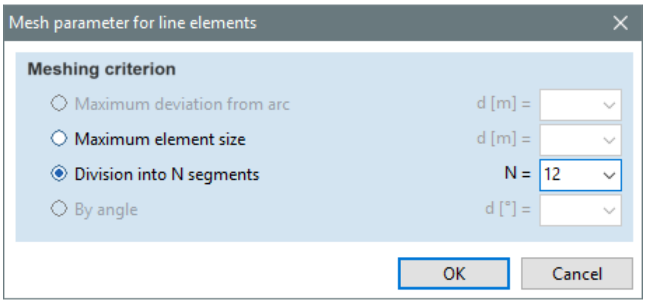


The beam should be divided into parts (line meshing). (The type and number of the longitudinal reinforcement can be adjusted to the parts only.) If necessary, the meshing of line elements can be deleted or modified later. Removing a mesh does not delete loads and properties assigned to the line element.

Click on ***Mesh*** tab and select ***Meshing of line elements*** icon, that is the only active function now on this tab.

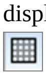


Firstly, select the beam element on the left side and type in value ***12*** in field ***Division into N segments***.

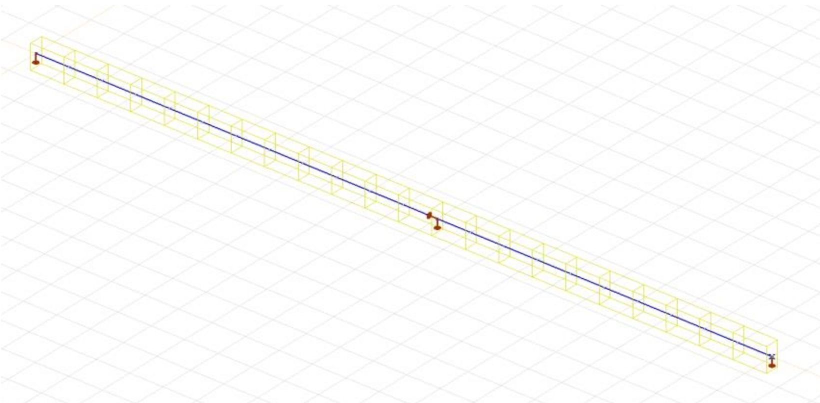


After select the beam element on the right side and divide it into ***10*** parts with the same method above.

Mesh display on/off

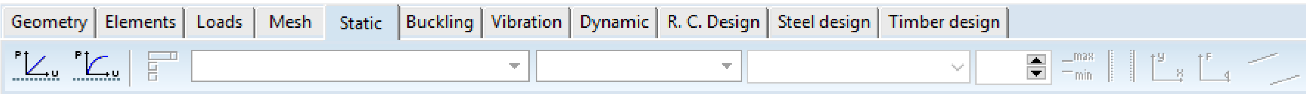


In the bottom right corner between ***Speed buttons*** find ***Mesh display on/off*** icon. With this icon the visibility of mesh can be switched on or off. Ask for a spatial view, the following shows up:

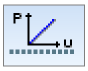


Static

Click on ***Static*** tab for running analysis:



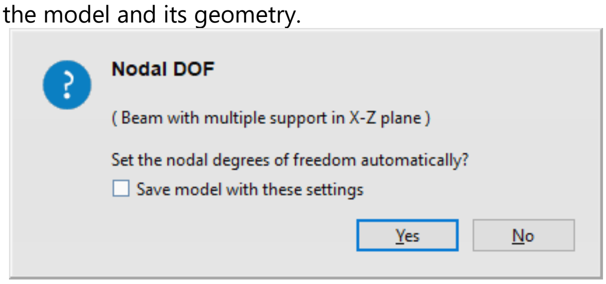
Linear static analysis



Click on ***Linear static analysis*** icon to run analysis. Before calculation, the software warns to save the model if it was not saved previously.

Nodal degrees of freedom

The analysis starts with a warning message. The nodal degrees of freedom must be set, that have not been done before. In this case the software examines the model and offers a possible setting based on the model and its geometry.



In our model the beam is not fixed in ***Y*** direction, on the other hand it is free to rotate around ***X*** direction. Please note: the software only filters out the basic faults in the data input, but cannnot find every fault beforehand that can cause problem in the stiffness matrix.

On the dialogue panel check ***Save model with these settings*** and click ***Yes*** to accept suggested settings (***Beam with multiple support in X-Z plane***).

Remarks: on ***Elements*** tab with ***Nodal degrees of freedom*** function the nodal ***DOF***s can be set.
For all the nodes the following setting should be done: e_x – free, e_y – constrained, e_z – free, Q_x – constrained, Q_y – free, Q_z – constrained. This is the same as default settings for ***Frame in X-Z plane***, see in list above the components.

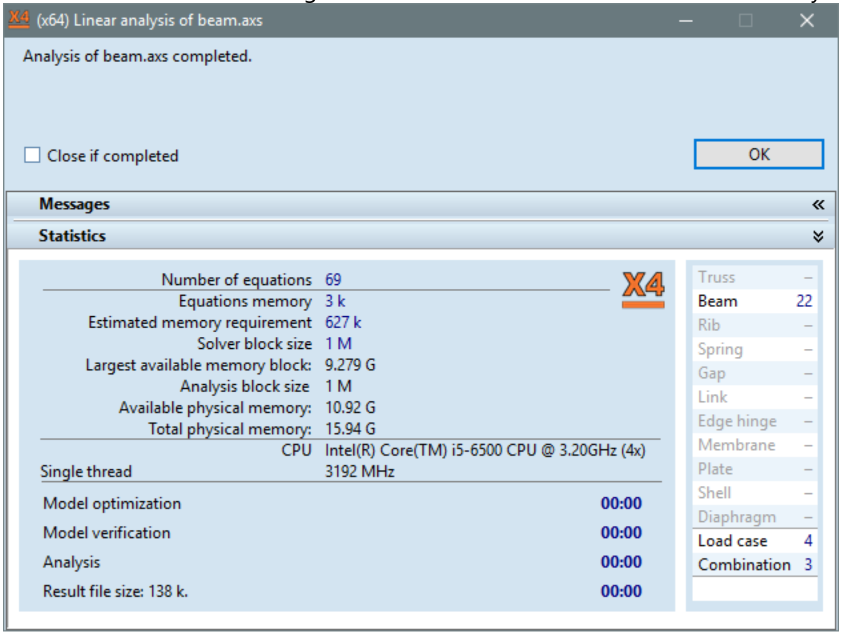
Statistics

The calculation continues and finishes with the following message:

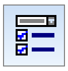
Click on ***Statistics*** to see more information about the analysis:

Static

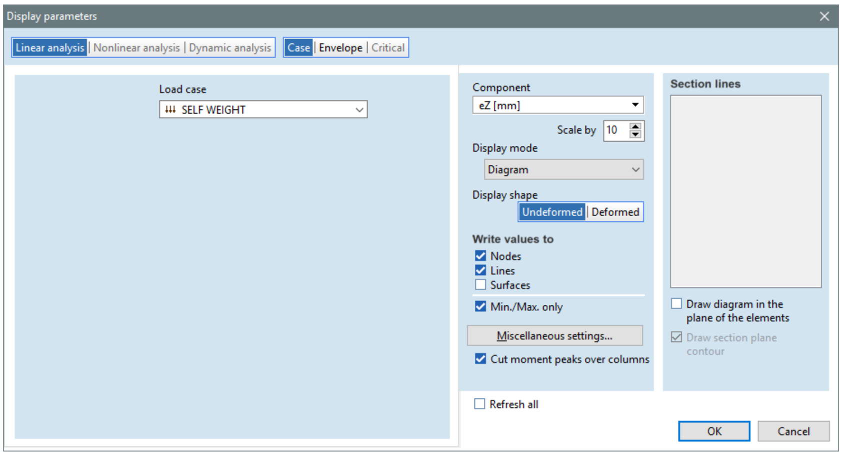
Close window, click on ***OK*** button. The software activates the result display: ***SELF WEIGHT*** load case is active, the vertical deformation ***ez [mm]*** is shown on the screen in ***Isosurfaces 2D*** display mode. Note, the deformation calculated in linear analysis are not equal to the real deformations of the reinforced concrete beam. After defining the reinforcement of the beam, nonlinear analysis must be done.



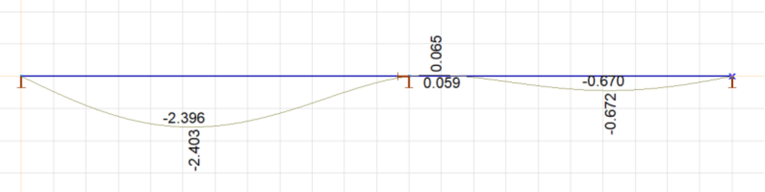
Result display parameters



The following window shows up after clicking on ***Result display parameters*** icon and do the following settings:

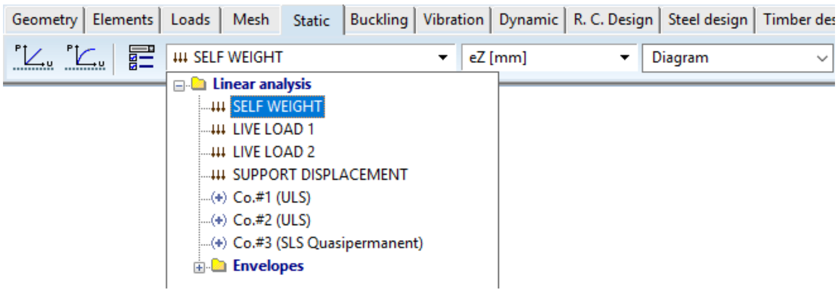


Select ***SELF WEIGHT*** load case and result component ***Displacement***, ***ez [mm]***. Change ***Display mode*** to ***Diagram***. If we leave ***Display shape*** on ***Undeformed*** the results are drawn to **undeformed** model. Check ***Lines*** and ***Nodes*** boxes at ***Write Values to*** setting.
After click on ***OK*** the following diagram will be displayed:

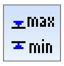


The scale of the diagram can be given or set at the field next to the ***Display mode***.

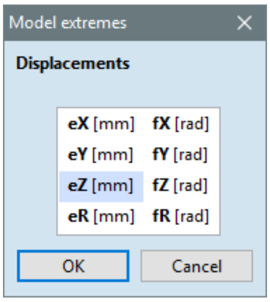
Check deformations in all load cases and combinations. Most of the errors in input data can be filtered out with thorough inspection. To do this, click on the list right next to ***Result display parameters*** icon. Select the first load combination ***Co.#1***.



Min, max values

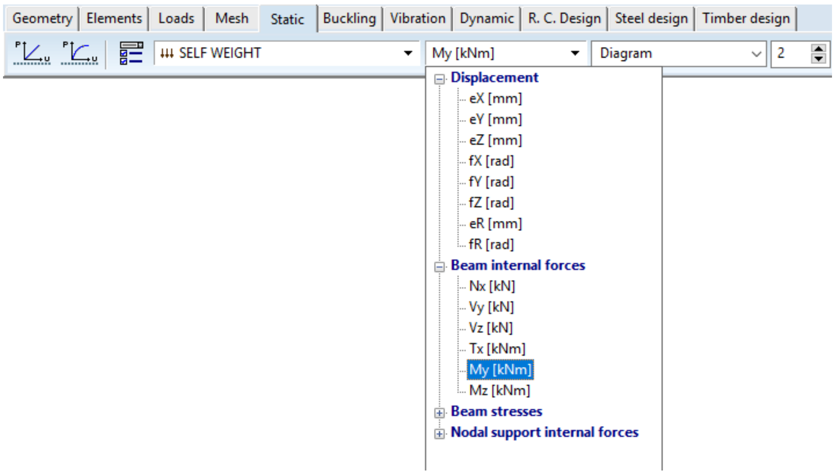


To find minimal and maximal values click on ***Min, Max Values*** icon. The following window shows up:

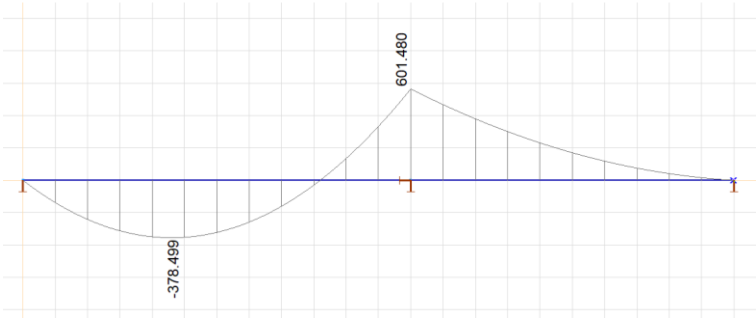


Select ***ez*** and click on ***OK***. The result panel shows the highest negative deformation and its position, after clicking on ***OK*** the highest positive deformation is shown. Click again on ***OK*** to close.

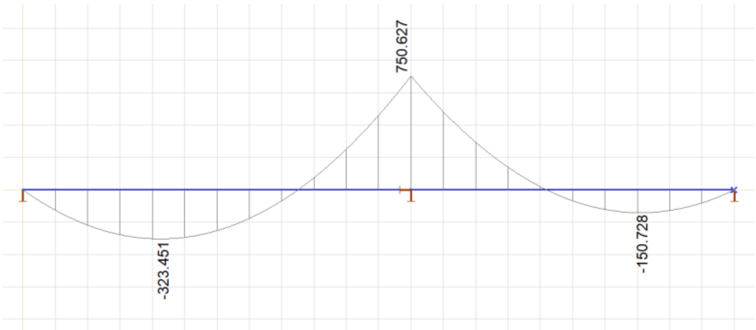
Click on the second drop-down list (next to load case list) and appropriate result component can be selected from this list: displacement, internal forces, stresses, etc... Select internal forces My [kNm] first-ly for the first and second load combination (***Co.#1***, ***Co.#2***).



Co.#1 – ***My*** [kNm]:

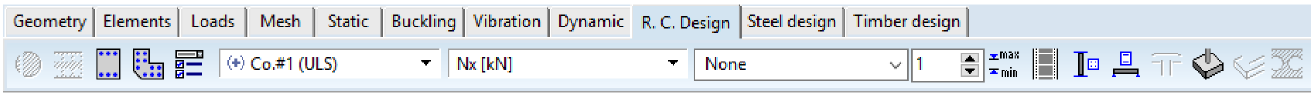


Co.#2 – ***My*** [kNm]:



R. C. design

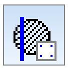
To design the reinforcement of the beam, click on ***R. C. design*** tab:

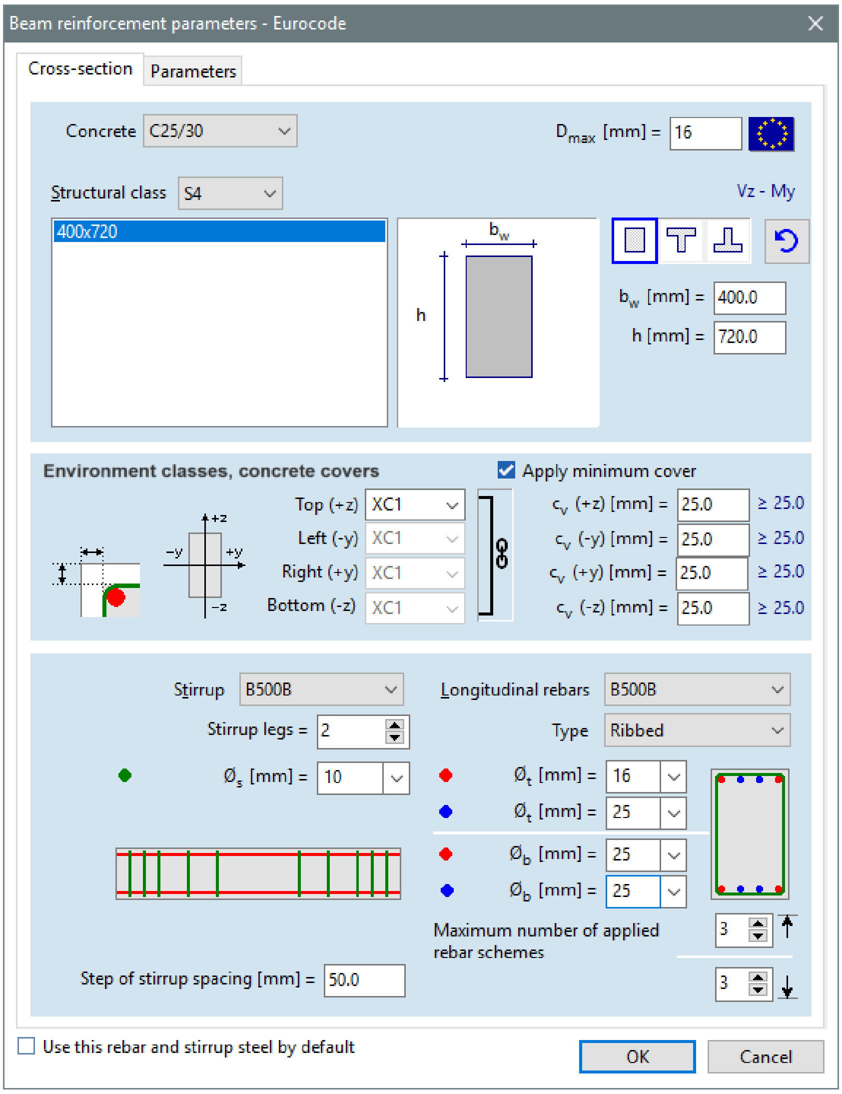


Beam reinforcement design

Click on ***Beam reinforcement design*** icon then select all the beam elements with ***All*** (*) button and finally click on ***OK***. In ***Beam reinforcement parameters*** window set the followings:

Reinforcement parameters





On ***Cross-section*** tab the cross-section of the beam can be modified if it necessary, but please note that the internal forces will be recalculated only if a new static analysis is done. So, this option should be used only in justified cases.

The material of ***Concrete*** can be modified, the structural class and the maximum aggregate size of concrete can be set. Leave the latter at the default setting of ***16*** mm.

Environment classes and concretes covers must be specified on all four sides of the cross-section. In our example select ***XC1*** environment class on every side. By clicking on the link symbol right to the environment classes will set the same environment class on all sides. ***Apply minimum cover*** to calculate the concrete cover automatically.

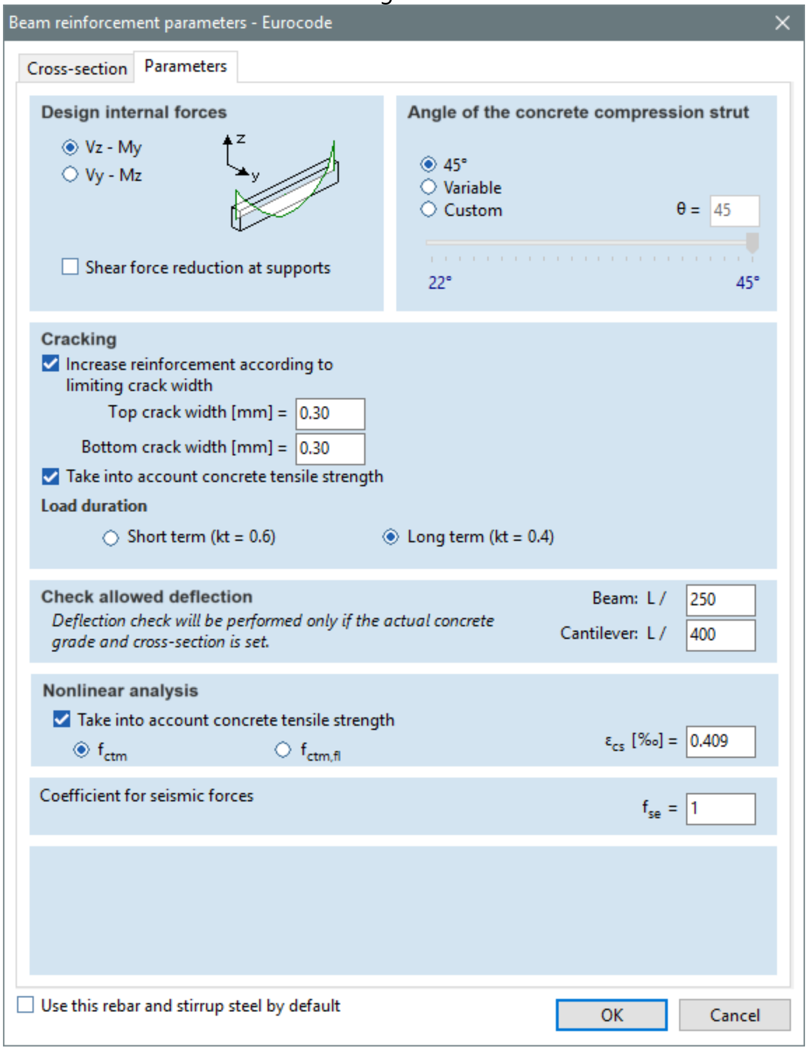
The reinforcement parameters can be set: stirrup parameters are on the left, parameters of longitudinal rebars are on the right. Stirrup is displayed in green, rebars at the corners are red, other longitudinal rebars are blue, their diameter at the top / bottom (*Øt*, *Øb*) can be set separately.

Set ***B500B*** material for the link and longitudinal rebars as well. Set diameter of link to ***10*** mm, the step of stirrup spacing should be the default value of ***50*** mm.

Set ***25*** mm for the diameter of bottom longitudinal rebars, but the top rebars at the corners should be ***16*** mm, for the internal top bars set ***25*** mm.

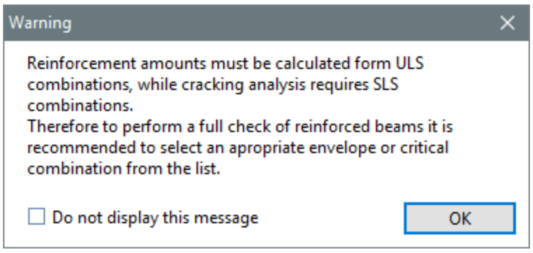
Maximum number of applied rebar schemes limits the number of different rebar distribution schemes applied along the beam (number of top and bottom distribution can be set separately).

On ***Parameters*** tab set the followings:



Checking ***Shear force reduction at supports*** allows the application of shear force reduction methods according to the current design code, but in our example, do not switch on this function.
Eurocode 2 allows specification of the ***θ*** angle of the concrete compression strut, leave this on the default value of ***45°***.
Checking ***Increase reinforcement according to limiting crack width*** the maximum allowed crack width values can be entered. In this case the program increases the top / bottom reinforcement to reduce the crack width under the specified value. To perform cracking analysis the load duration must be specified.
For ***Load duration*** set option ***Long term*** (kt=0.4).
Set ***L/250*** criteria for the ***allowed deflection*** of ***Beam***. Sign ***L*** represents the beam length. This check will be performed only if the actual concrete grade and cross-section is set.
Take into consideration of concrete ***tensile strength*** in ***Nonlinear analysis***.
Seismic load was not defined, so do not overwrite seismic coefficient, leave the default value of 1.

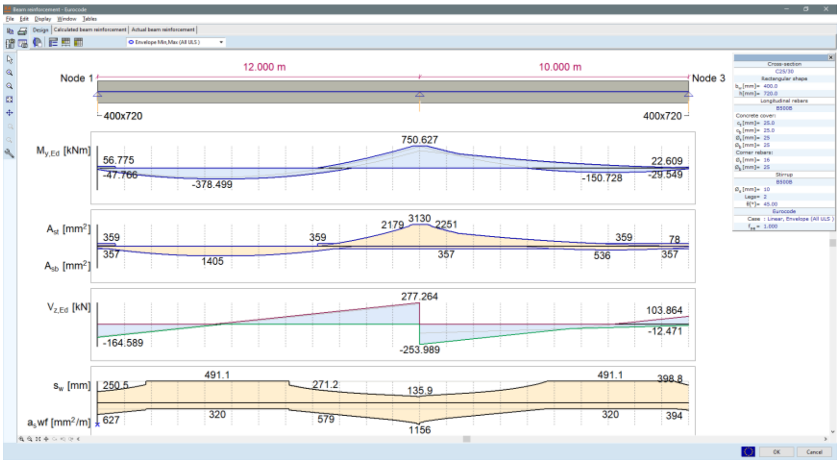
After click on ***OK*** to close window. The software shows a warning message as follows.



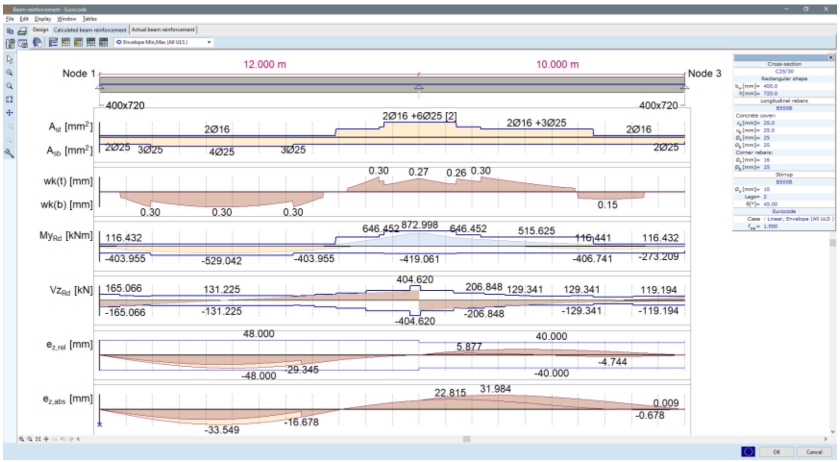
After taking notice of the warning, ***Beam reinforcement*** window in the background becomes active.

The following diagrams are displayed on ***Design*** tab:

- schematic view of the beam model with dimensions
- design moment $M_{y,Ed}$. If ***Envelope Min, Max (All ULS)*** is selected in the drop-down list, the envelope moment curves are displayed. The blue curve with thick line represent the offset design curve in accordance with national design code.
- the third figure presents the required top/bottom reinforcement.
- the fourth figure shows the envelope of design shear force
- the last figure represents the required shear reinforcement and stirrup spacing.



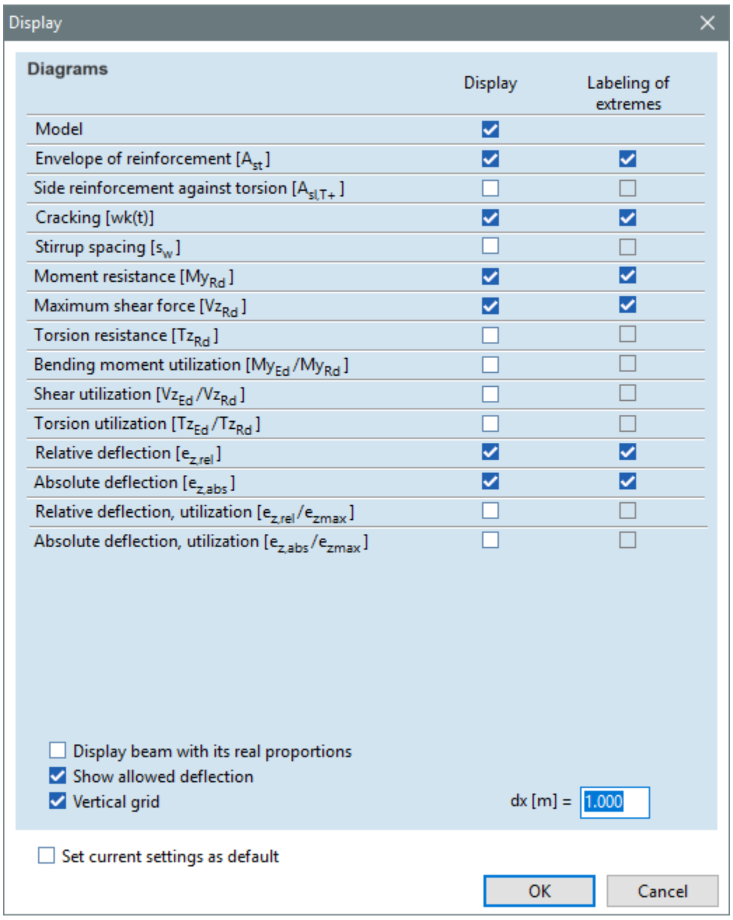
After clicking on ***Calculated beam reinforcement*** tab, the following diagrams can be displayed: Envelope of reinforcement *(As)*, cracking *(wk)*, moment resistance (*MyRd)*, maximum shear force *(VzRd)*, relative deflection *(ez,rel)*, absolute deflection *(ez,abs)*. To display other types of results and diagram set the ***Display parameters***.



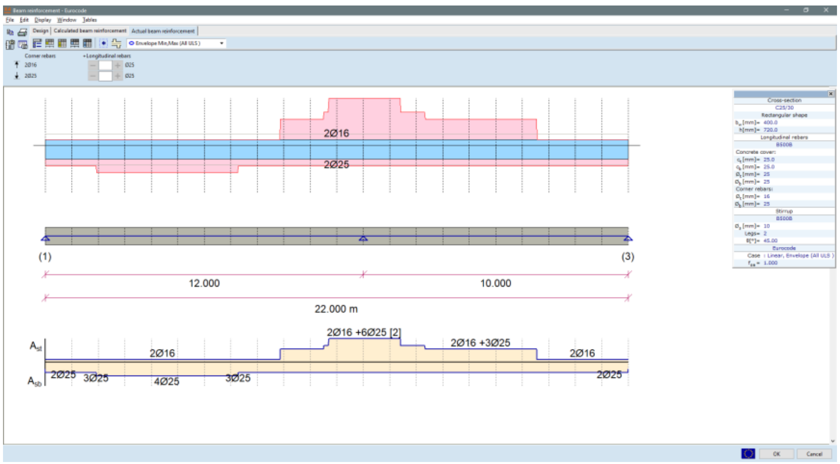
When determining the distribution of longitudinal reinforcement, the given parameters of rebars and the line meshing (see before) are considered.

Result display parameters

By clicking on ***Result display parameters*** button the following panel shows up, where you can optionally choose the diagrams you want to see.

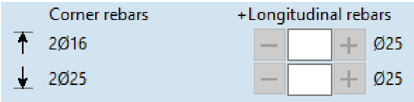


By closing the ***Result display parameters*** window, on the ***Actual beam reinforcement*** tab the calculated reinforcement can be applied as actual reinforcement. Now, click on that tab and apply the offered reinforcement.

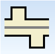


In the upper figure, pink coloring indicates the required reinforcement in case of the selected load combination (or load case). The blue coloring indicates the actual reinforcement (minimum reinforcement in the corners of the link). The vertical dashed lines represent the meshing of the beam (see before), the number of rebars can be set for each section. To select a line section, click on the sections within the area of the section. Additional sections can be added to the selection by holding down the right mouse button and dragging the mouse in the right direction or pressing ***Shift***.

The number of the longitudinal reinforcement can be modified. Edit boxes allow changing the number of top and bottom rebars in the selected elements. The **–** and **+** buttons decrease/increase the number of rebars.



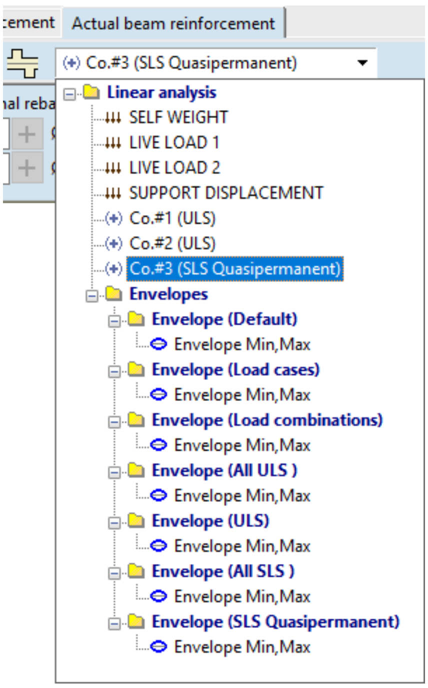
Apply calculated reinforcement



Select ***Envelope Min, Max (ULS)*** combination in the dropped-down list next to ***Apply calculated reinforcement*** icon. The following result will show up:

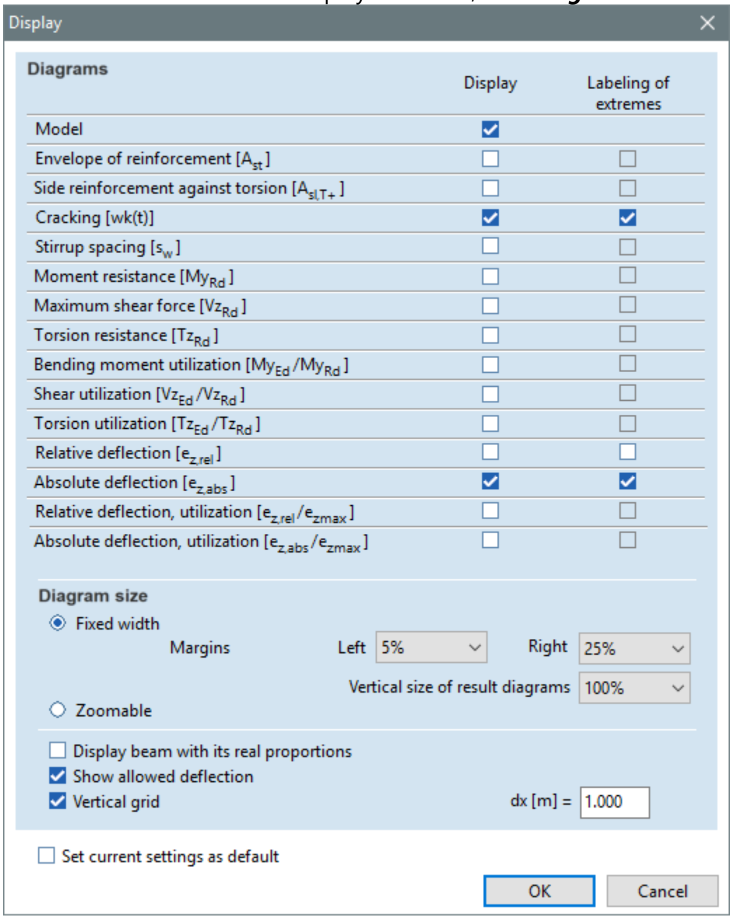
The software generates the actual reinforcement followings the shape of the internal forces by increasing or decreasing the number of rebars.

If ***Co.#3*** (***SLS Quasipermanent***) load combination is selected in the drop-down list, the deflection and crack width can be queried.

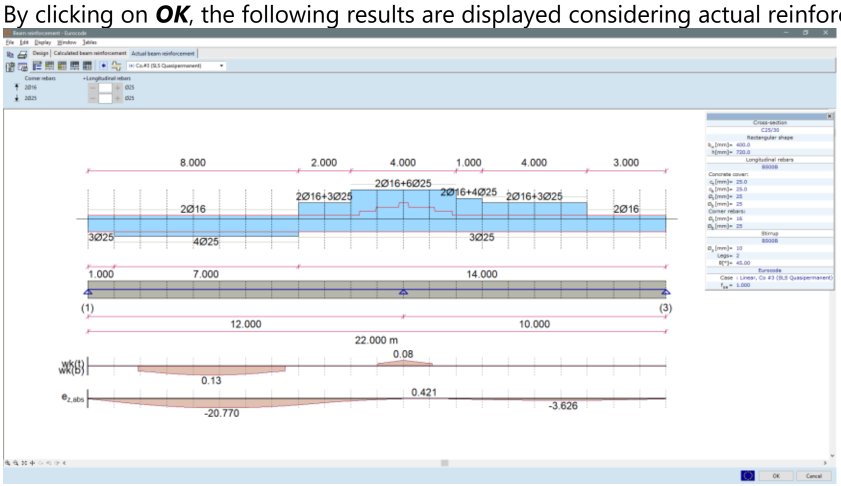


Sets display parameters

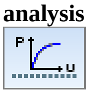
By clicking on ***Display parameters*** icon select only the required diagrams, the others should be switched off. Switch on the display of ***Model***, ***Cracking*** and ***Absolute deflection*** diagrams.



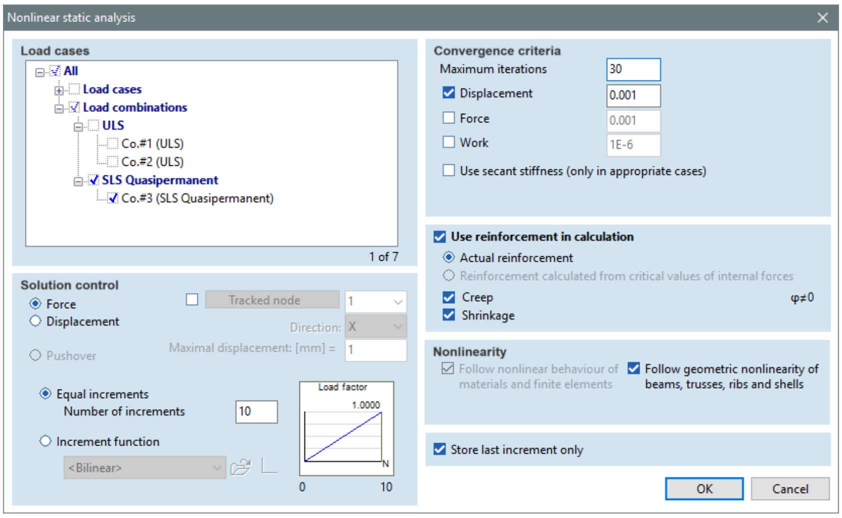
By clicking on ***OK***, the following results are displayed considering actual reinforcement.



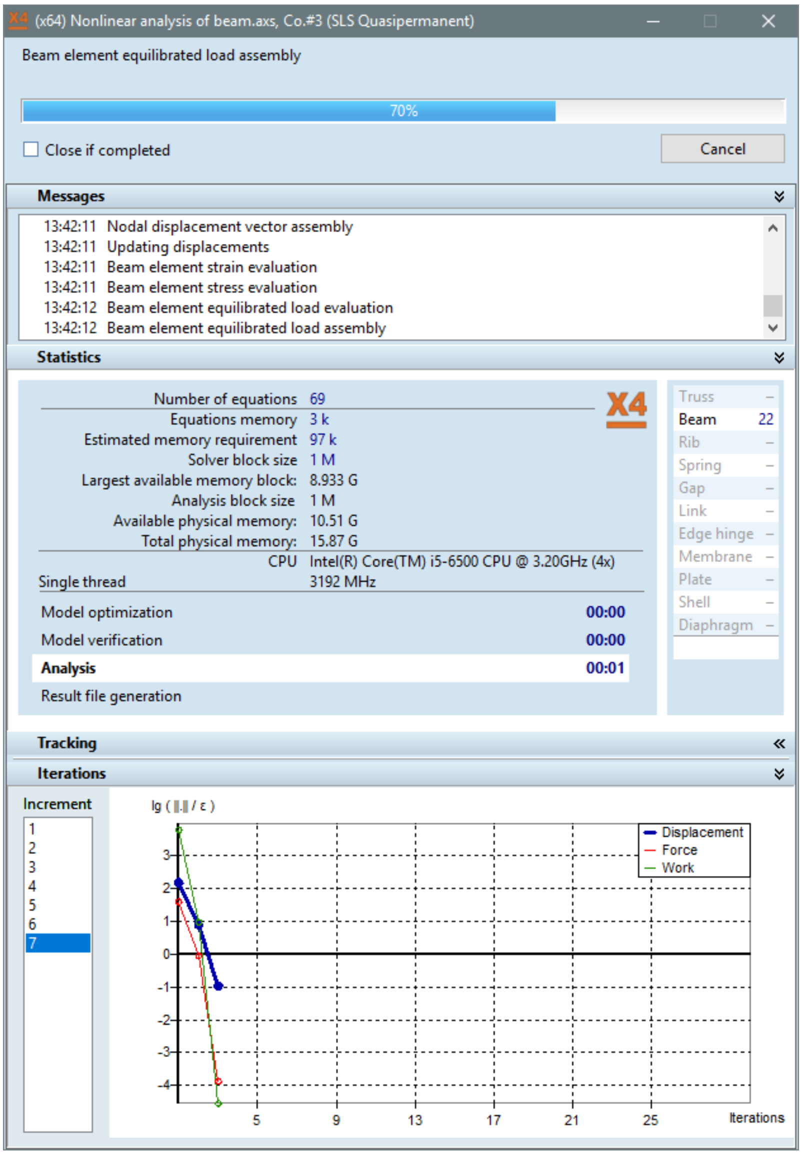
Nonlinear static analysis



The above results for deflection are results of an approximate calculation (creep is approximated but without considering the effect of shrinkage). A more accurate calculation can be made by nonlinear static analysis. This can be done by closing the ***Beam reinforcement*** window and going back to ***Static*** tab. Click on ***Nonlinear static analysis*** icon, the following screen will appear.

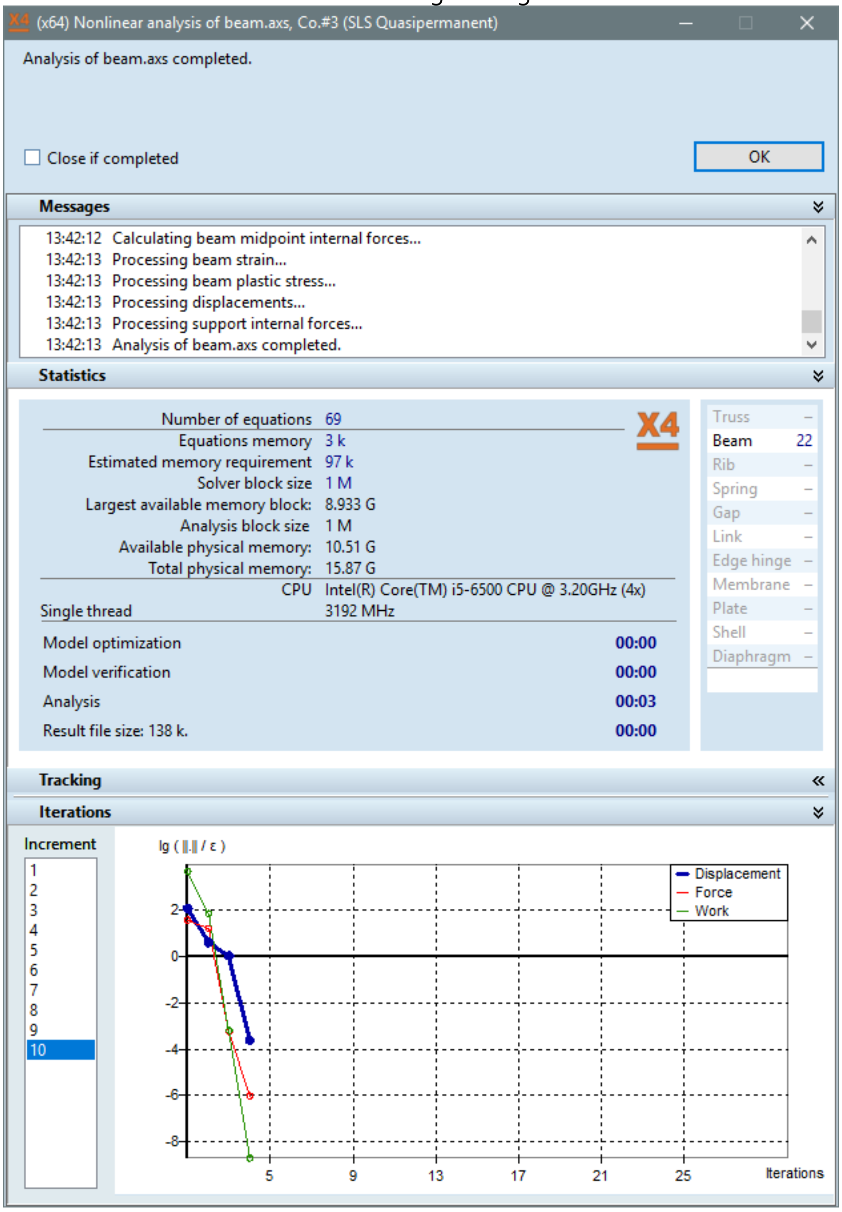


The only purpose of the nonlinear analysis is to study the deformation of the structure, so check ***Co.#3*** (***SLS Quasipermanent***) load combination in ***Load cases*** list. Check ***Use reinforcement in calculation***, and considering ***Creep*** and ***Shrinkage***. Set the other parameters according to the above window. Start analysis by pressing ***OK*** button, the following shows up:

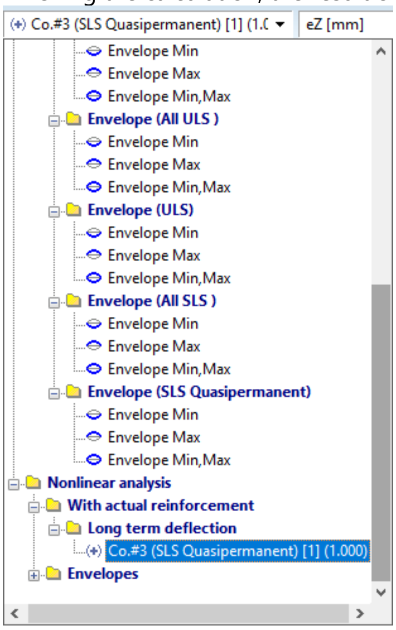


While running, information about the current steps of the calculation can be seen in the ***Messages*** list. In the ***Iterations*** window, the actual convergence of the calculation is presented during each increment.

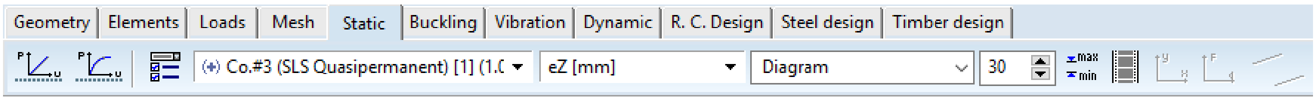
The calculation finishes with the following message:



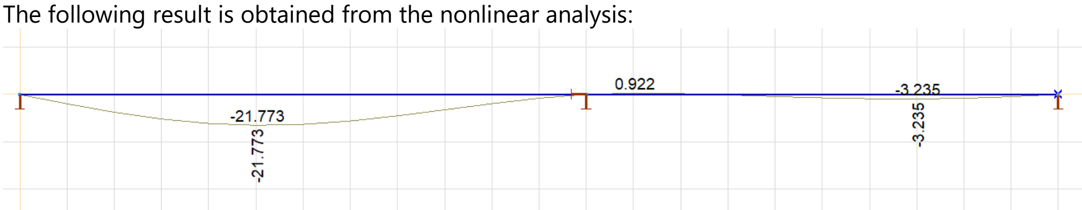
Finishing the calculation, the result of the nonlinear analysis can be found in the drop-down list.



Display the deflection curve of the beam as a result of nonlinear analysis (***Displacement – ez [mm]***). Set ***Diagram*** display mode, its scale can be freely adjusted (field next to ***Display mode***)



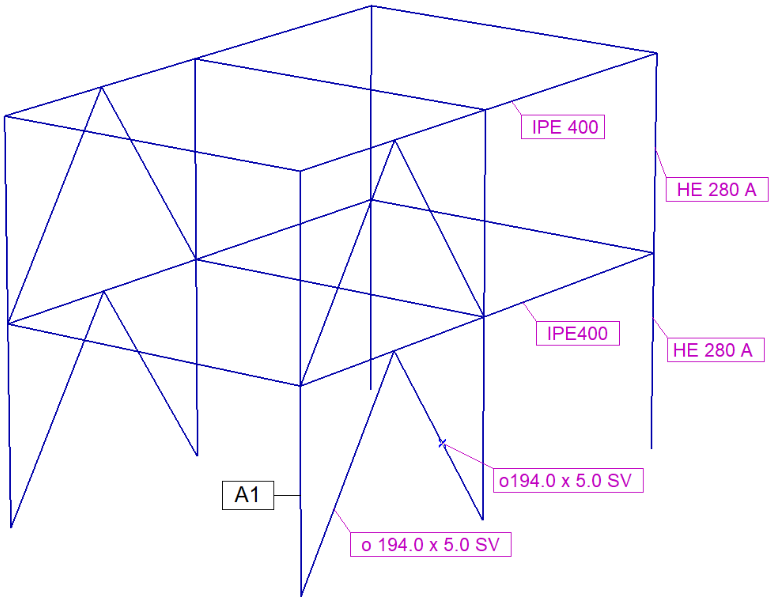
The following result is obtained from the nonlinear analysis:



2. FRAME MODEL

Objective

Analyse the following frame and design column **A1** and its pad footing:

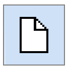


Consider IPE 400 rolled steel cross-section for horizontal beam elements and HEA 280 for columns. The cross-section of bracing elements is O194.0 x 5.0 SV. Use material S235 and apply Eurocode standard for the design.

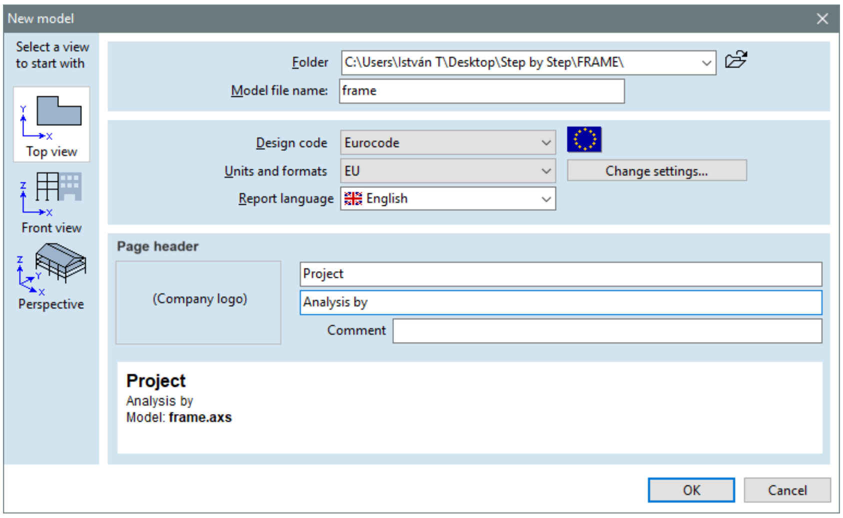
Start

Start ***AxisVMX4*** by double-clicking on ***AxisVMX4*** icon in its installation folder, found in ***Start*** – ***Programs*** menu.

New



Create a new model by clicking on ***New*** icon. In the dialogue window replace ***Model file name*** with '***frame***', select ***Eurocode*** from ***Design code***s and set ***Unit and formats*** to ***EU***.

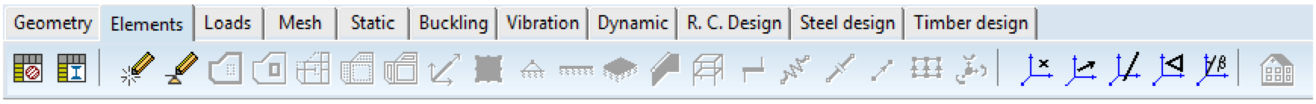


Starting workplane also can be set on the left in this window. Change workplane to ***X-Y Top view***. This setting can also be done using ***Choosing view*** icon on the editing interface.
Click ***OK*** to close the dialog window.

The geometry of the frame will be created by using of editing tools. This can be done in several ways. The geometry can be defined by drawing line elements, then structural elements and material properties can be assigned to those. In the following, the faster and complex ***Draw objects directly*** function is presented.

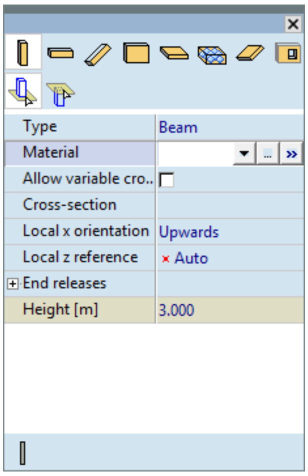
Define of geometry - Elements

Select ***Elements*** tab to bring up ***Elements*** toolbar.



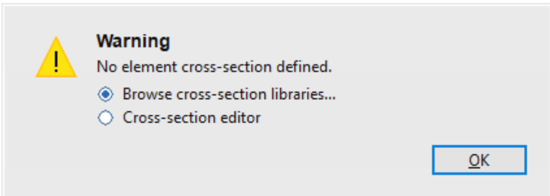
Draw objects directly

Firstly, the columns will be defined. By clicking on ***Draw objects directly*** icon shows following window:

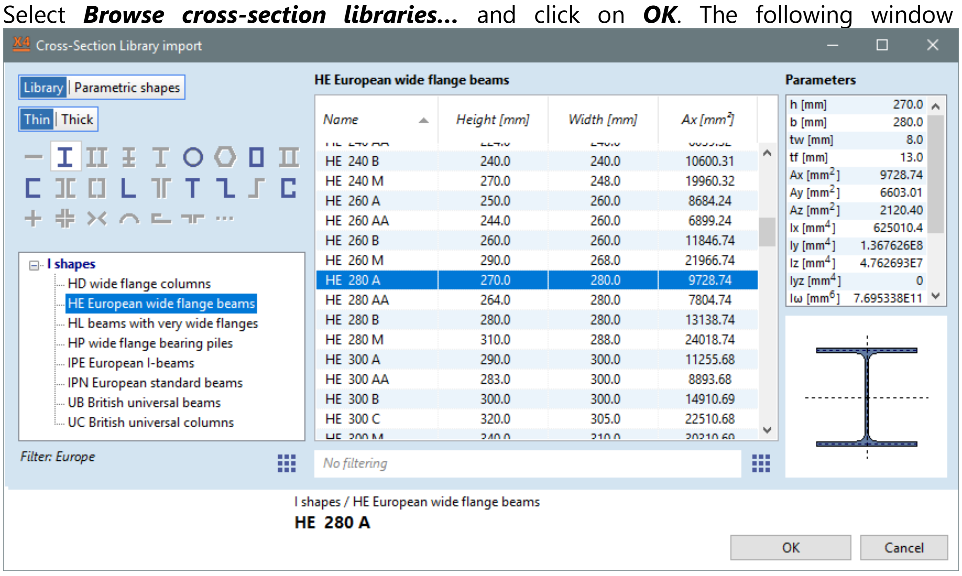


Click on ***Column*** icon even it is already selected.

The following window shows after clicking:

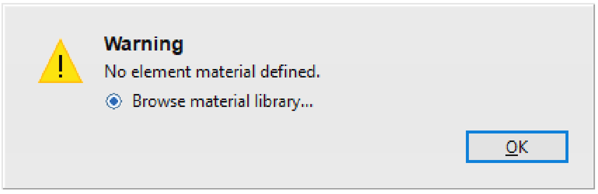


Select ***Browse cross-section libraries...*** and click on ***OK***. The following window shows up:



Roll down in the list of ***Cross-section tables*** by using vertical sliding bar (or roll mouse wheel) and select ***HE European wide flange beams***, and click on ***HEA 280 A***. Finish with ***OK***.

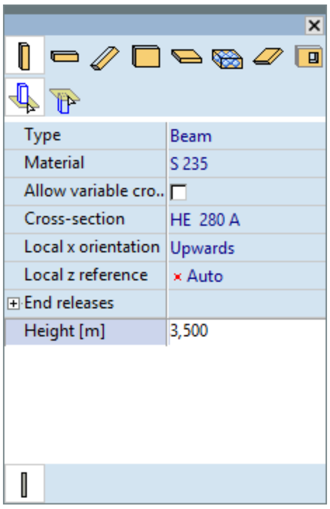
The following message shows up:



Roll down in the list of ***Materials*** by using vertical sliding bar (or roll mouse wheel) and click on ***S235***, then click on ***OK***.

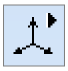
The selected cross-section and material are displayed in the ***Drawing objects directly*** window.

Set ***Height*** to ***3.5*** m in the window:

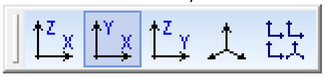


Note: in the window several parameters can be set which are not mentioned here. ***End releases*** can also be specified or the local co-ordinate system of the element can be modified. In our example, we use default setting for these. (The end releases of an element are fixed by default settings.)

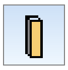
Views



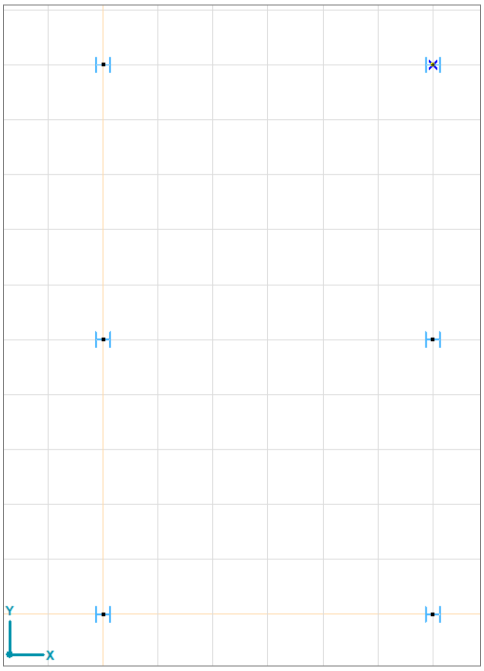
Change view to top view (***X-Y*** plane), if it is not the actual view. Note: if necessary, view can be modified with this icon, even if an editing command is active.



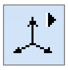
Column



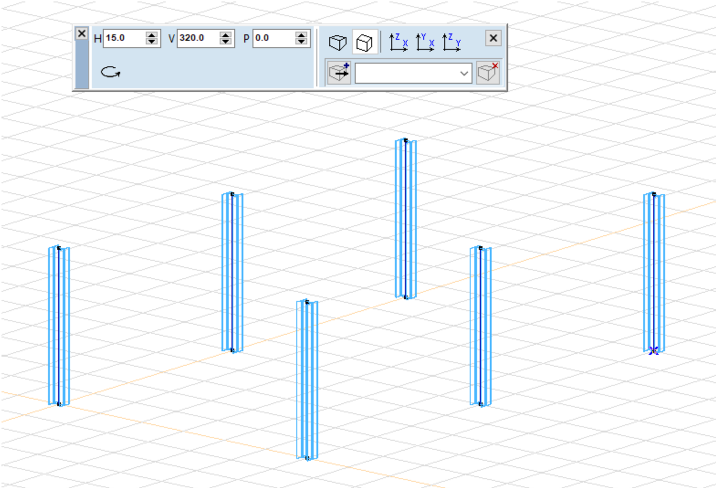
Click on ***Column*** icon below to place the columns.
Click on the following positions in the graphical interface:
(0; 0) – (6; 0) – (0; 5) – (6; 5) – (0; 10) – (6; 10).
The next figure shows the result, the columns in ***X-Y*** plane view:



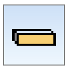
Views



Change view to ***Perspective***, activate ***Perspective palette*** (in ***Window*** menu) and set the next parameters: ***H=15***, ***V= 320*** and ***P=0***:

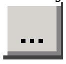


Horizontal beam

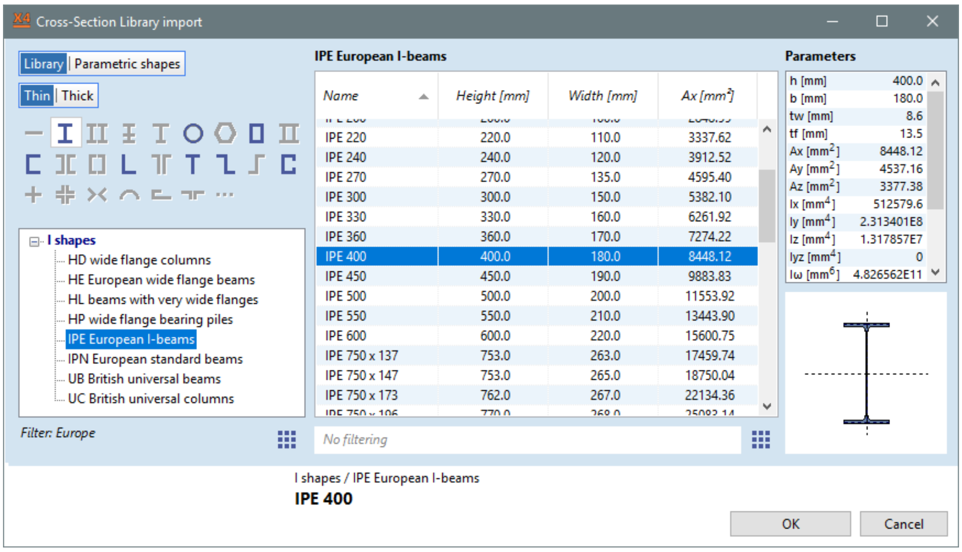


Click on ***Horizontal beam*** icon.

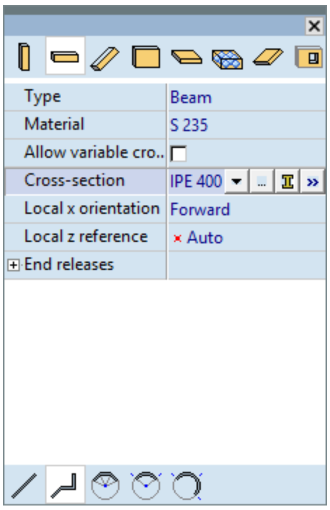
Cross-section library import



Click on ***Cross-section*** row in the window, then on ***Load from database*** icon. Select ***IPE 400*** from group ***IPE European I-beams***.

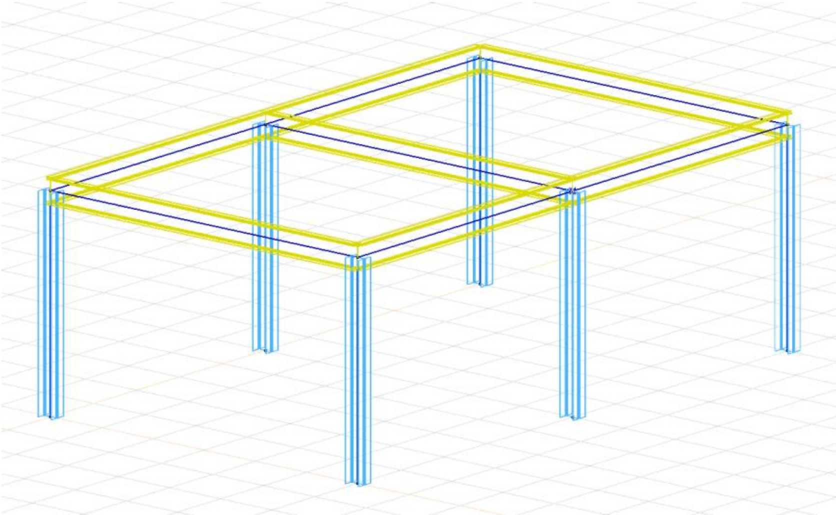


Close window with ***OK***, then the selected cross-section is now displayed on the panel:

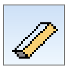


Beam polygon

Activate ***Beam polyline*** icon, then click on top of columns and draw the axis of beams as a polyline. Draw the perimeter beams first, then press ***Esc*** to cancel drawing. Finally, draw the inner beam as well. The next will be the result:

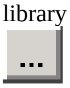


Beam



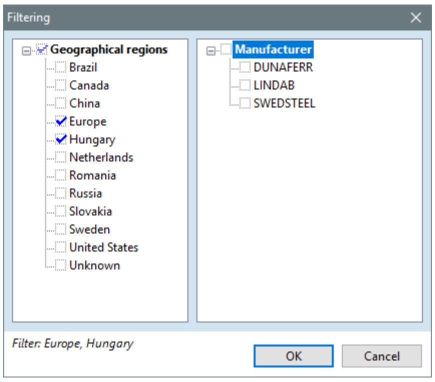
Change to ***Beam*** function on ***Draw objects directly*** panel.

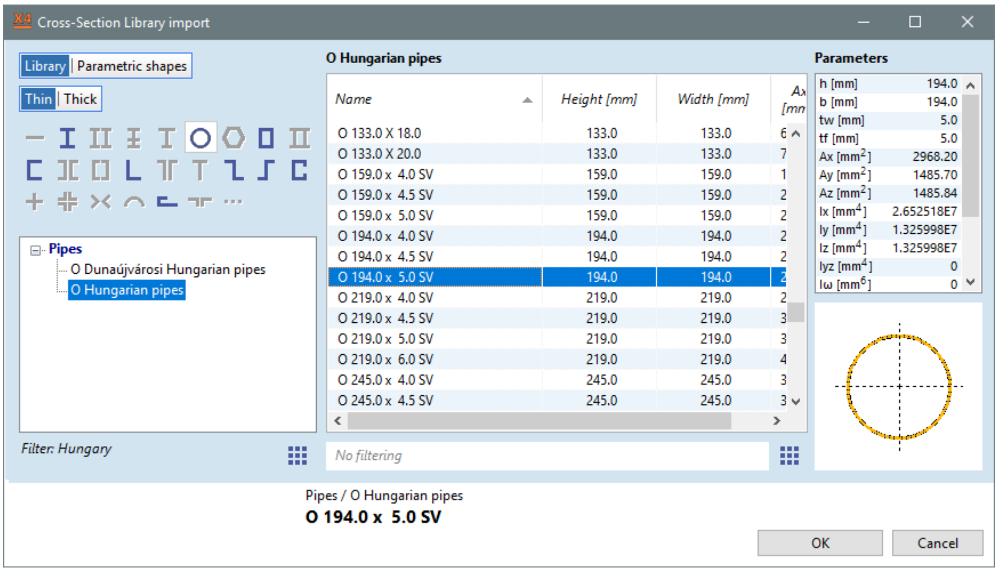
Cross-section library import



Filter

Click on ***Cross-section*** row in the window, then on ***Cross-section library import*** icon. Select ***Pipes***, but geographical region must be change to view also ***Hungarian pipes*** in the list. Activate Filter function and select also Hungarian shapes. Select ***o Hungarian pipes*** under ***Pipes*** group in the list of ***Cross-section tables***, then find and select ***O194.0 x 5.0 SV*** in ***Cross-section*** list:

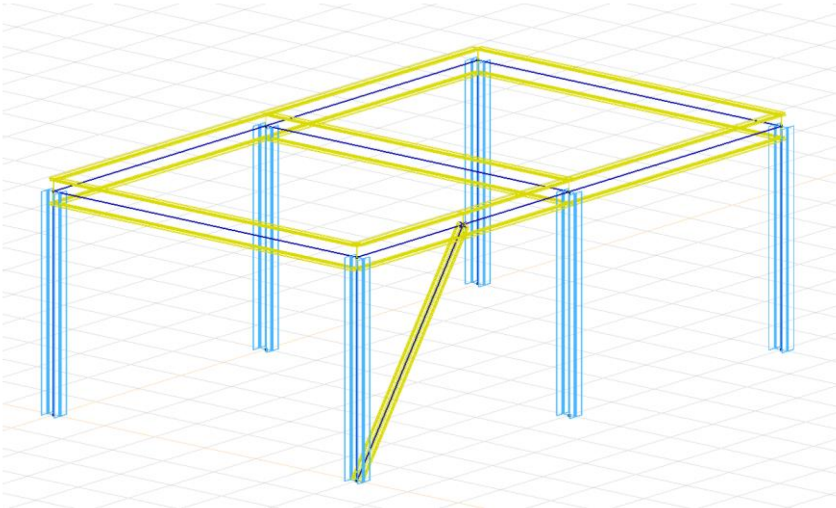




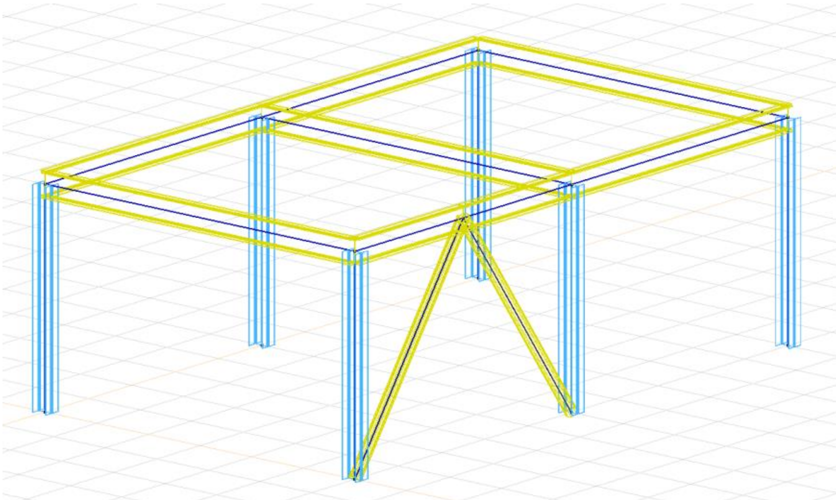
Close window with ***OK***.

Beam polygon

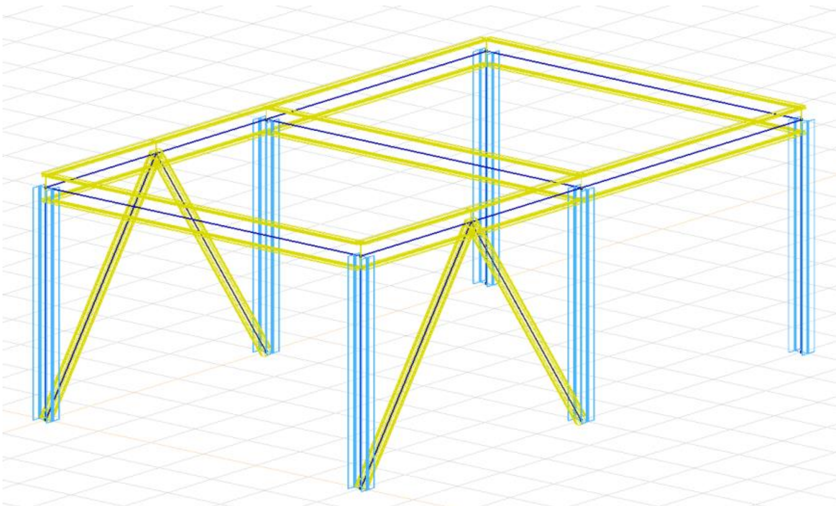
Draw polygon from the bottom node of ***A1*** to the centre of beam in ***Y*** direction as shown below:



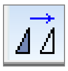
then continue drawing to the bottom node of the next column.



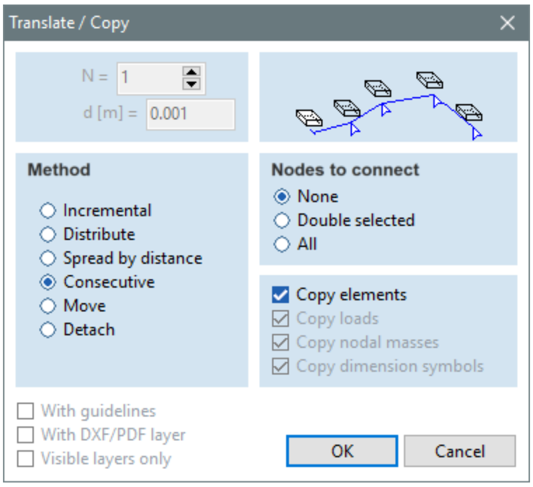
Press ***Esc*** to close polyline function, then repeat previous steps to specify the bracing elements on the other side of the structure.



Translate / Copy

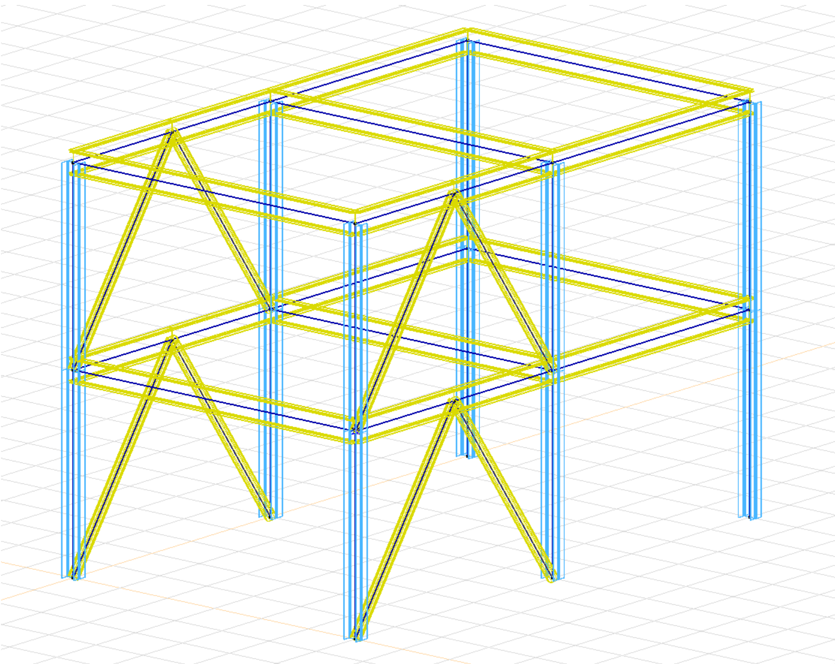


Now, all the structure will be copied in ***Z*** direction to define the upper storey. Activate ***Translate*** icon, select all the elements and in ***Translate*** window set the next:

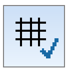


Select ***Consecutive*** and ***None*** checkbox under ***Nodes to connect*** title. Click on bottom node of one of the column, then on the top of it, finally press ***Esc*** to close command.

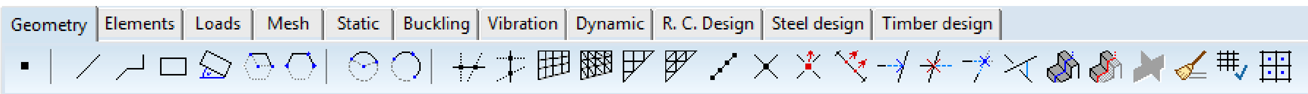
The following will be the result:



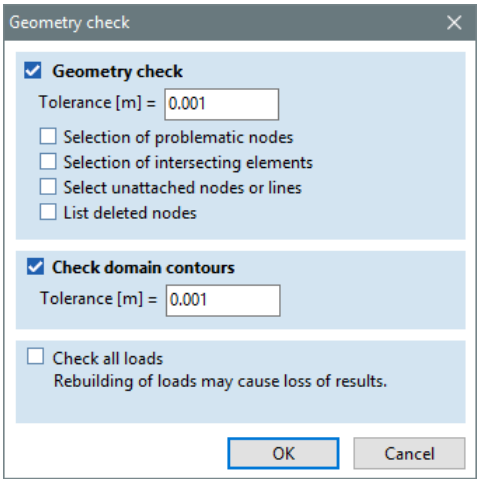
Geometry check



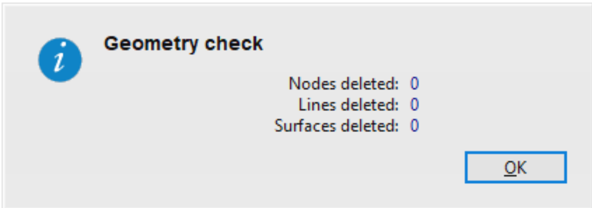
Click ***Geometry check*** icon to check the geometry of the frame, which can be on ***Geometry*** tab.



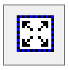
In the window that appears, user can set maximum ***Tolerance*** for checking points and user may also request highlighting of nodes and elements found during checking.



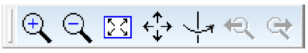
After the geometry check, a report is displayed about the events:



Zoom to fit

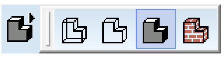


Select ***Zoom to fit*** from ***Zoom*** functions for better view.

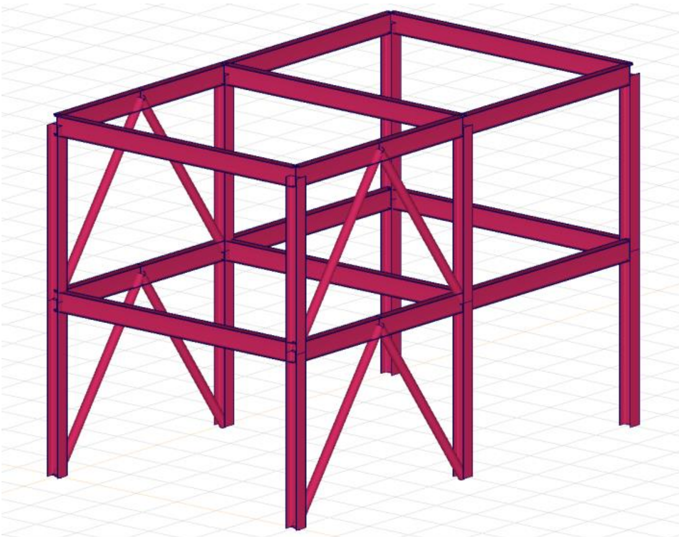


Rendered view

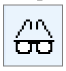
Select ***Rendered*** view from ***View*** modes:



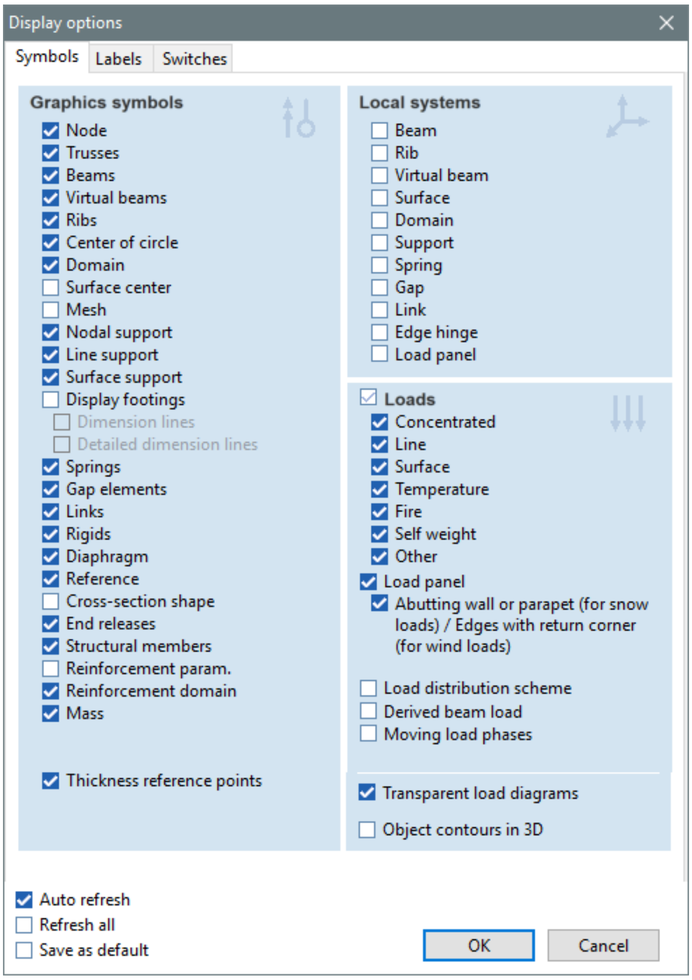
The following will be displayed:



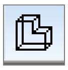
Display options



Activate ***Display options*** and uncheck ***Object contours in 3D*** on ***Symbols*** tab.



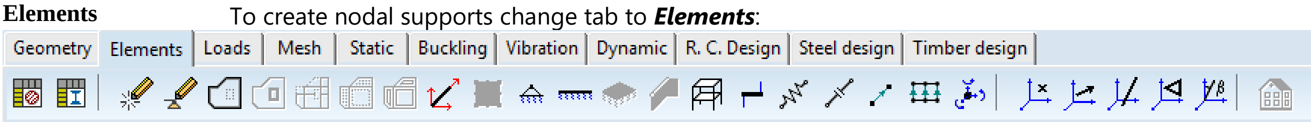
Wireframe view



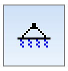
Change back to ***Wireframe*** view:

Elements

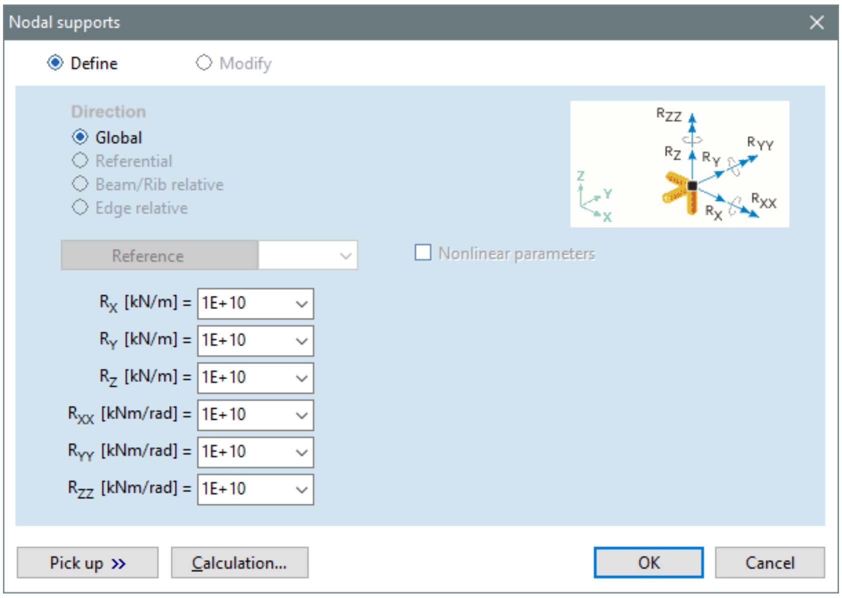
To create nodal supports change tab to ***Elements***:



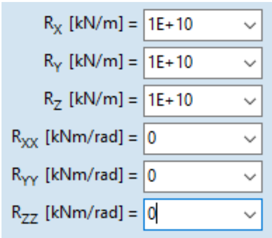
Nodal support



Click on ***Nodal support*** icon and select bottom nodes of columns, then confirm with ***OK***. In the next window the stiffness can be set for the supports:



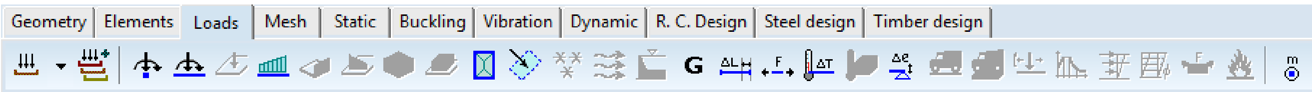
Set rotational stiffness components to ***0***, but leave translational stiffness on default value (***1E+10***), as shown below:



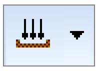
then close window with ***OK***.

Loads

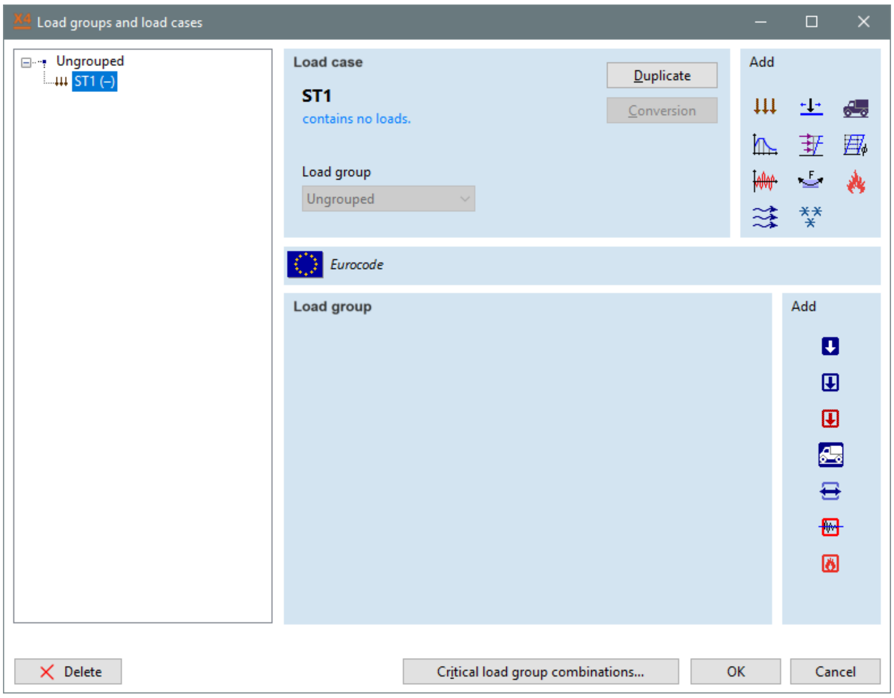
To specify the loads on the frame, change tab to ***Loads***:



Load cases and load groups

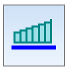


Various loads should be separated into load cases. Click on ***Load cases and load groups*** icon to add new load cases.

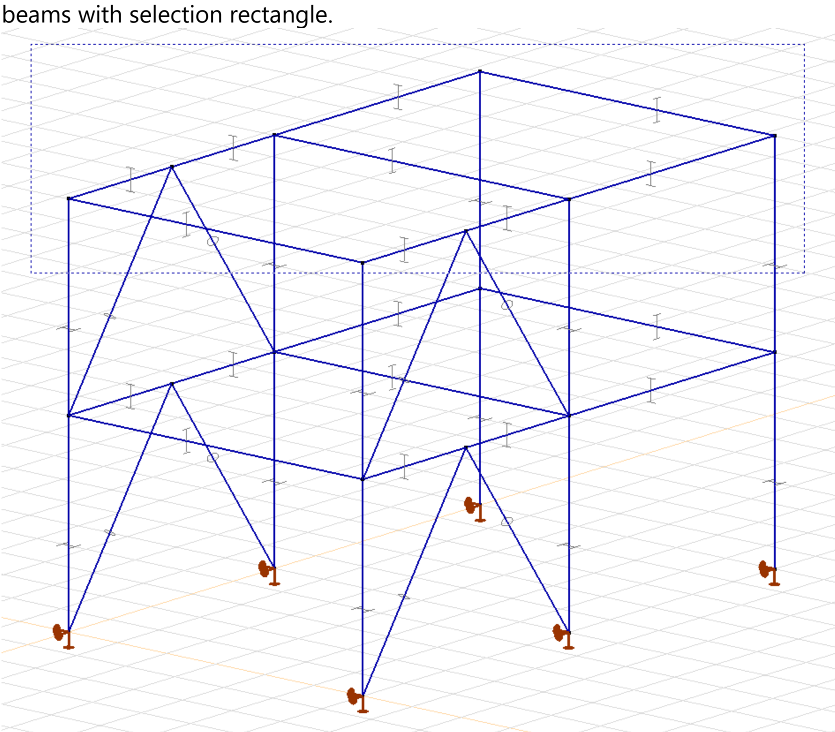


In the window that appears, click on the name **ST1** in the top left corner and rename it to ***LIVE 1***. (**ST1** is an automatically generated load case, which should be renamed in our example.) Close window with ***OK***, then the previously edited load case (***LIVE LOAD 1***) is active. The actual load case is indicated on ***Info palette***, shown below:

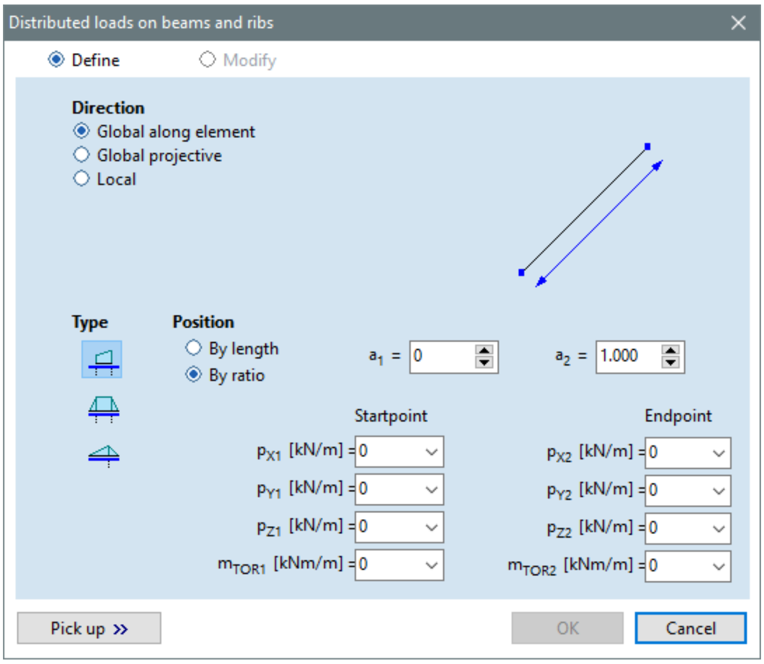
Load along line elements



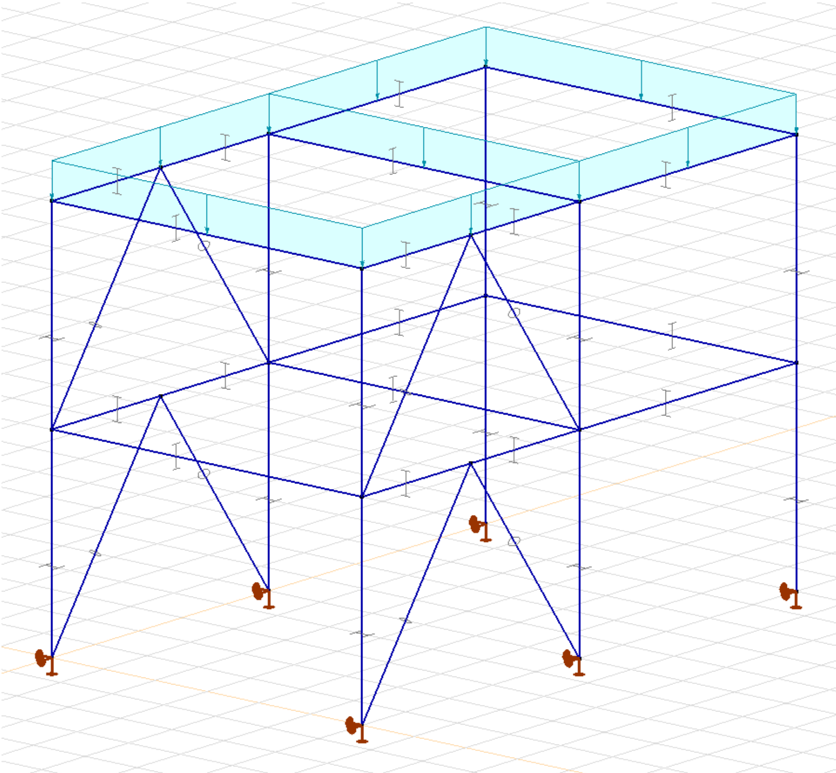
Add line loads to all horizontal beams. Specify **50 kN/m** to beams at lower level and **25 kN/m** to beams at upper level in ***-Z*** direction. Activate the ***Load along line elements*** function and select upper beams with selection rectangle.



Confirm selection with ***OK***, then in the following window set load parameters.

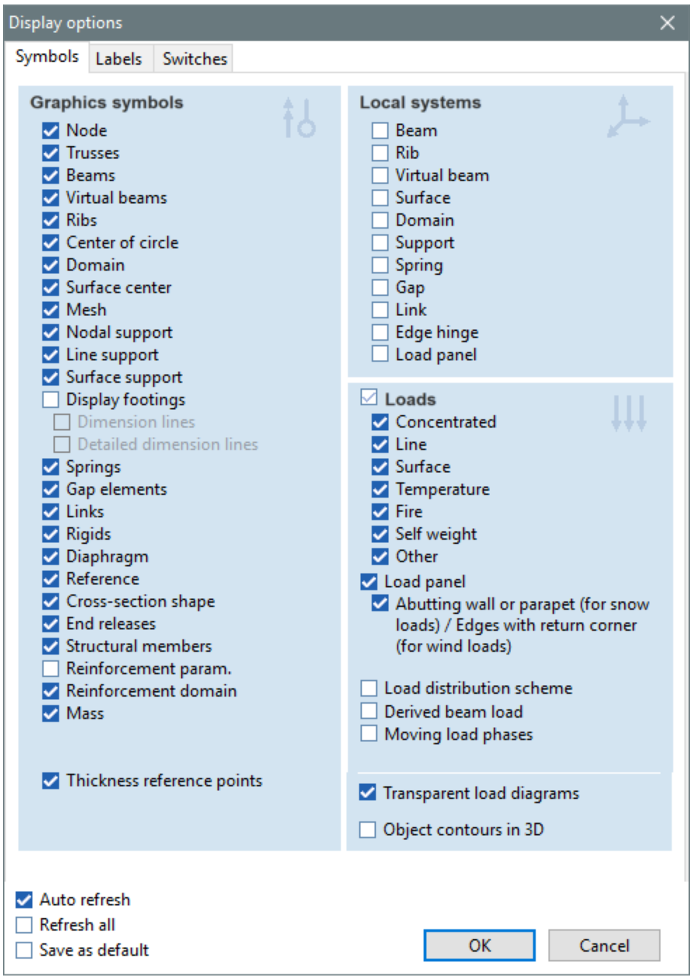


Set ***p_{Z1}*** and ***p_{Z2}*** to ***–25***, then finish with ***OK***, then the load is indicated on the beam elements in cyan:

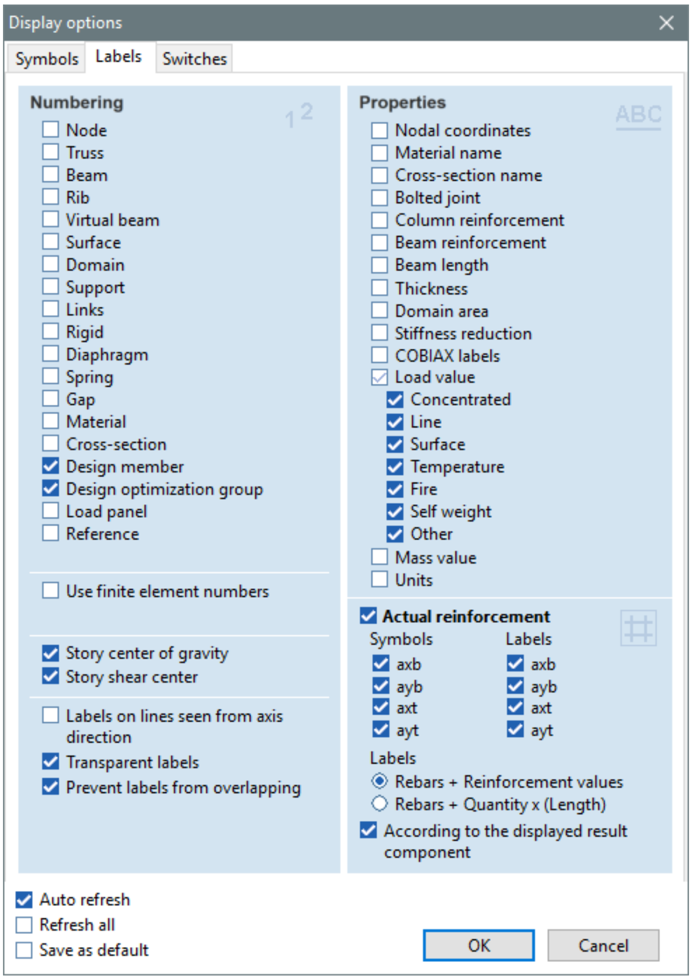


Display options

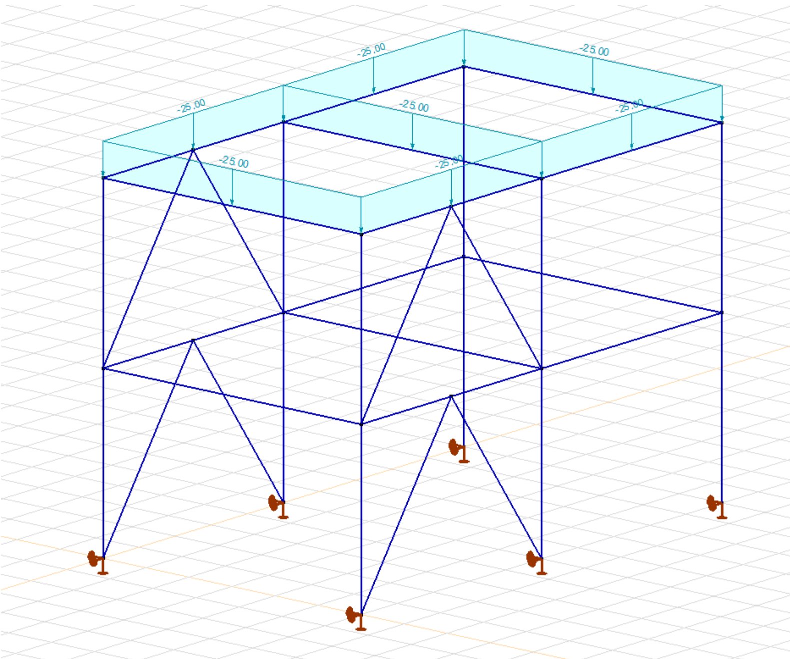
If the load intensity is not labelled automatically (this function is not active), then open ***Display options***:



In the window, select ***Labels*** tab and check ***Load value*** under ***Properties*** title:



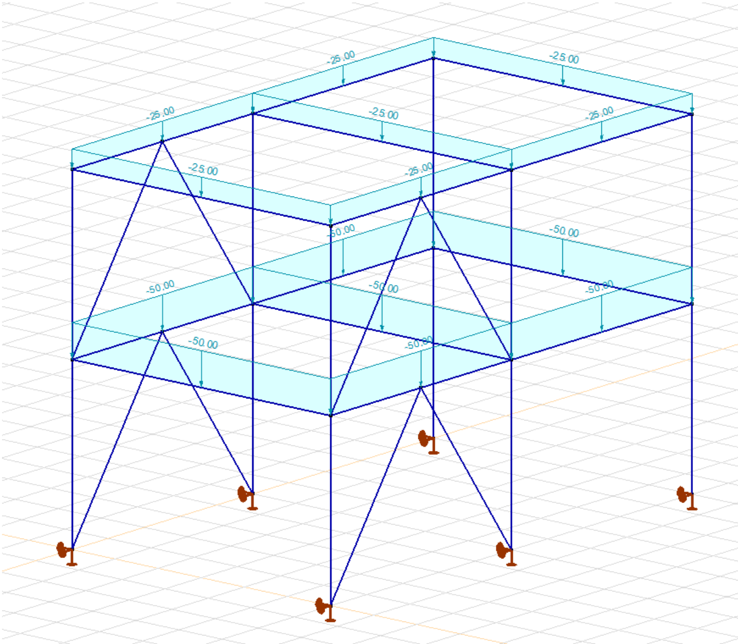
Close with ***OK***, then load intensity will be displayed:



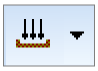
Load along line elements

Activate again ***Load along line elements*** and select lower beams, then confirm with ***OK*** and set p_{Z1} and p_{Z2} to ***–50***.

Close function with ***OK***, the following will be displayed:

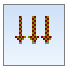


Load cases and load groups



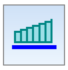
Click on ***Load cases and load groups*** icon.

Static load case

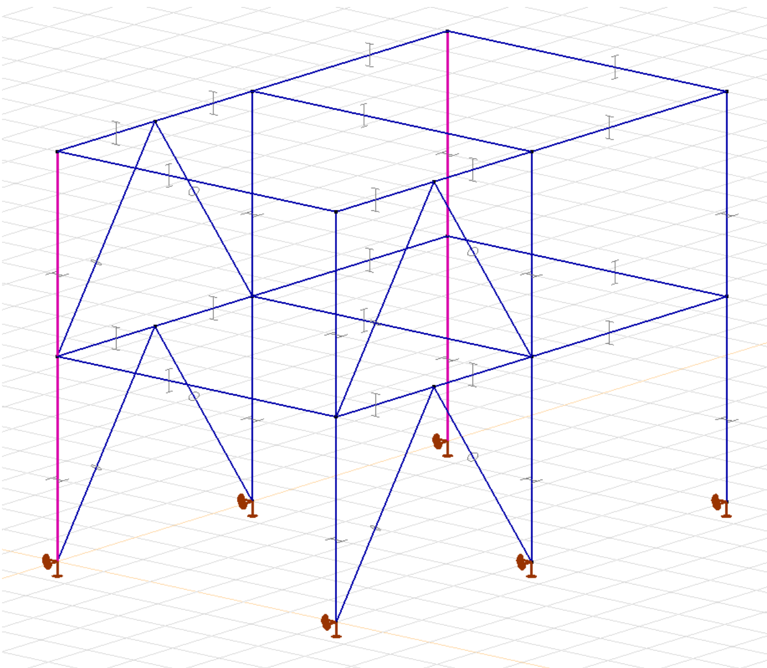


Add a new load case to the list by clicking on ***Static*** icon in ***New case*** top icon row, and rename it to ***WIND***. After confirming with ***OK***, all loads predefined will 'disappear' because the active load case will be the newly defined one, as it is shown on ***Info palette***.

Load along line elements

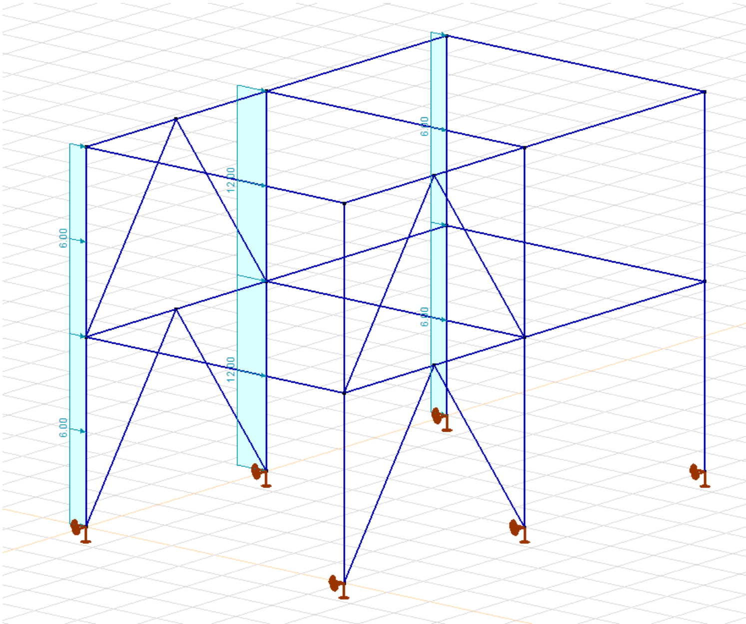


Click on ***Load along line elements*** icon then select the rear two columns in the corner, as shown below:



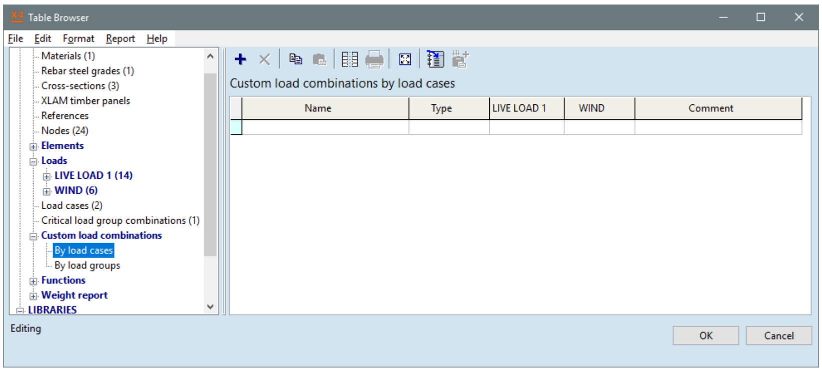
then apply ***6*** kN/m line load in ***x*** direction (p_{X1} and p_{X2}). The same way, apply ***12*** kN/m to the central column in ***x*** direction.

The next is the result:



Load combinations

Create load combinations on the basis of load cases. Using load combinations, load cases can be added together, combining them by various factors.
Activate ***Load combinations*** icon, the following window shows up:



New row

Create new ***ULS (ultimate limit state)*** load combination by clicking on ***New row*** icon.

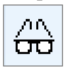
Assign the following factors to each load cases:

LIVE LOAD 1	1.50	<**Tab**>
WIND	1.50	<**Tab**>

Enter the values in the appropriate data field, then click ***OK*** to finish.

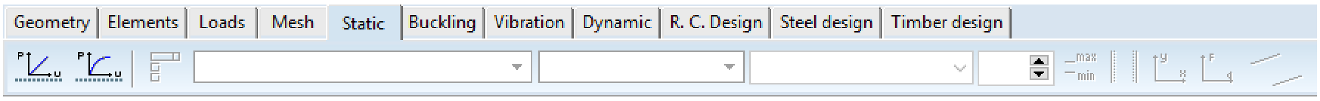
With this final step the data input has been finished.

Display options

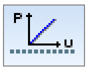


Activate ***Display options*** and change to ***Symbols*** tab. Uncheck ***Node***, ***Cross-section shape*** and ***Loads*** checkboxes then uncheck ***Load value*** on ***Labels*** tab.

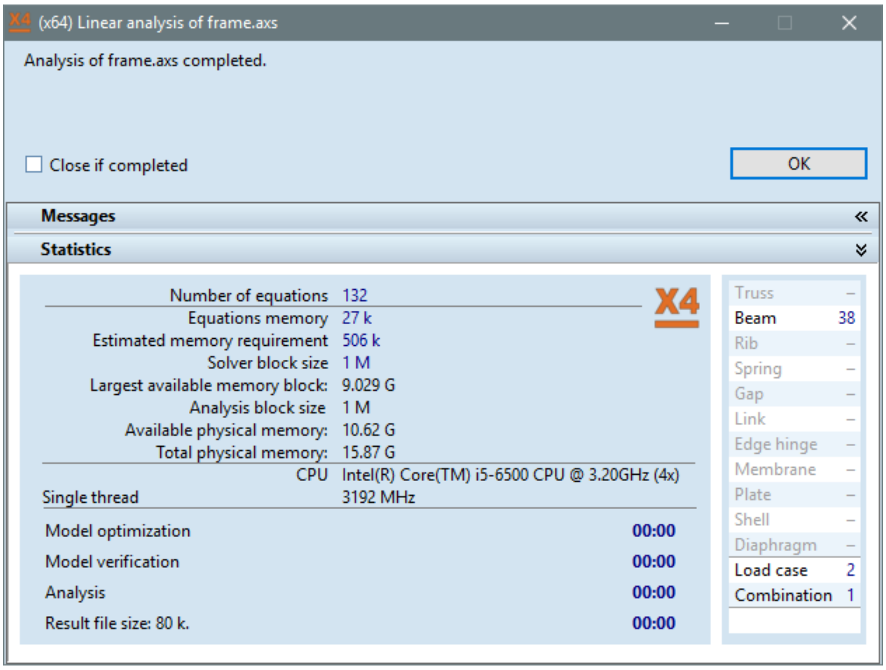
Static Click on ***Static*** tab for running analysis:



Linear static analysis

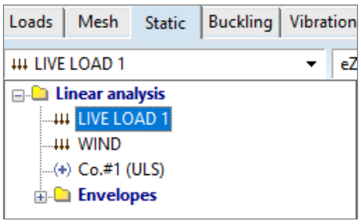


Click on ***Linear static analysis*** icon to run analysis, the following shows up:

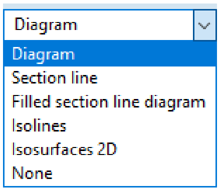


Click on ***Statistics*** to see more information about the analysis.

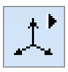
Click on ***OK*** after analysis. Automatically ***Static*** tab is activated with ***ez [mm]*** vertical deformations for ***LIVE1*** load case in ***Isosurfaces 2D*** view. To see results of other load combination, select combination ***Co.#1*** from drop-down list.



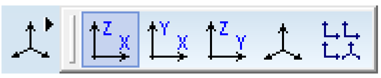
Change results view from ***Isosurfaces 2D*** to ***Diagram***:



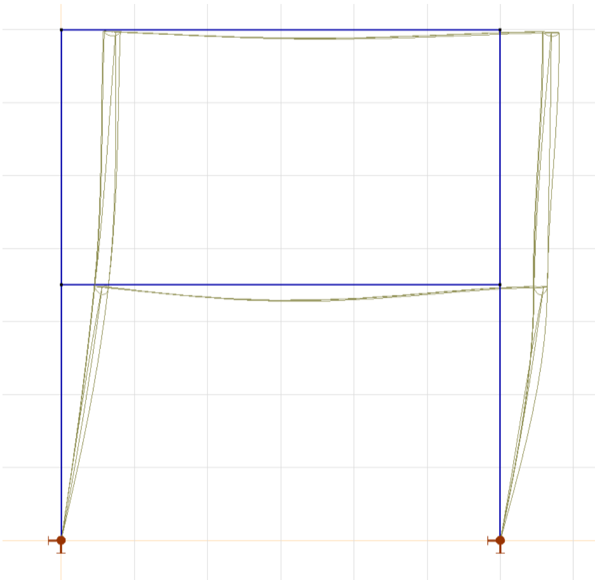
Views



Change view to ***X-Z*** plane!



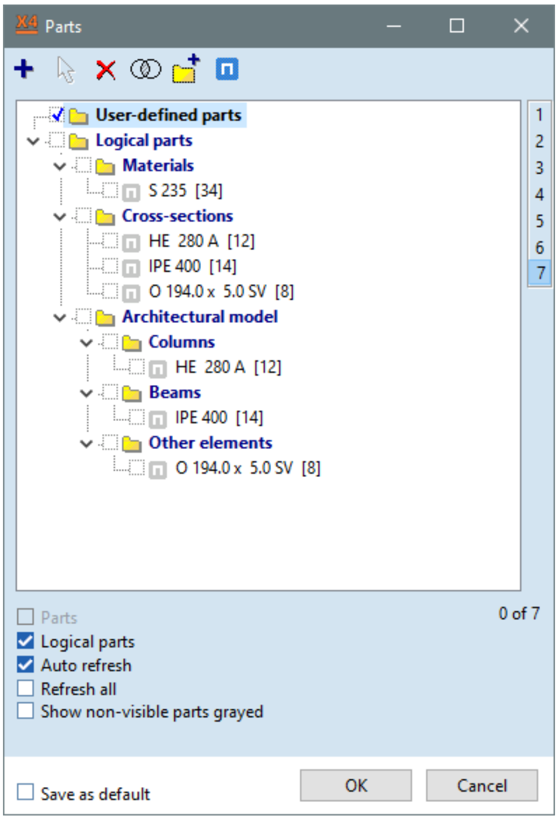
The following will be displayed:



Parts

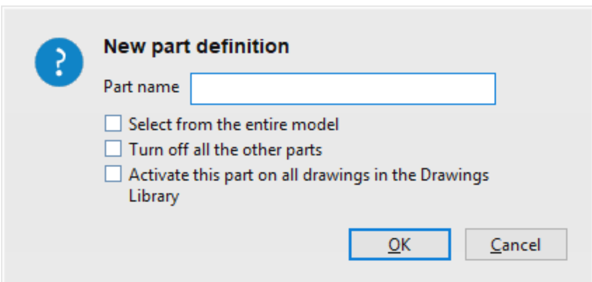
Let us define parts to see only specified parts or elements of the entire model.

By clicking on ***Parts*** icon shows the following window:

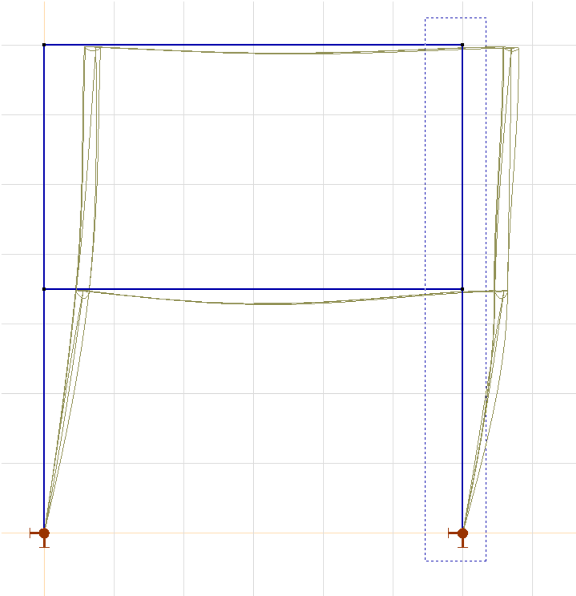


New

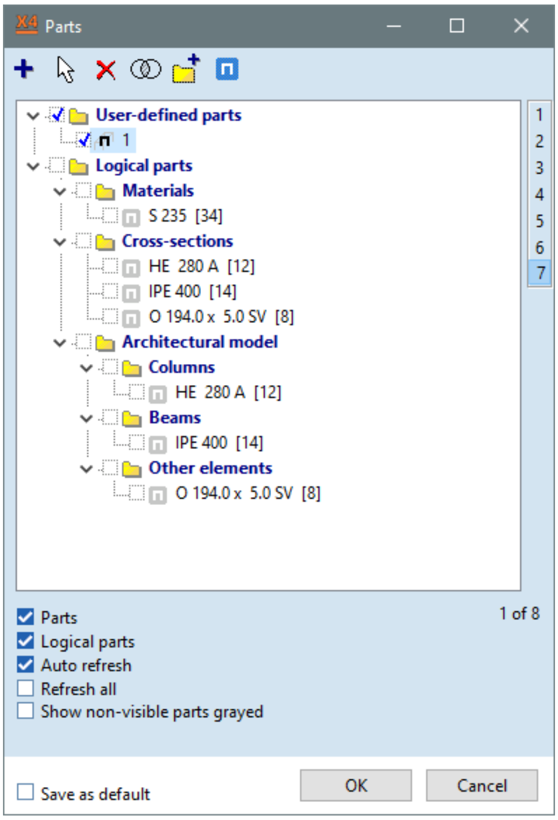
Click on ***New*** icon and the following window enter ***1*** to the name data field, then close with ***OK***.



Select elements of the frame which will be in part name ***1***. The next figure shows how to select beam elements on the right side with selection rectangle.

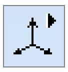


Finish selection with ***OK***, then the defined part is added to the list in the window:

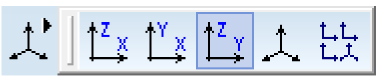


By closing window with ***OK***, then the part with name ***1*** is created.

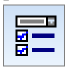
Views



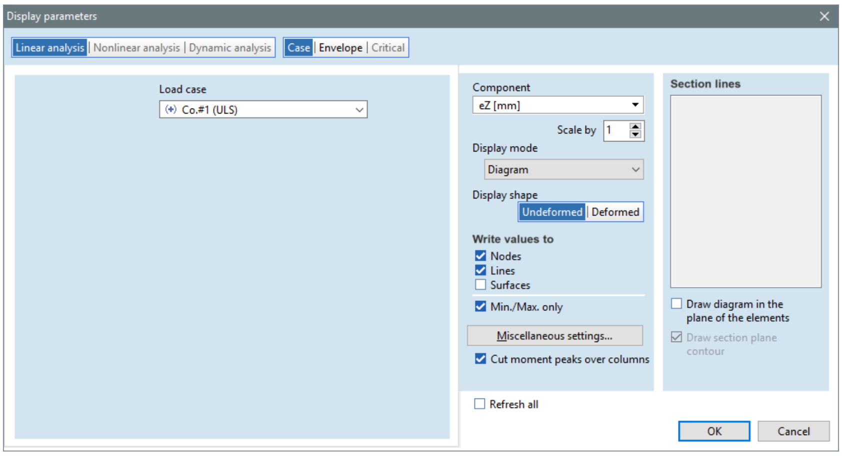
Change view ***Y-Z*** side view!



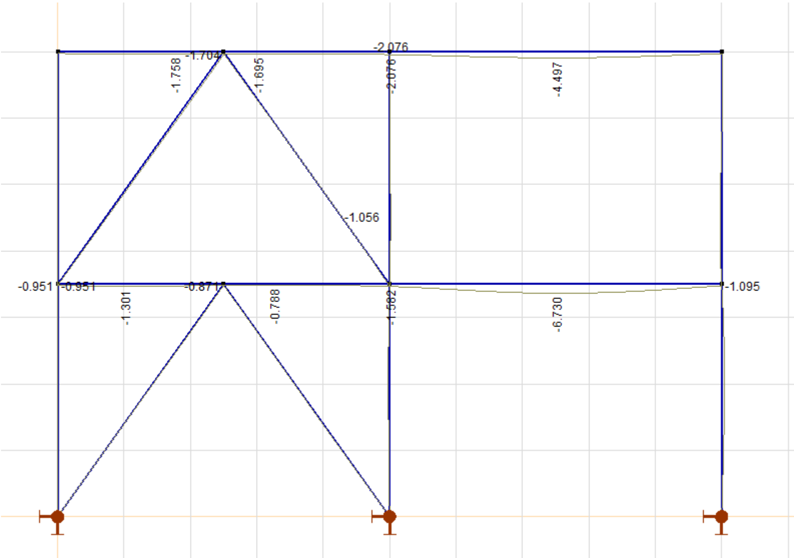
Result display parameters



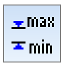
Click on ***Result display parameters*** icon, then check option ***Write values to*** - ***Lines*** checkbox in the window:



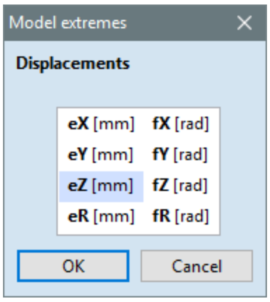
Close window with ***OK***, the min./max. results are displayed on the line elements:



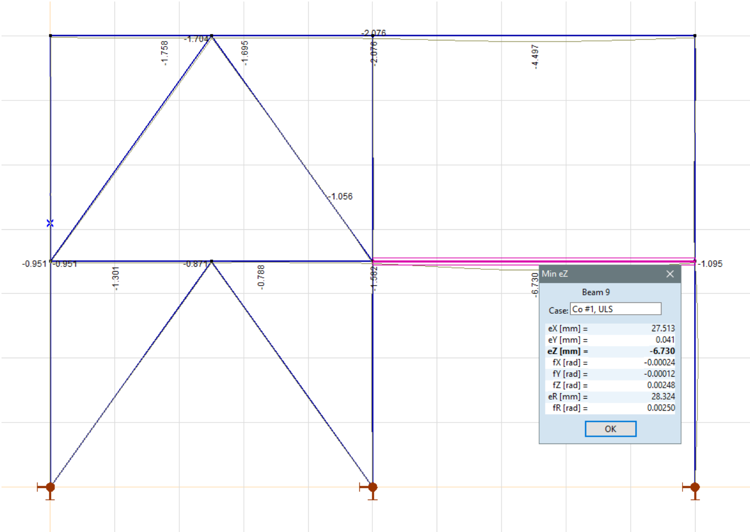
Min, max values



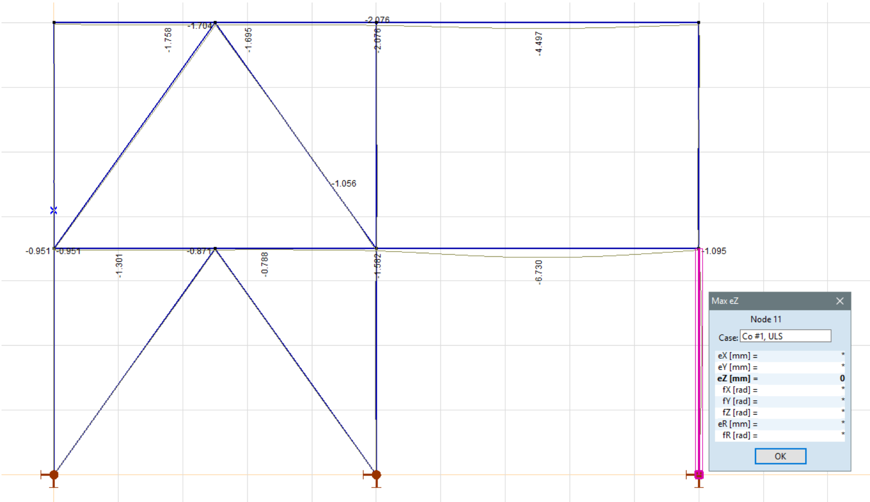
To find location of maximum deformation use ***Min, Max values*** function. Click on icon and select one of the deformation components in the following window:



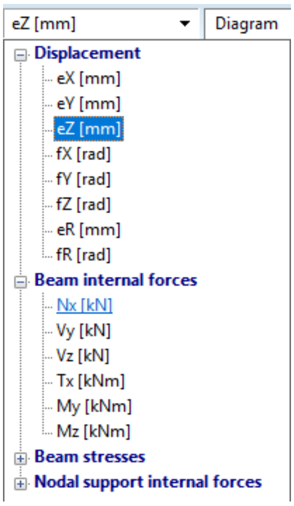
Confirm with ***OK***, the software shows the maximum negative value and its location as well:



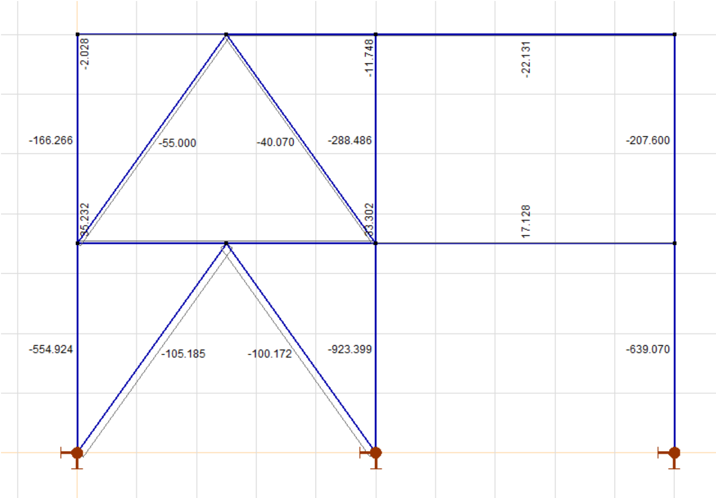
After click on ***OK***, the result panel jumps to the maximum positive value showing its location.



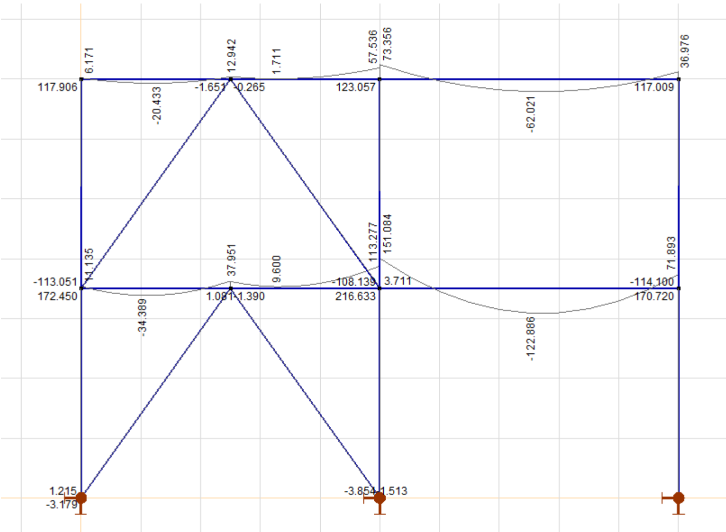
Select ***Beam internal forces*** – ***Nx [kN]*** component from the drop-down list among result components.



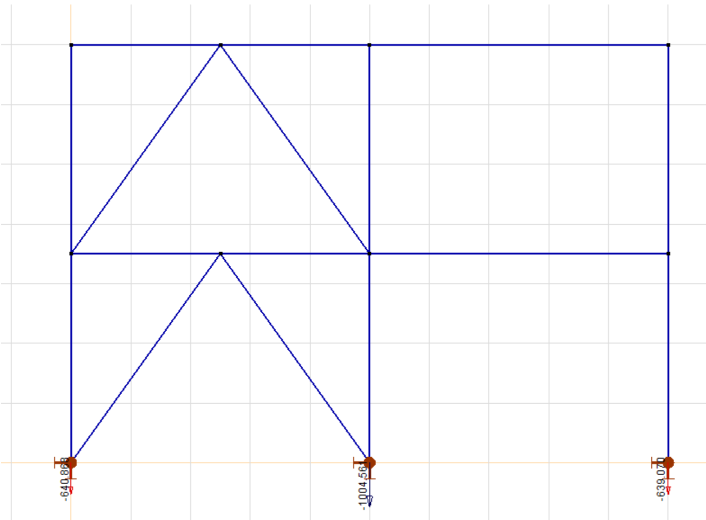
The next figure shows the results for ***Nx*** component:



Change result component to ***My*** to see the bending moment diagram:

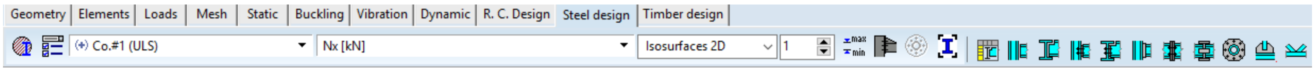


Finally select ***Rz*** component among ***Nodal support internal forces***:

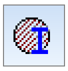


Steel design

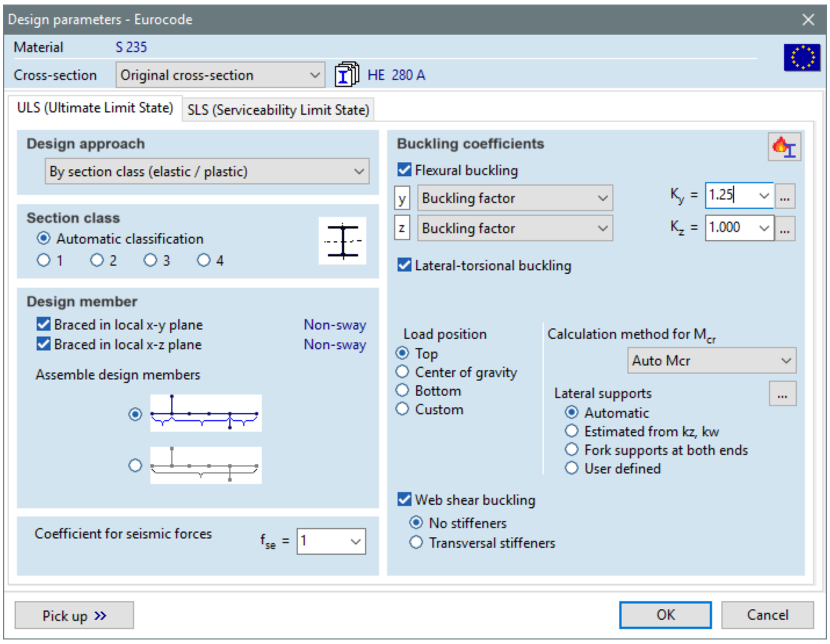
Change to ***Steel design*** tab to check the corner columns and optimize them.



Design parameters



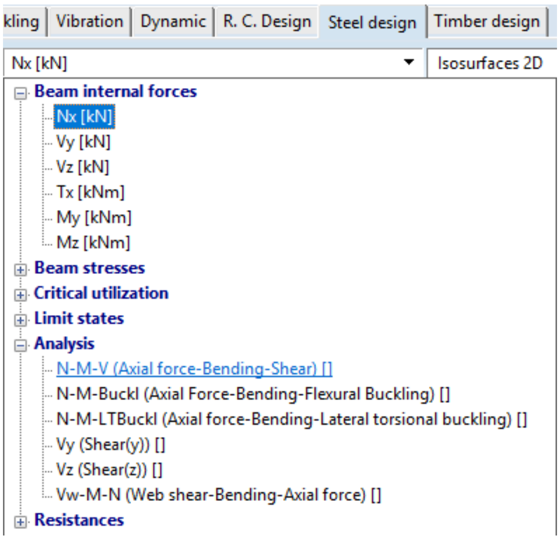
Firstly, design parameters should be assigned to the corner columns. Click on ***Design parameters*** icon, then select all the columns in the corners. Confirm selection with ***OK*** and specify the followings in the ***Design parameters window***:



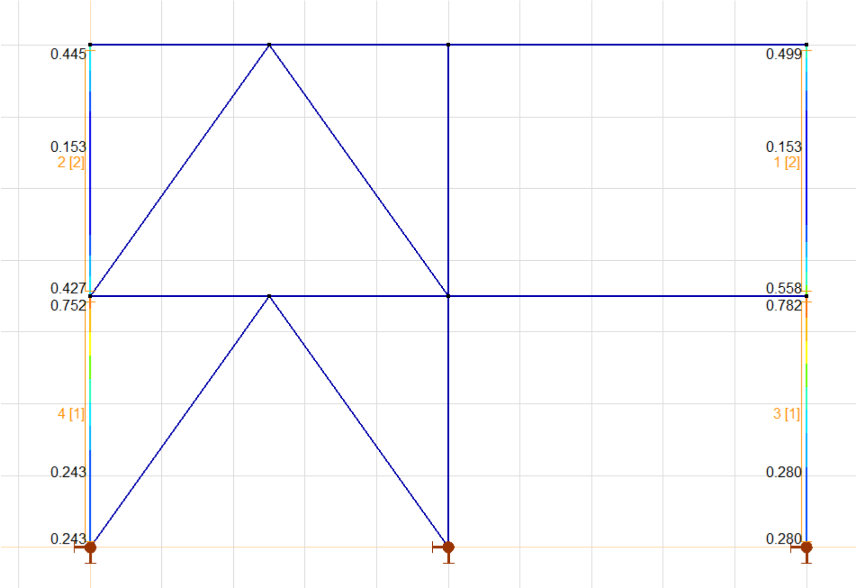
Set K_y to ***1,25***-re, and select ***Auto Mcr*** method to calculate M_{cr}, and close window with ***OK***.

N-M-V Axial force-Bending-Shear

Select ***Analysis*** - ***N-M-V (Axial force-Bending-Shear)*** results from listbox among toolbar:

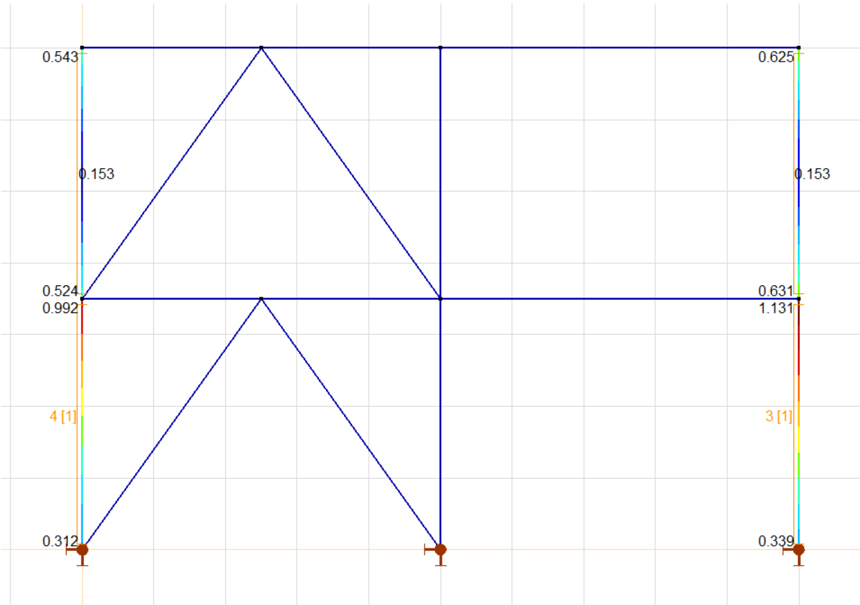


The next figure shows the result for the selected analysis:



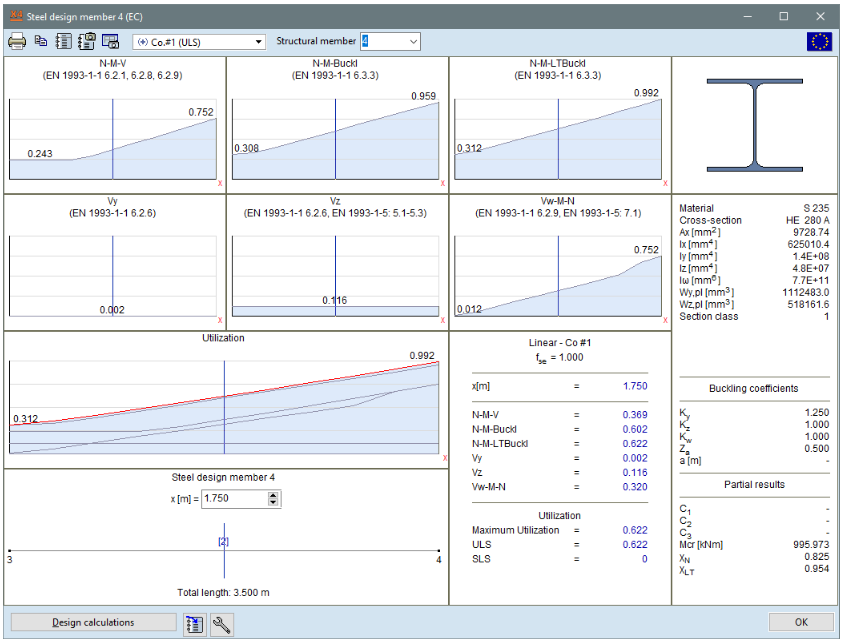
Utilization

Select ***Limit states*** – ***Utilization ULS [ ]*** from the same listbox. The following diagram will be displayed:



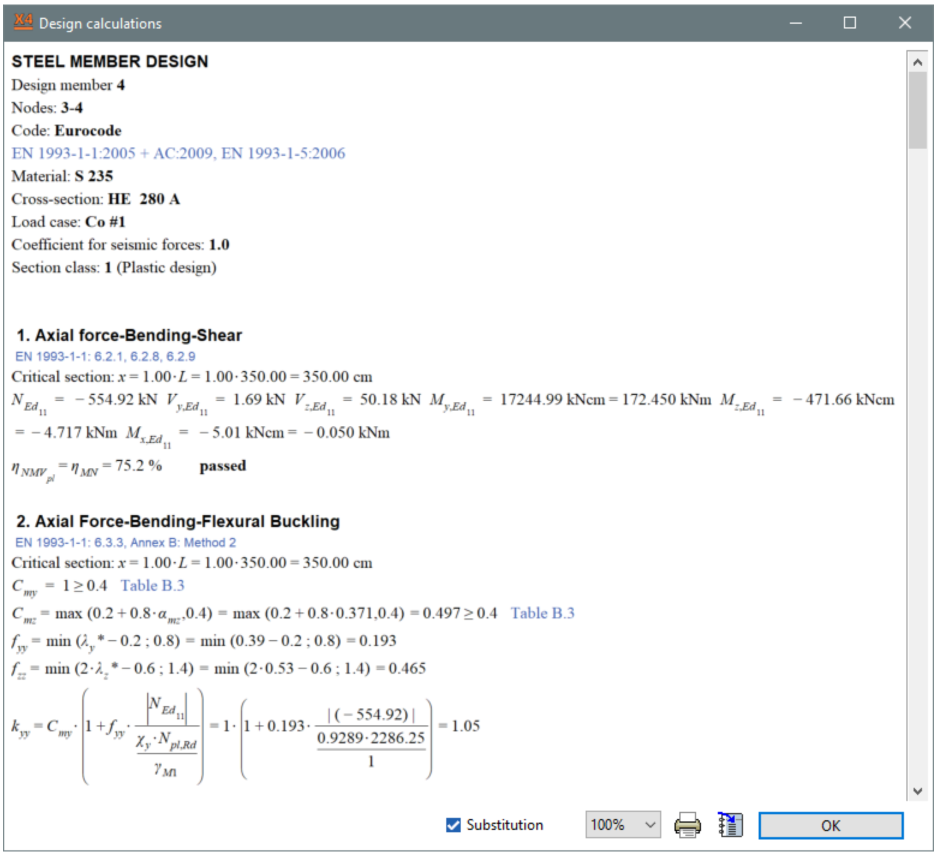
The lower right column is not safe for the load combination, because the maximum utilization (***1.131***) exceeds value ***1***. The design error will be corrected by using ***Optimization*** function.

Click on column ***A1*** and see the results of all checks:



Design calculations

Click on ***Design calculations*** button to see detailed calculations which can be printed or added to the ***Documentation*** as well:


Design calculations

STEEL MEMBER DESIGN
Design member **4**
Nodes: **3-4**
Code: **Eurocode**
EN 1993-1-1:2005 + AC:2009, EN 1993-1-5:2006
Material: **S 235**
Cross-section: **HE 280 A**
Load case: **Co #1**
Coefficient for seismic forces: **1.0**
Section class: **1** (Plastic design)

1. Axial force-Bending-Shear
EN 1993-1-1: 6.2.1, 6.2.8, 6.2.9
Critical section: $x = 1.00 \cdot L = 1.00 \cdot 350.00 = 350.00$ cm
$N_{Ed_{11}} = -554.92$ kN $V_{y,Ed_{11}} = 1.69$ kN $V_{z,Ed_{11}} = 50.18$ kN $M_{y,Ed_{11}} = 17244.99$ kNcm $= 172.450$ kNm $M_{z,Ed_{11}} = -471.66$ kNcm $= -4.717$ kNm $M_{x,Ed_{11}} = -5.01$ kNcm $= -0.050$ kNm
$\eta_{NMV_{pl}} = \eta_{MN} = 75.2\ \%$ **passed**

2. Axial Force-Bending-Flexural Buckling
EN 1993-1-1: 6.3.3, Annex B: Method 2
Critical section: $x = 1.00 \cdot L = 1.00 \cdot 350.00 = 350.00$ cm
$C_{my} = 1 \geq 0.4$ Table B.3
$C_{mz} = \max(0.2 + 0.8 \cdot \alpha_{mz}, 0.4) = \max(0.2 + 0.8 \cdot 0.371, 0.4) = 0.497 \geq 0.4$ Table B.3
$f_{yy} = \min(\lambda_y{}^* - 0.2\ ;\ 0.8) = \min(0.39 - 0.2\ ;\ 0.8) = 0.193$
$f_{zz} = \min(2 \cdot \lambda_z{}^* - 0.6\ ;\ 1.4) = \min(2 \cdot 0.53 - 0.6\ ;\ 1.4) = 0.465$
$$k_{yy} = C_{my} \cdot \left(1 + f_{yy} \cdot \frac{|N_{Ed_{11}}|}{\frac{\chi_y \cdot N_{pl,Rd}}{\gamma_{M1}}}\right) = 1 \cdot \left(1 + 0.193 \cdot \frac{|(-554.92)|}{\frac{0.9289 \cdot 2286.25}{1}}\right) = 1.05$$

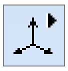
Substitution 100% OK

Close report with ***OK***, and click again on ***OK*** to close ***Steel Design member*** window.

Parts

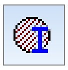
To view the full model, click on ***Parts*** icon in the vertical toolbar on the left side. In the window that appears, uncheck ***User-defined parts***, and close window with ***OK***.

Views

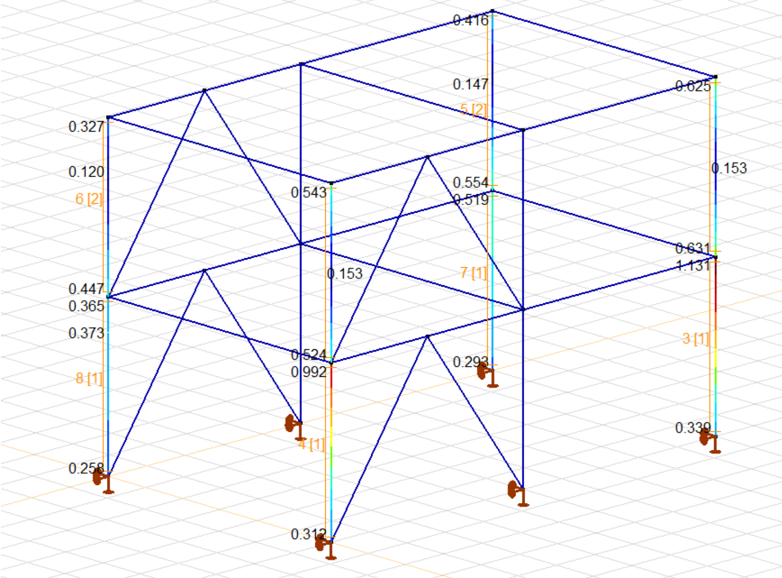


Change view to ***Perspective***!

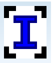
Design parameters



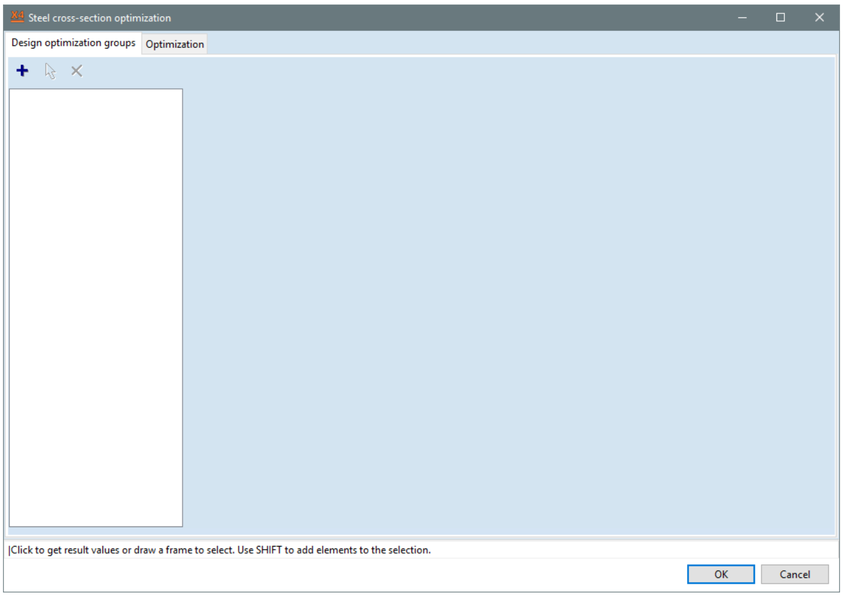
To specify design parameters for the corners columns behind, click on ***Design Parameters*** icon and select the four columns (2 on top, 2 on bottom). Confirm selection with ***OK*** and in the ***Design parameters*** window, and click on ***Pick up*** button. Point to column ***A1*** by clicking on it, then the software picks up the set values. Close window with ***OK***. The following will be displayed:



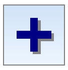
Steel cross-section optimization



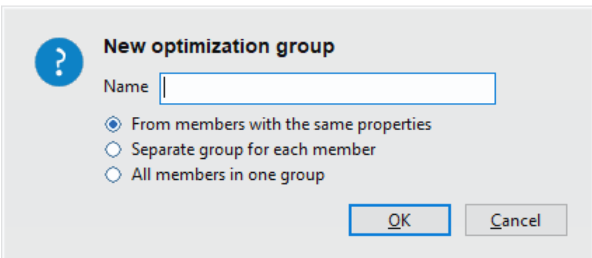
Optimize the cross-section of the examined corner columns and correct design error mentioned above. Click on ***Steel cross-section optimization***, the next window shows up:



New

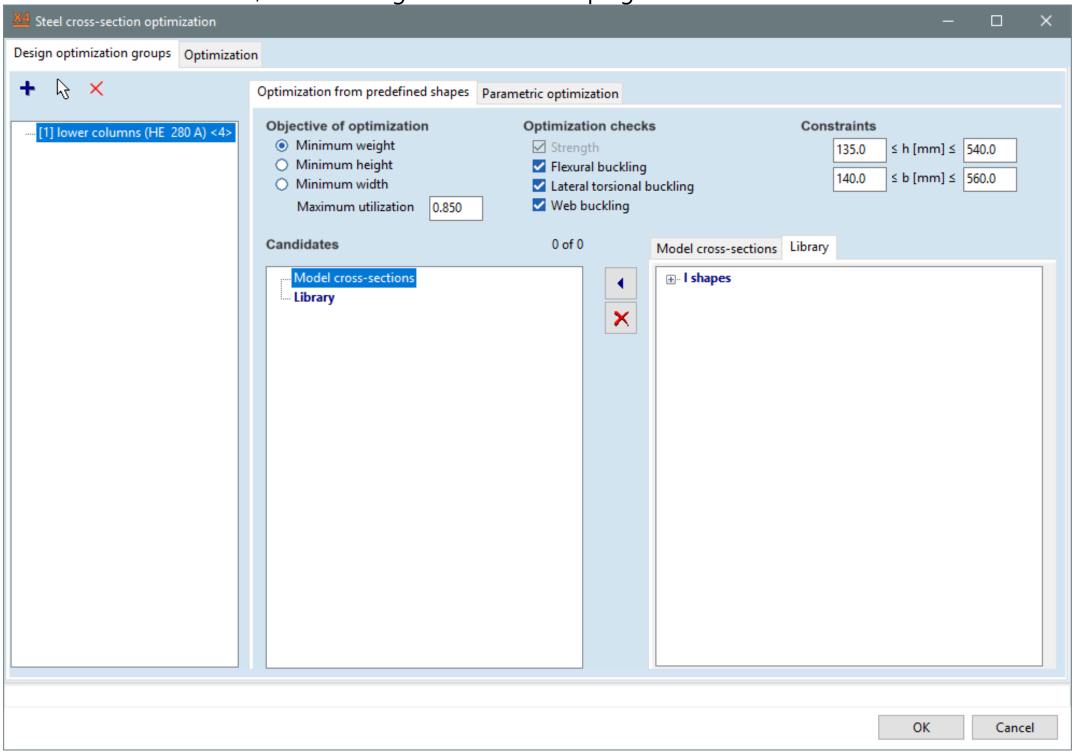


Click on ***New*** icon to specify ***New optimization group*** and set the followings in the window:



For the group of lower columns, enter the Name '***lower columns***' and select ***All members in one group***. Close with ***OK*** and select 4 lower columns in the corners.

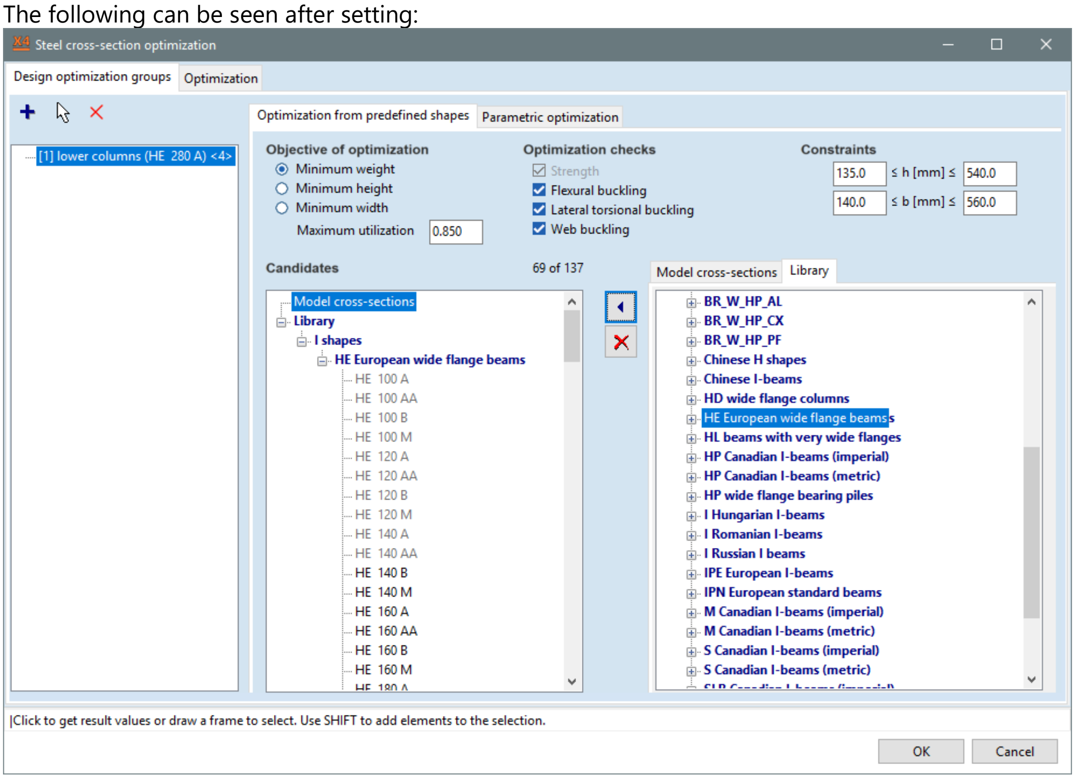
Finish selection with ***OK***, the following window shows up again:



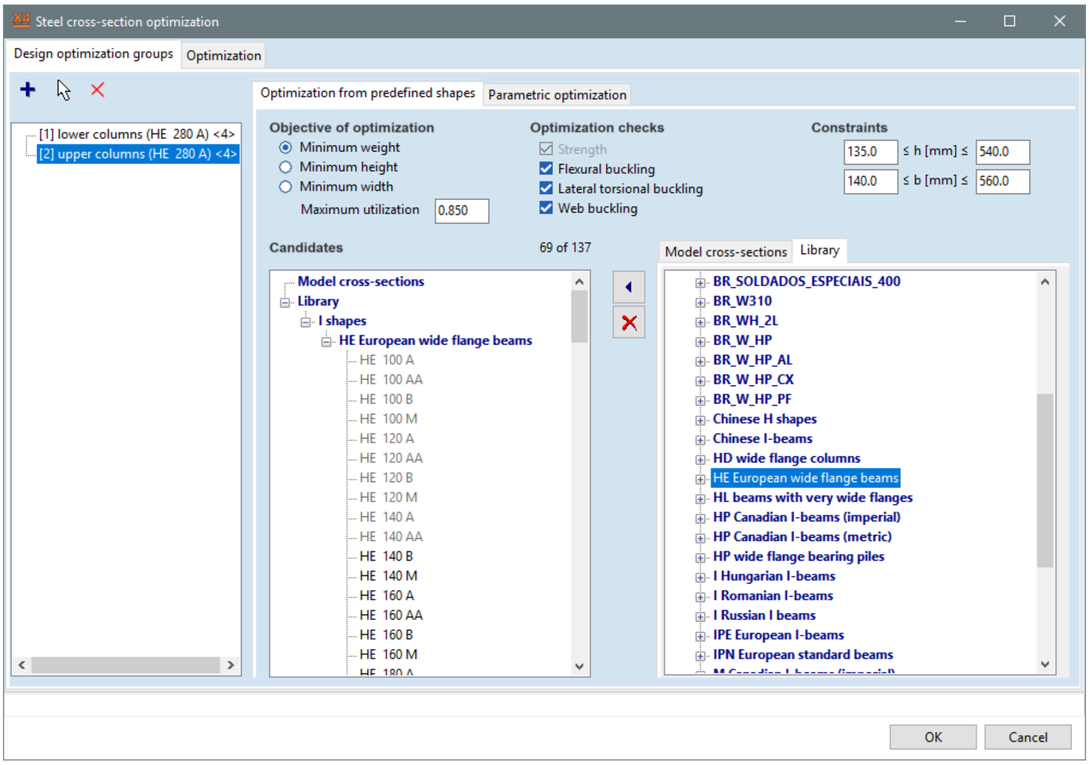
Library

Click on ***Library*** tab and select cross-section groups, and add to the list of ***Candidates*** for optimization. By clicking on the '**+**' sign before label ***I-shapes***, the list of ***I shapes*** opens. From the list select ***HE European wide flange beams***, and add to list on the left by clicking on the blue arrow between the two listboxes.

The following can be seen after setting:



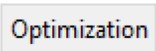
Also create an optimization group for upper corner columns with name '***upper columns***' and add ***HE European wide flange beams*** shape group to the list of ***Candidates***. The following will be the result:



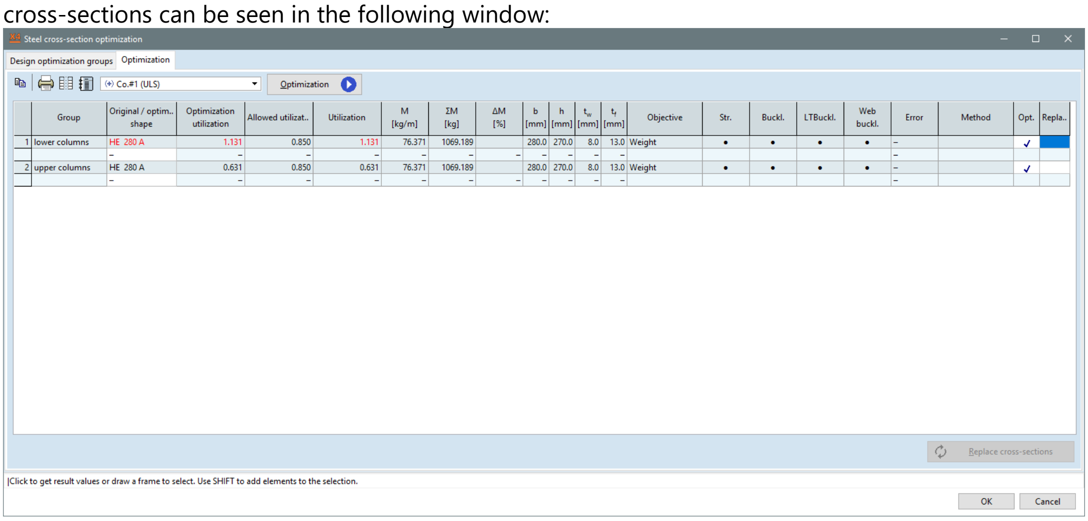
The ***Objective of optimization*** can be ***Minimum weight***, ***Minimum height*** or ***Minimum width*** and ***Maximum utilization*** can also be set. It is possible to ignore certain checks during the optimization process. All strength checks are always performed but checks for ***Flexural buckling***, ***Lateral torsional buckling*** and ***Web buckling*** can be deactivated if necessary. These settings must be set for all the optimization groups individually. In our example, select ***Minimum weight*** for optimization and set ***Maximum utilization*** to ***0.85***.

Remark: optimization calculation is based only on the results of previously run static analysis. If cross-sections are replaced in the model as a result of the optimization process, the internal forces may change (can grow) because of the different stiffness parameters. Therefore, a new static calculation is recommended to check the design members again.

Optimization

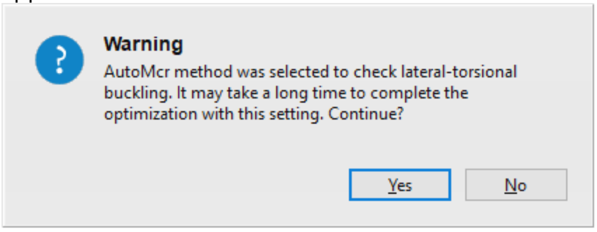


By clicking on ***Optimization*** button, the specified optimization groups and present utilizations of the cross-sections can be seen in the following window:

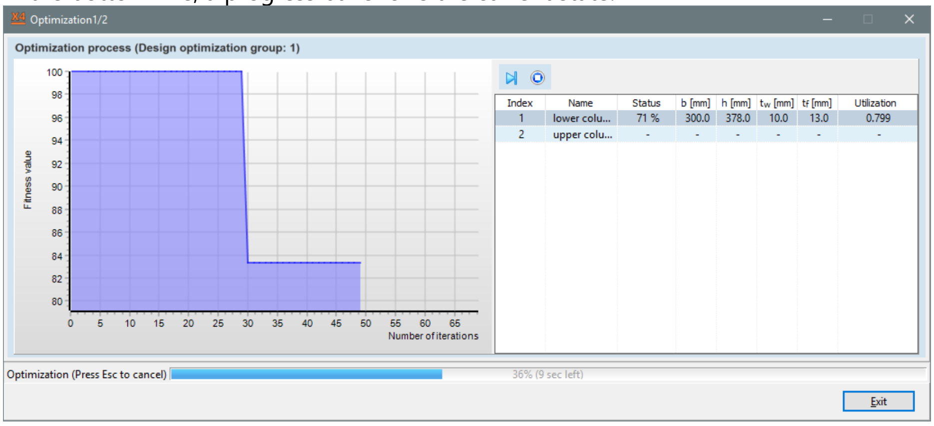


Optimization

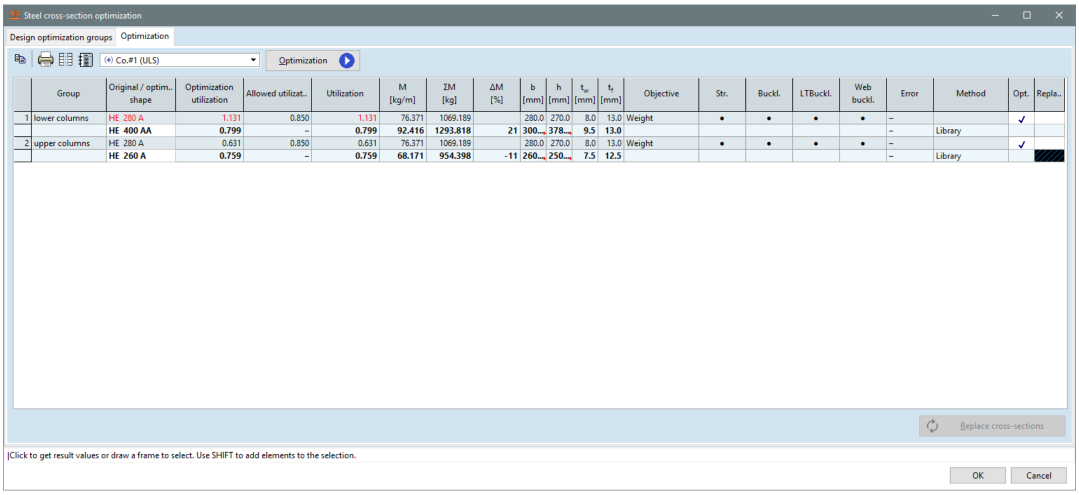
Click on ***Optimization*** button, then the software starts calculation procedure. A warning message will appear, click on ***OK*** to continue.



In the bottom line, a progress bar shows the current state.

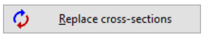


When the optimization process is complete, as a result the suggested cross-section is displayed under the row of the current cross-section:

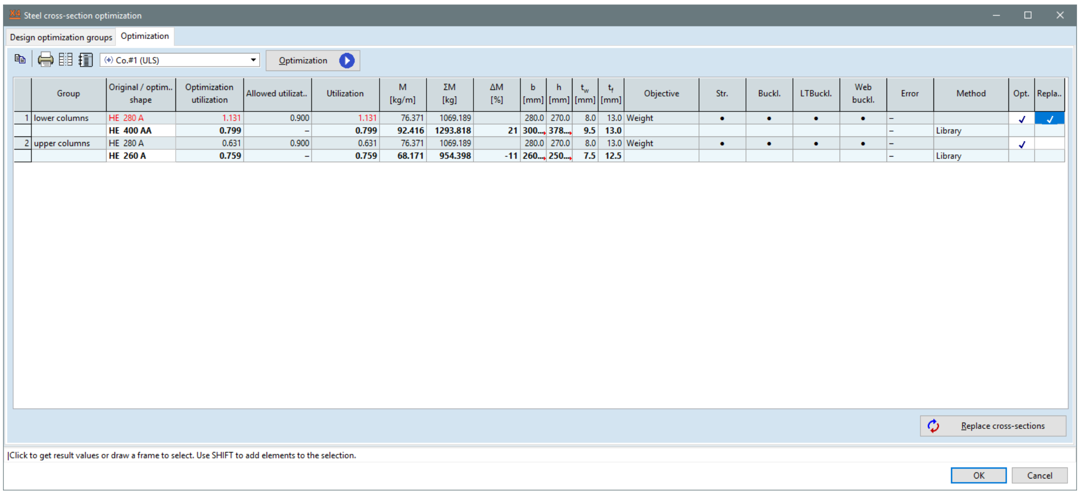


Considering the specified parameters for optimization, the software suggests to apply shape **HE 260 A** for upper columns, and **HE 400 AA** for lower columns.

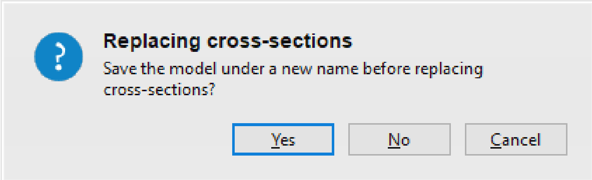
Replace cross-sections



In our example, only the lower columns will be replaced, the cross-section of the upper columns will be kept. To replace cross-section, click on ***Replace*** field in the row of lower columns, then ***Replace cross-section*** icon in the lower right corner becomes active:

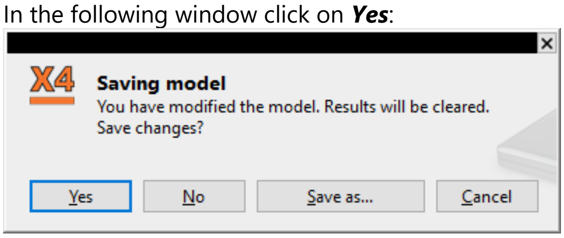


Activate ***Replacing cross-section*** icon, then software offers to save the model under a new name before replacing cross-sections:



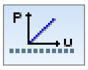
Click on ***No***, keep the original file name.

In the following window click on ***Yes***:



Changing the model by replacing cross-sections, the model must be saved and previous results of static analysis will be deleted.

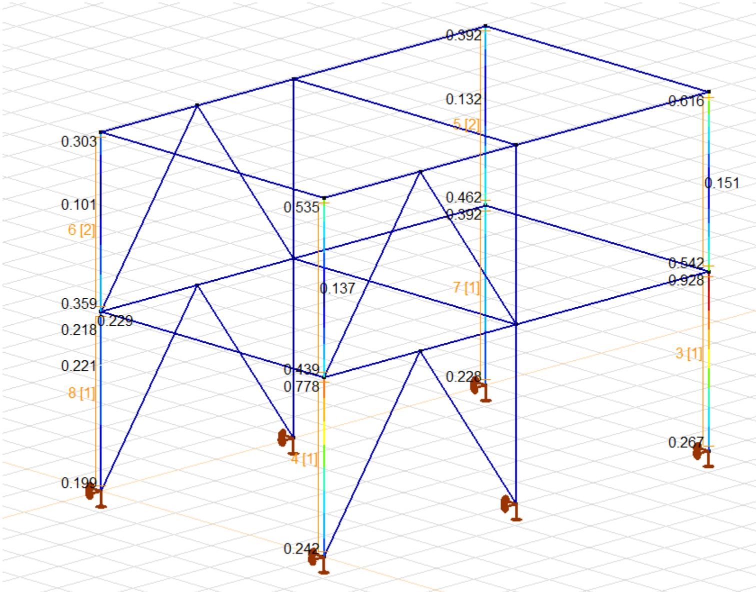
Linear static analysis



On ***Static*** tab, run a new ***Linear static analysis***.

Steel design

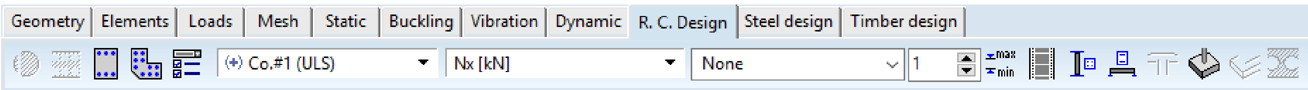
Let us go back to ***Steel design*** tab and check utilization of the design members. Every member is safe after the strengthening, the optimization can be finished:



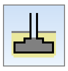
In the figure can be seen, that the maximum utilization of the critical member in the right corner is ***0,928*** (92,8%) after modification. This result is greater than the value that was set for optimization. This increase is due to the change in internal forces.

R. C. design

Design pad footing of column ***A1***, change to ***R. C. design*** tab:



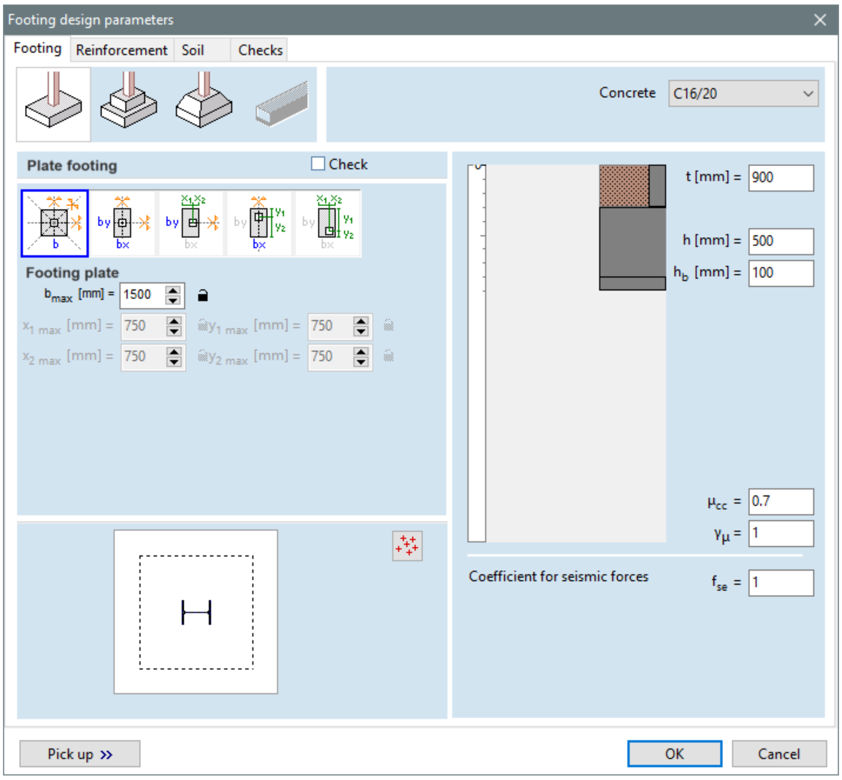
Pad footing design



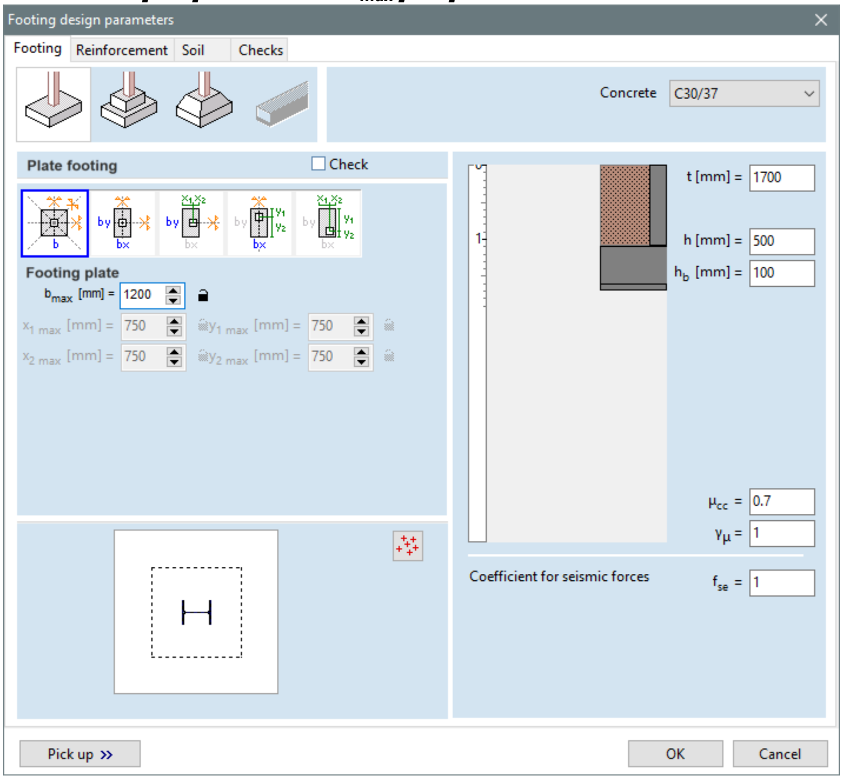
Click on ***Pad footing design*** icon and select the nodal support of column ***A1***. Confirm selection with ***OK***.

Footing design parameters

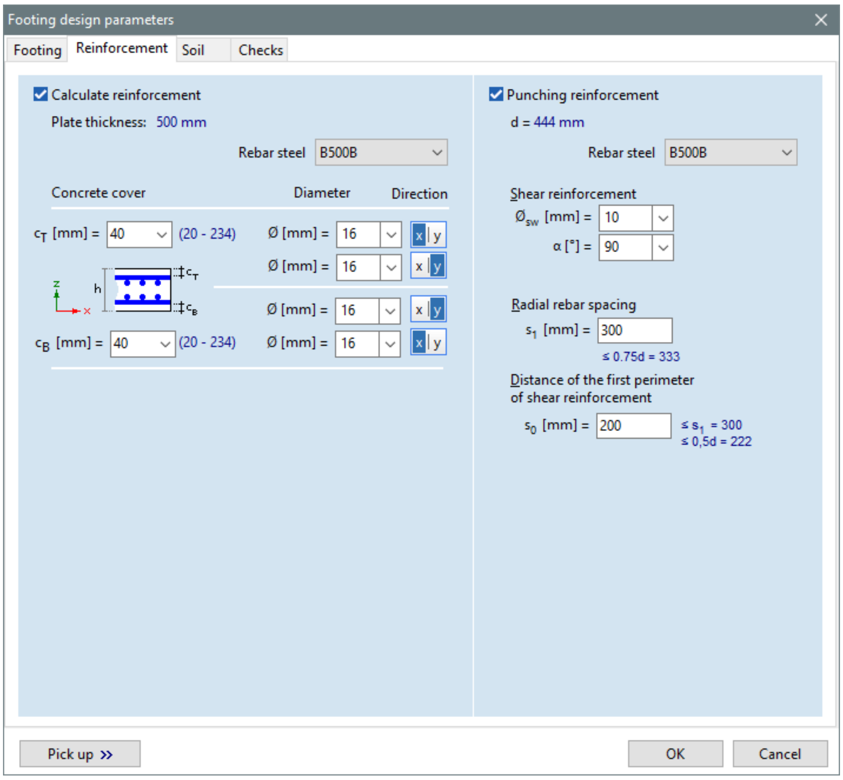
Then ***Footing design parameters*** window shows up:



In the parameters window select ***quadratic*** pad footing, set concrete ***Concrete*** strength to ***C30/37***, ***soil cover*** - ***t [mm]*** to ***1700*** and ***b_{max} [mm]*** to ***1200***:

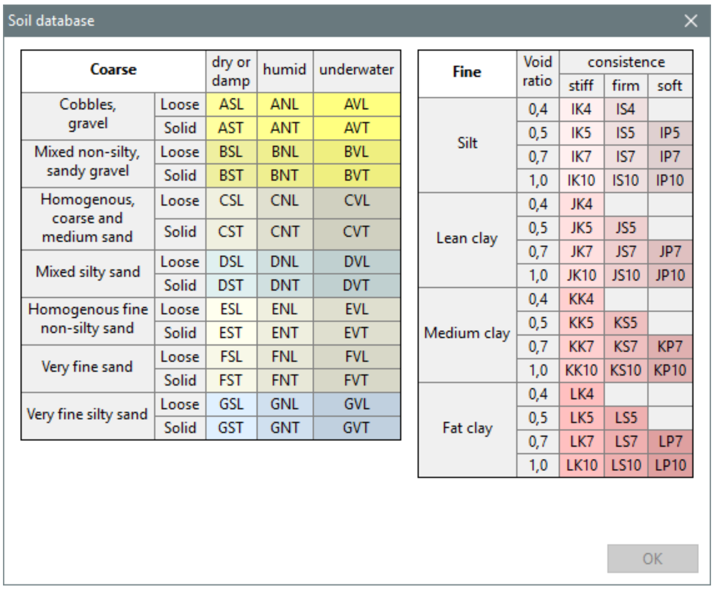


On reinforcement tab check ***Calculate reinforcement*** and ***Punching reinforcement*** function, and set reinforcement parameters as shown below:



Soil database

On ***Soil*** tab, click on ***Soil database*** icon to select predefined soil characteristics, then the following window shows up:



Select type ***CST*** (***Homogenous, coarse and medium sand,*** in ***dry or damp*** column and ***Solid*** row), then parameters of the selected soil are shown in bottom left corner:

CST	γ [kg/m^3]	φ [°]	φ_T [°]	φ_{ZS} [°]	E_0 [N/mm^2]	μ []
	2000	35.00	32.00	27.00	63.00	0.20
Solid, dry sand						

Confirm selection with ***OK***.

Layer thickness

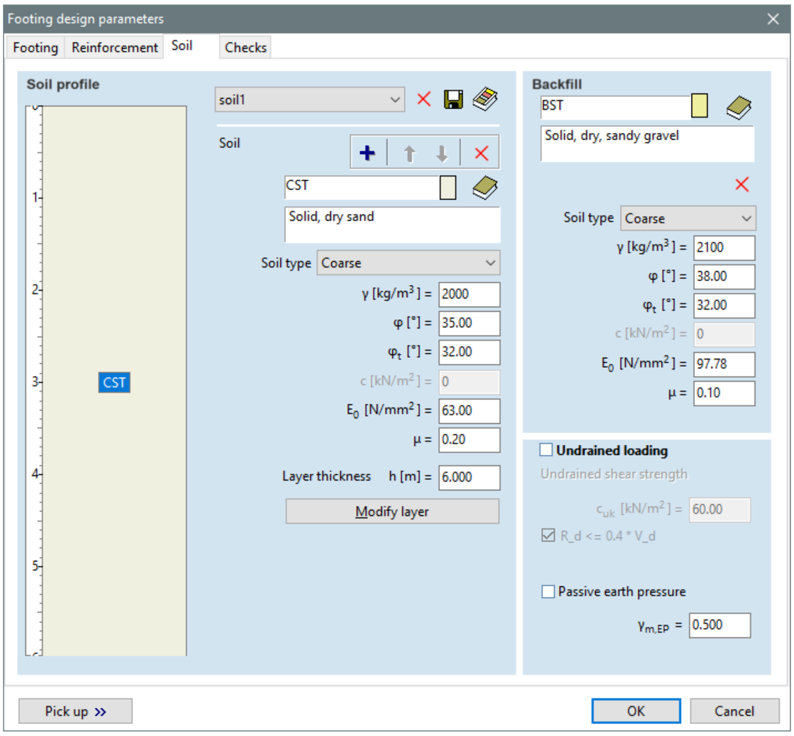
Enter ***6,0*** m in ***Layer thickness*** field.

Add new soil layer

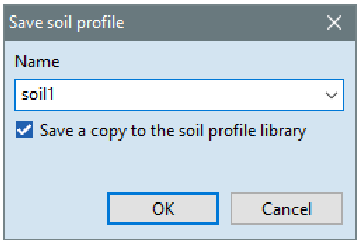
Add defined soil layer to the ***Soil profile*** with **+** icon.

Soil database

Click on ***Soil database*** icon in ***Backfill*** options to define soil type for soil fill above pad's bottom level. Select ***BST*** (***Mixed Non-silty, sandy gravel*** in ***dry or damp*** column and ***Solid*** row). Then confirm with ***OK***. The following window will be displayed:

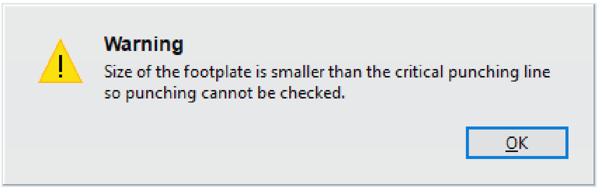


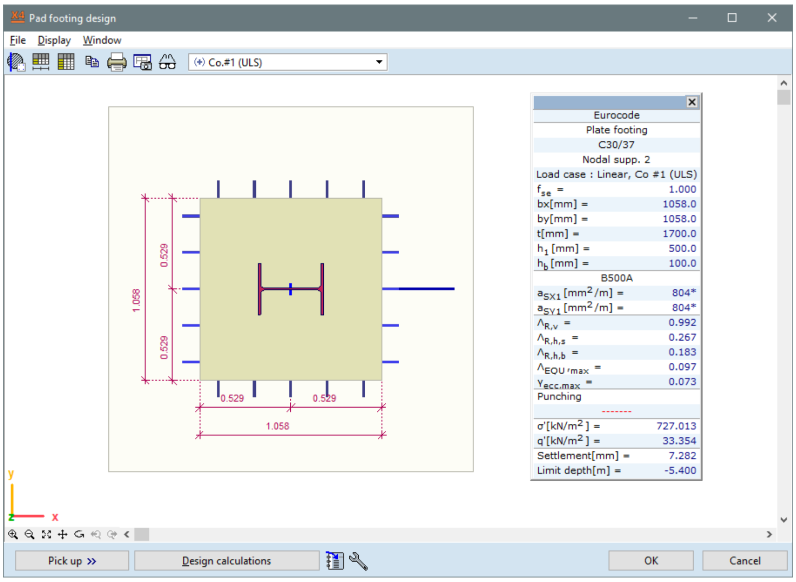
After click on ***OK*** and enter the ***Name*** of the specified soil profile:



Type in name '***soil1***' and save profile with ***OK***. Program calculates the required dimensions and reinforcement of pad footing.

The next warning message shows up, close it with ***OK***.

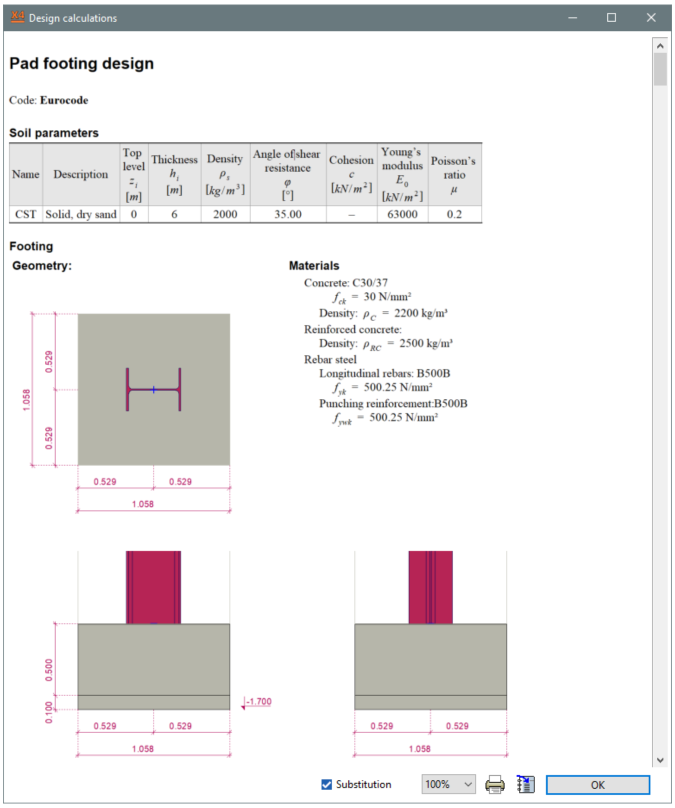




The required dimensions and reinforcement are shown in plain view and in ***Info window***. The view can be rotated as usual or a perspective view can be requested, if necessary (see ***Display*** menu). By clicking on ***Settings*** icon, the desired display parameters can be set.

Design calculations

By clicking on ***Design calculations*** button, the software presents a detailed design calculation report:

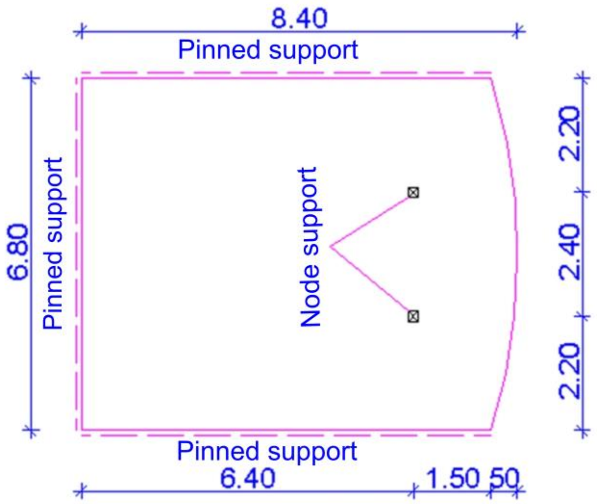


Click on ***OK***, to close window and function.

3. SLAB MODEL

Objective

Calculate maximum bending moment, the required reinforcement of the slab shown below. Define actual reinforcement and calculate the maximum deflection by nonlinear analysis.

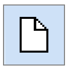


The thickness of the slab is 200 mm, concrete strength is C25/30, the rebar is B500A type. Use Eurocode 2 for design.

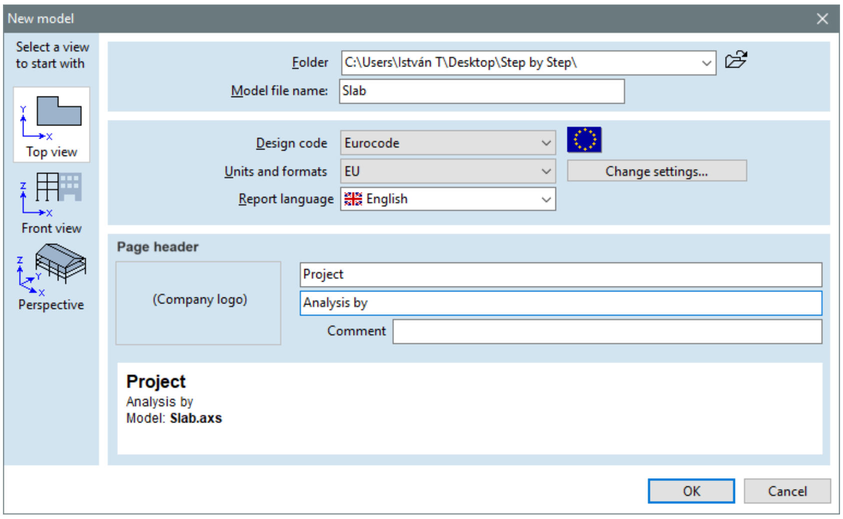
Start

Start ***AxisVMX4*** by double-clicking on ***AxisVMX4*** icon in its installation folder, found in ***Start*** – ***Programs*** menu.

New



Create a new model by clicking on ***New*** icon. In the dialogue window replace ***Model file name*** with '***Slab***', select ***Eurocode*** from ***Design code***s and set ***Unit and formats*** to ***EU***.

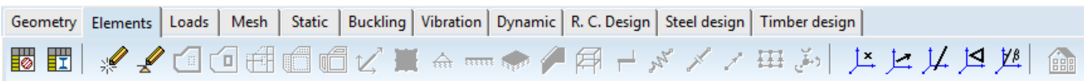


Starting workplane also can be set on the left in this window. Change workplane to ***X-Y Top view***. Our slab will be parallel to this plane, and the gravity load acts in ***-Z*** direction. This setting can also be done using ***Choosing view*** icon on the editing interface.
Click ***OK*** to close the dialog window.
The geometry of the slab will be created by using of editing tools.

Define of geometry -

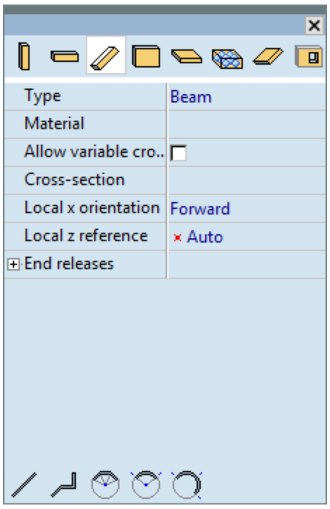
Select the ***Elements*** tab to bring up ***Elements*** toolbar.

Elements

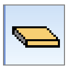


Draw objects directly

By clicking on ***Draw objects directly*** icon shows following window:

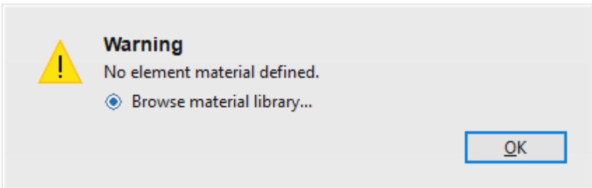


Slab



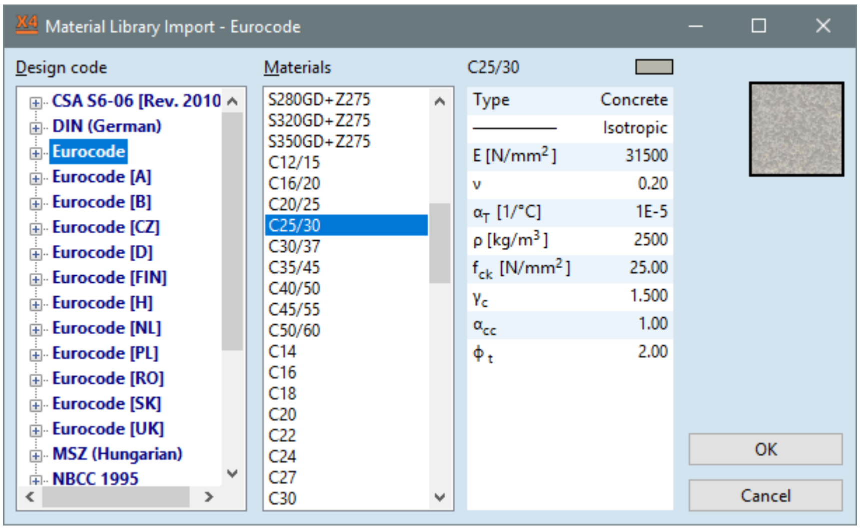
Change element type to ***Slab***. Click on this icon even it is already selected because of the order of following steps.

The following window shows after clicking:



Material library import

Click on ***OK*** to ***Browse cross-section libraries...***:



Roll down in the list of ***Materials*** by using vertical sliding bar (or roll mouse wheel) and click on ***C25/30***, then click on ***OK***.

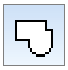
Type

Change object ***Type*** to ***Plate***.

Thickness

Set ***Thickness [mm]*** to ***200***.

Complex slab



Click on ***Complex slab*** icon. To define the geometry, user can draw the slab directly or define the coordinates. Firstly, use global coordinates (when ***d*** button is not pressed in ***Coordinates*** palette).

Define coordinates by the following.
For the first point press keys below:

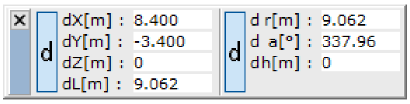
x 0 y 0 z 0 <**Enter**>.

Nodes in Relative coordinate system

To enter additional points, select the relative coordinate system input. User can change coordinate system by pressing the ***d*** button on the ***Coordinates palette***.

If ***d*** button is down (pressed) it denotes relative coordinates. ***X*** turns to ***dX*** and so on indicating that we are defining relative coordinates. The relative point of origin is shown as thick blue ***X*** (cross turned 45°.)

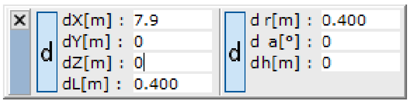
The next figure shows state when ***d*** button is active:



Nodes in Relative coordinate system

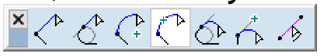
Continue defining the next point with relative coordinates. Press following keys:

x 7.9 y 0 z 0 <**Enter**>.



Arc by three points

Now, click on ***Arc by three points*** icon on the tool palette under ***Draw objects directly*** window:

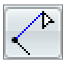


Press following keys to define next contour points:

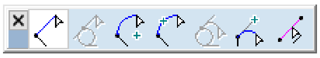
x 0.5 y 3.4 z 0 <**Enter**>

x 0 y 6.8 z 0 <**Enter**>

Line



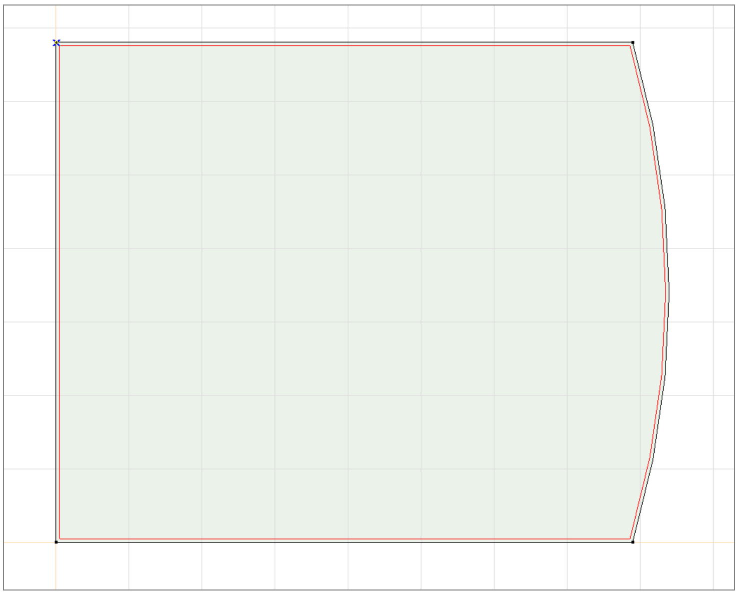
Click on ***Line*** icon on the auxiliary palette:



Press following keys to define next points:

x -7.9 y 0 z 0 <**Enter**>,

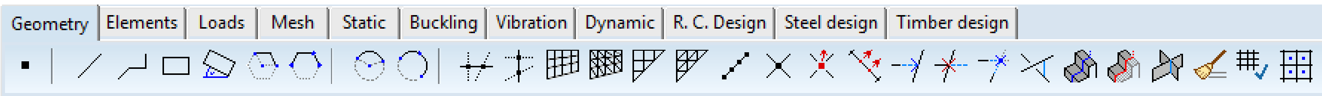
Then press ***Enter*** again to finish (close) contour definition.
Exit from ***Draw objects directly*** by pressing ***Esc***.
The following will be displayed in the main window:



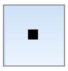
To move the origin of the local coordinate system to the bottom left corner of the slab, move cursor to that position and press ***Insert*** key.

Geometry

Click on ***Geometry*** tab:



Node



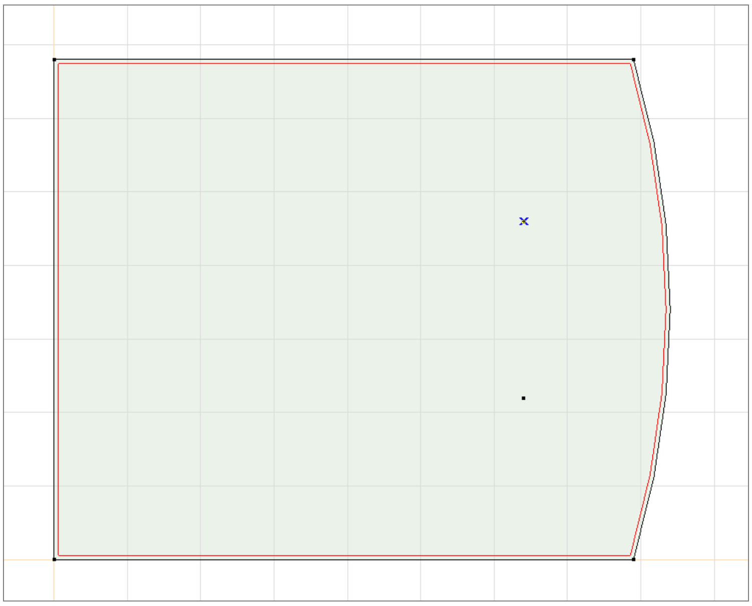
Click on ***Node*** icon to create additional inner points and enter the following coordinates.

x	6.4	y	2.2	<**Enter**>
x	0	y	2.4	<**Enter**>

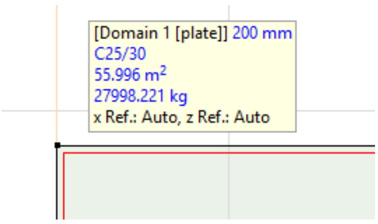
The given coordinates are relatives. After the first node has been created, the relative origin jumps to the specified point.

Press ***Esc*** to close function.

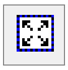
The following will be displayed in the main window:



Please note, the thin red line represents the contour and type of the domain. Moving the cursor over the contour, hint shows up displaying properties of the element:



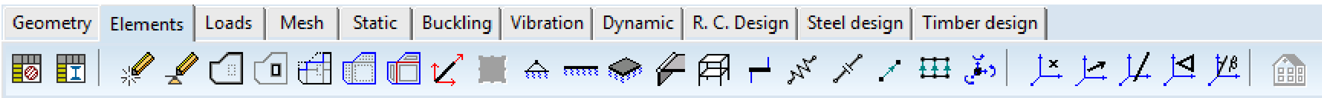
Zoom to fit



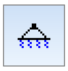
Click on ***Zoom to fit*** for better view.

Elements

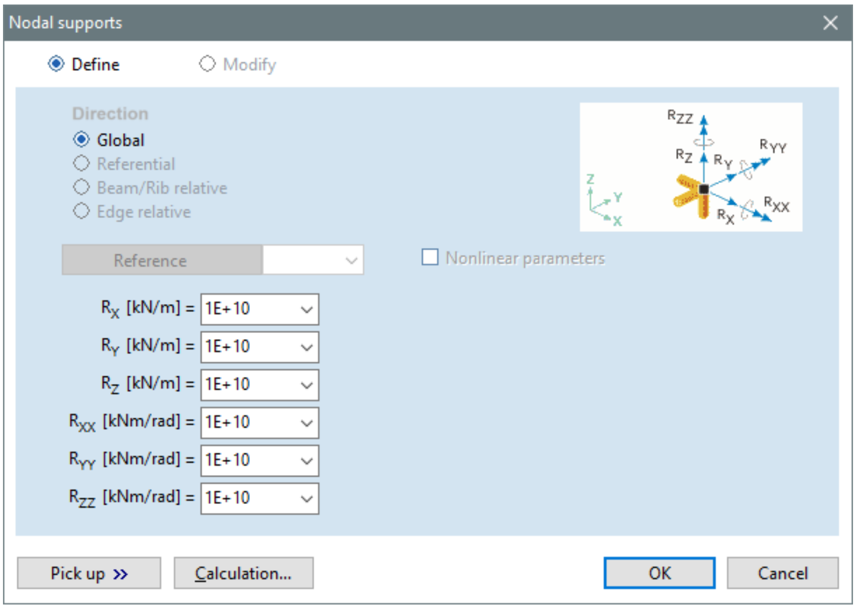
Change to ***Elements*** tab.



Nodal support

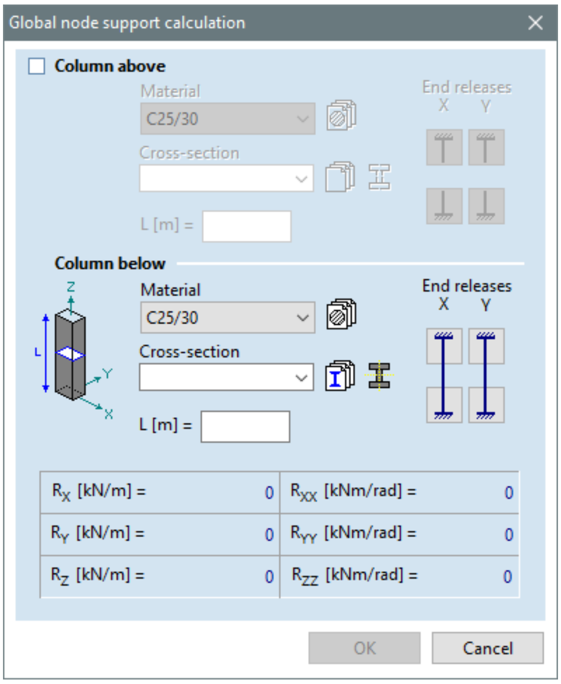


Define nodal supports. Click on ***Nodal support*** icon, then select inner nodes and confirm with ***OK***. The following window shows up:



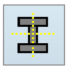
Global node support calculation

Click on ***Calculation...*** button, then the following window shows up:

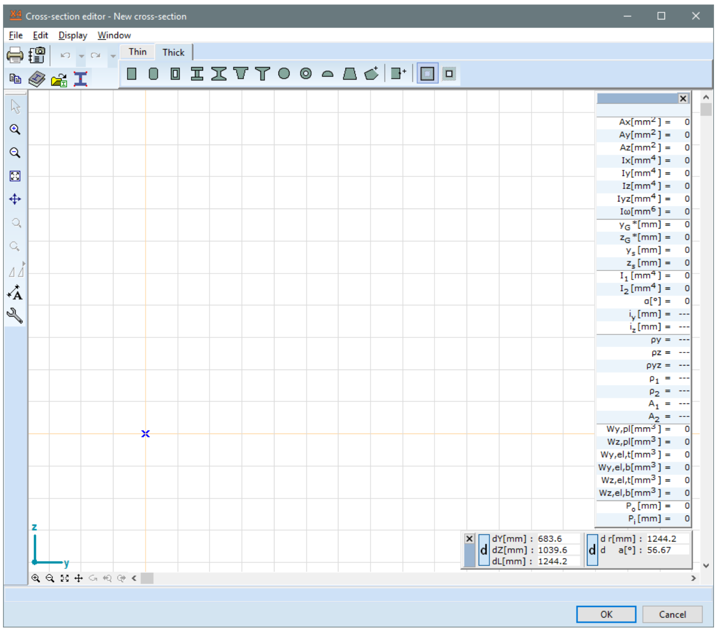


Here, stiffnesses of the support can be automatically calculated if the main parameters are specified for (lower/upper) columns.

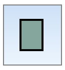
New cross-section



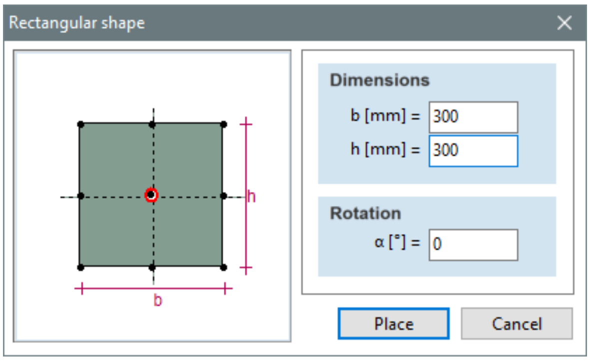
Click on ***New cross-section*** icon, the following window shows up:



Rectangular shape

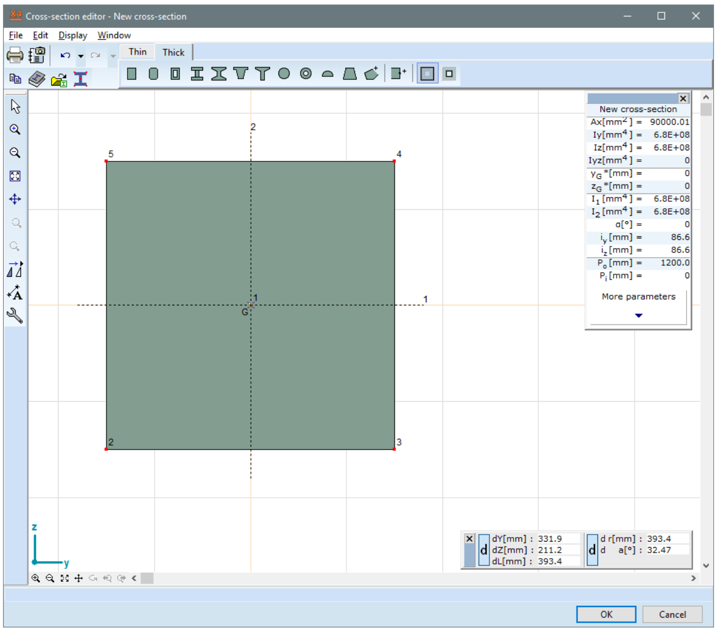


Click on ***Rectangular shape*** icon and fill the required fields in the window:

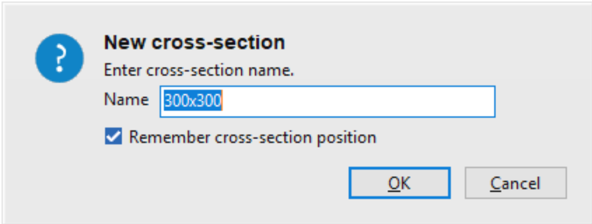


Set ***b*** and ***h*** sizes to ***300*** mm, and click on ***Place*** button. Click anywhere in the window to place the new cross-section.

The following will be displayed:

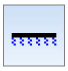


Click on ***OK*** and enter the ***Name*** of the new cross-section to save.

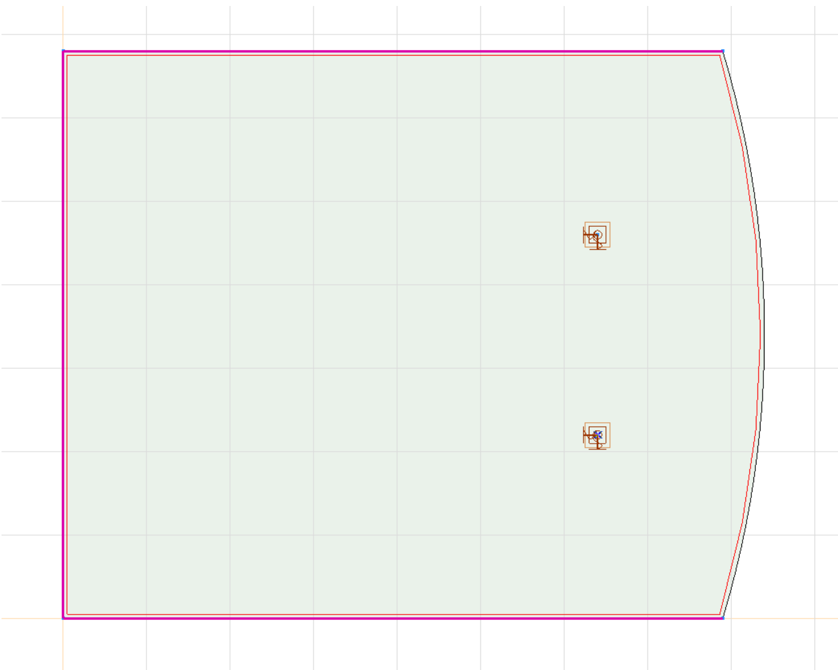


Set ***L [m]*** to ***3***-at, and click on ***OK***. In the ***Nodal support*** window, the stiffness components will be set automatically with the calculated values.
After click on ***OK*** to close.

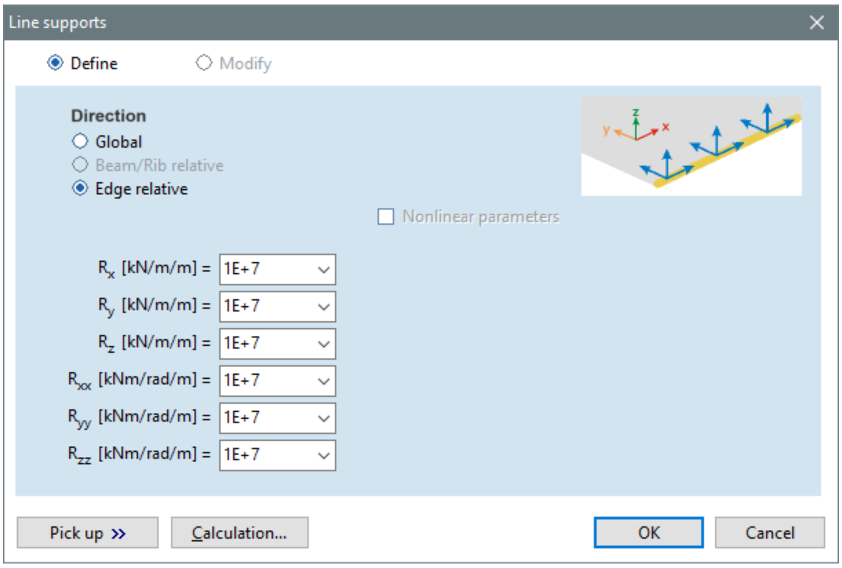
Line support



To define line supports, click on ***Line supports*** icon, then select the following lines on the contour: select the two parallel edges and the one on the left.

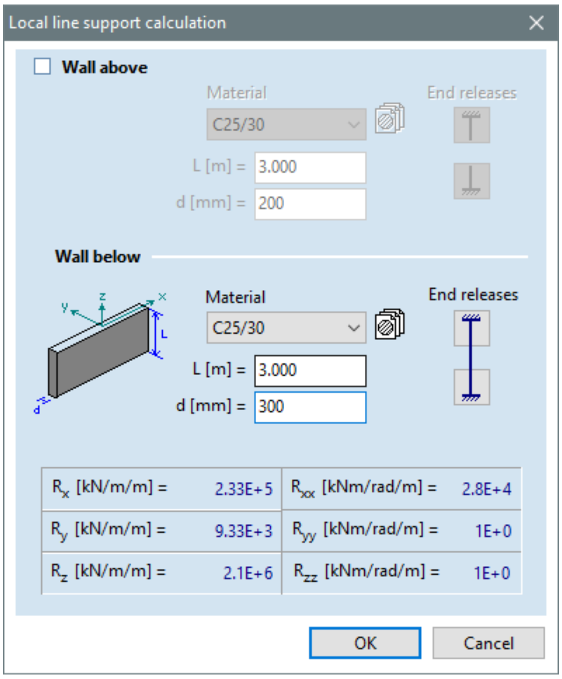


Confirm selection with ***OK*** and the following will be displayed:

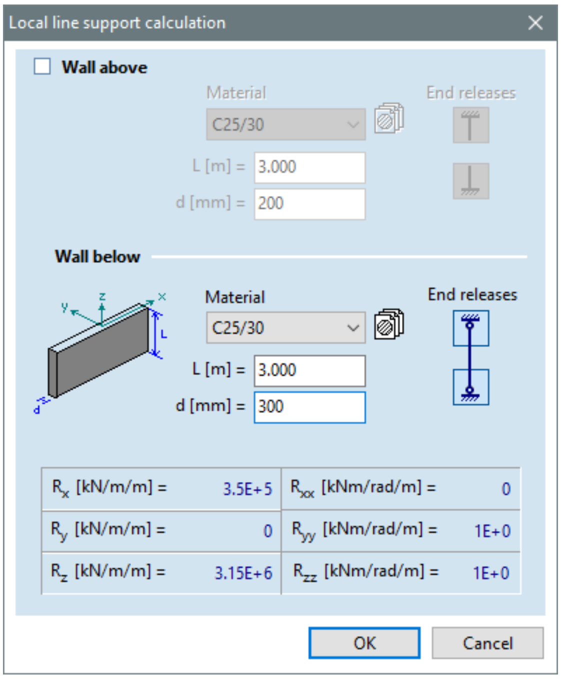


Local line support Calculation

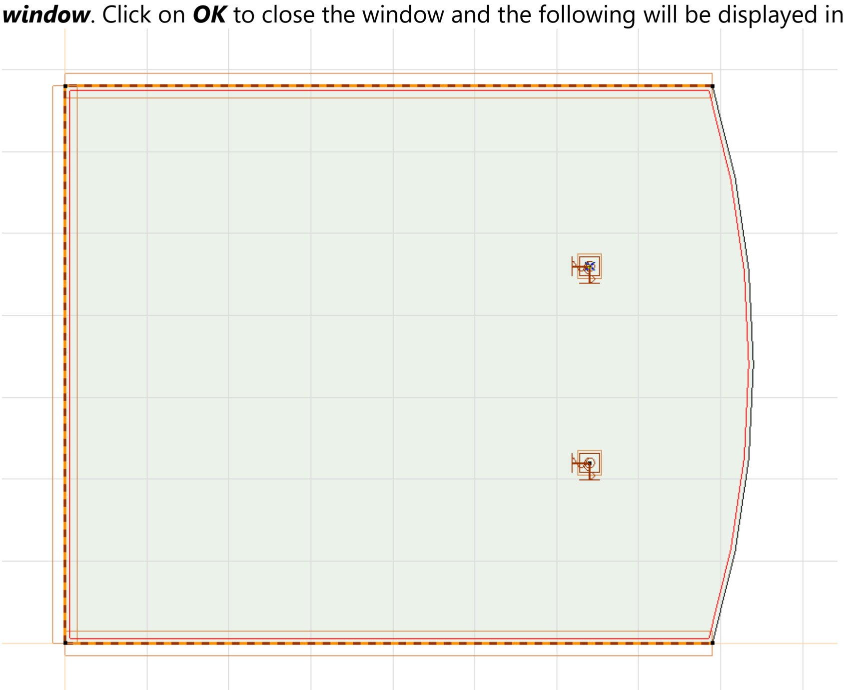
Click again on ***Calculation..*** button to calculate the stiffnesses automatically considering support conditions. In the next window set ***L [m]*** to ***3*** and ***d [m]*** to ***300***:



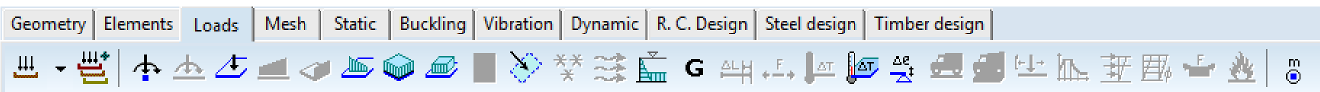
To set ***End Releases*** to pinned, click on both icons:



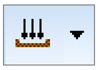
Confirm settings with ***OK*** and the calculated values are displayed at the components in ***Line support window***. Click on ***OK*** to close the window and the following will be displayed in the main window:



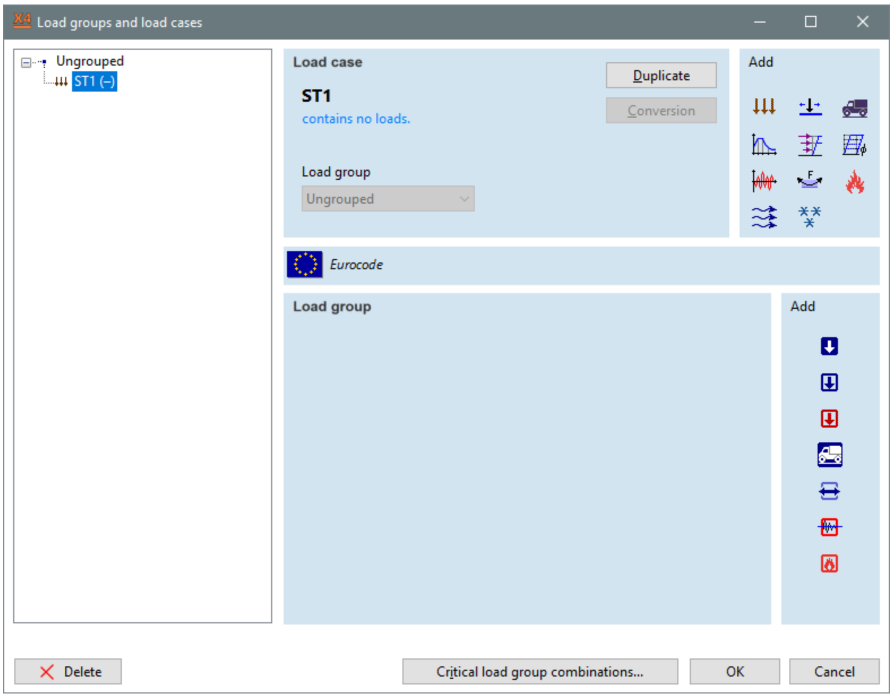
Loads Next step is to define loads on the slab, click on ***Loads*** tab:



Load cases and load groups



Various loads should be separated into load cases. Click on ***Load cases and load groups*** icon to add new load cases.

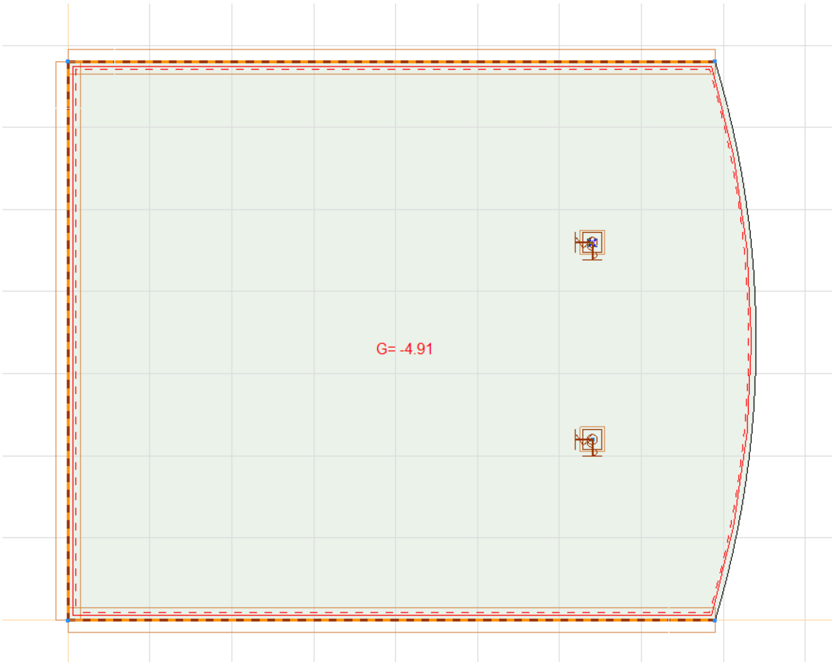


In the window that appears, click on the name ***ST1*** in the top left corner and rename it to ***SELF-WEIGHT***. (***ST1*** is an automatically generated load case, which should be renamed in our example.)

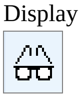
Close window with ***OK***, then the previously edited load case (***SELF-WEIGHT***) is active. The actual load case is indicated on ***Info palette***:

Self weight

Click on ***Self weight*** icon then select all elements with ***All*** (*) button. By clicking on ***OK,*** a red dashed line parallel to the contour line indicates the self weight of the domain.

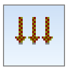


Display options



Click on ***Display options***, then on ***Symbols*** tab uncheck ***Mesh*** and ***Surface centre*** under ***Graphics Symbols*** group. Close with ***OK***.

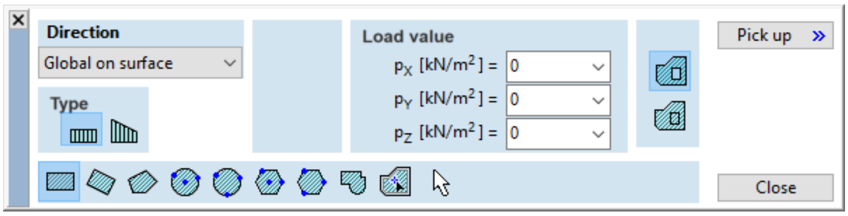
New Load case



Open again ***Load cases and load groups*** window and create new ***Static*** load case with ***Name FINISH-ES***, this load case will contain loads from slab finishes (in our example ***2,5*** kN/m²). Close with ***OK***.

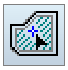
Distributed load on domain

Click on ***Distributed load on domain*** icon, then the following window shows up:



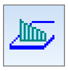
Set ***p_Z [kN/m^2]*** to ***-2,5*** (negative value denotes opposite direction to local ***z*** direction).

Distributed domain load

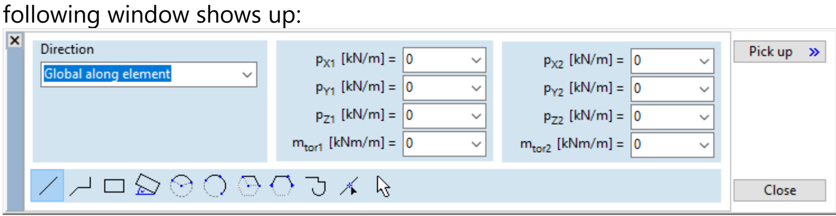


Assign load to the slab by clicking on ***Distributed domain load*** icon, then click inside the domain.

Domain line load

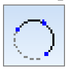


To define line load on the arched perimeter (load of balustrade), click on ***Domain line load*** icon. The following window shows up:



Set ***p_Z1*** and ***p_Z2*** load values at the endpoints to ***-1***.

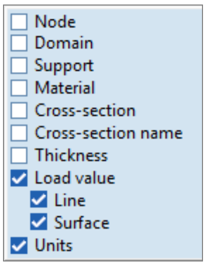
Arc by three points



Select ***Arc by three points*** icon from bottom row, then click bottom, central and top points of the curved edge. Then again click on top point of edge, finally press ***Esc*** to exit.

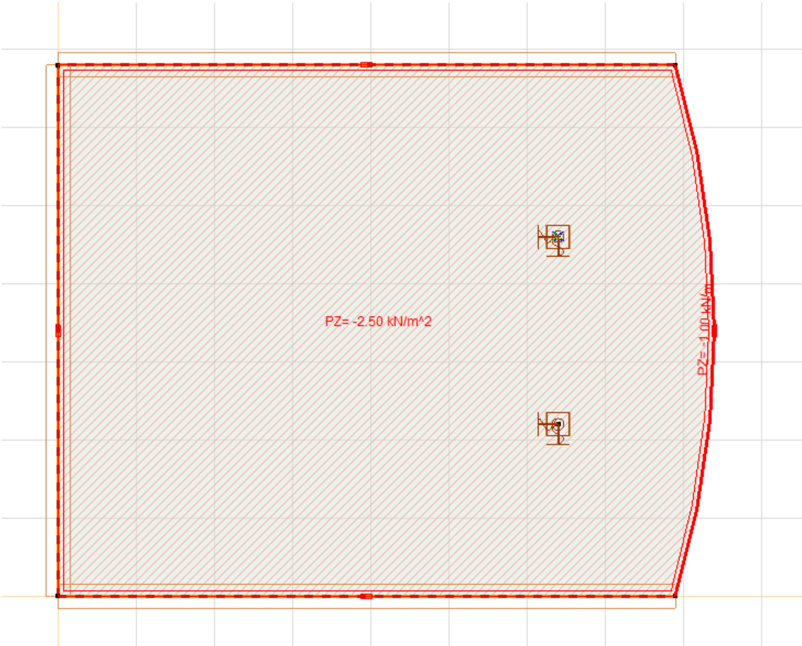
Load value

Switch on load intensity using ***Numbering*** function at ***Speed buttons*** (in the bottom right corner). Click on ***Numbering*** icon and the following list opens:

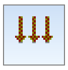


Here, check ***Load value*** and ***Units*** checkboxes.

The following will be displayed showing the load intensity and units on the domain:



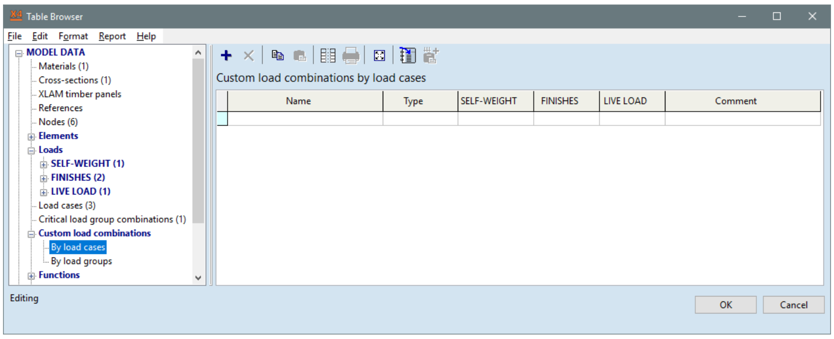
New Load case



Create a new load case with name ***LIVE LOAD***.
Specify live load on the slab in this load case. Define ***Distributed load on domain*** as shown previously. Set load intensity to ***-3,0*** kN/m^2.

Load combinations

Creating load cases have been finished, now load combinations should be set for design and checking. Activate ***Load combinations*** icon to open the following window:



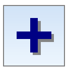
New row

Add two load combinations to the list and specify factors to combine load cases. Create first load combination by clicking on ***New row*** icon. Leave the name on default (***Co.#1***). Select ***SLS Quasipermanent*** combination type, and apply the following factors for each load case:

SELF-WEIGHT	1.00	<**Tab**>
FINISIHES	1.00	<**Tab**>
LIVE LOAD	0.30	<**Tab**>

Enter these values into the appropriate cells.

New row



Click on again ***New row*** icon to specify the second load combination. Keep the default name ***Co.#2***, select ***ULS*** combination type and specify the following factors:

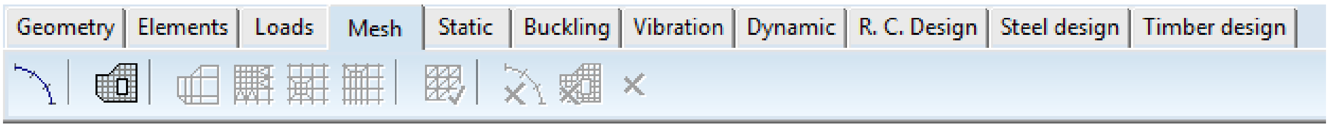
SELF-WEIGHT	1.35	<**Tab**>
FINISHES	1.35	<**Tab**>
LIVE LOAD	1.50	<**Tab**>

Click on ***OK*** to confirm load combinations.

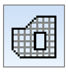
We have finished with basic data input but mesh must be generated before calculation.

Mesh

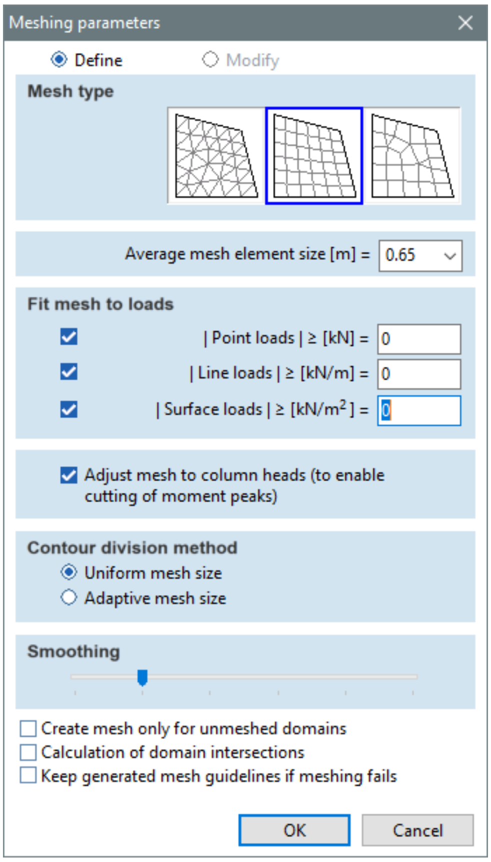
Select ***Mesh*** tab to define domain mesh:



Domain meshing

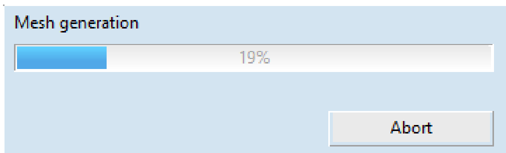


Click on ***Domain meshing*** icon, then select ***All*** (*) elements and confirm with ***OK***. In the following window select quadrangle as mesh type, set ***Average mesh element size [m]*** to ***0.65***. Check ***Fit mesh to load and activate Adjust mesh to column heads*** function. Using the last option, the mesh will be properly adjusted to column heads to prepare cutting of moment peaks (for more information, please see ***User's manual***).

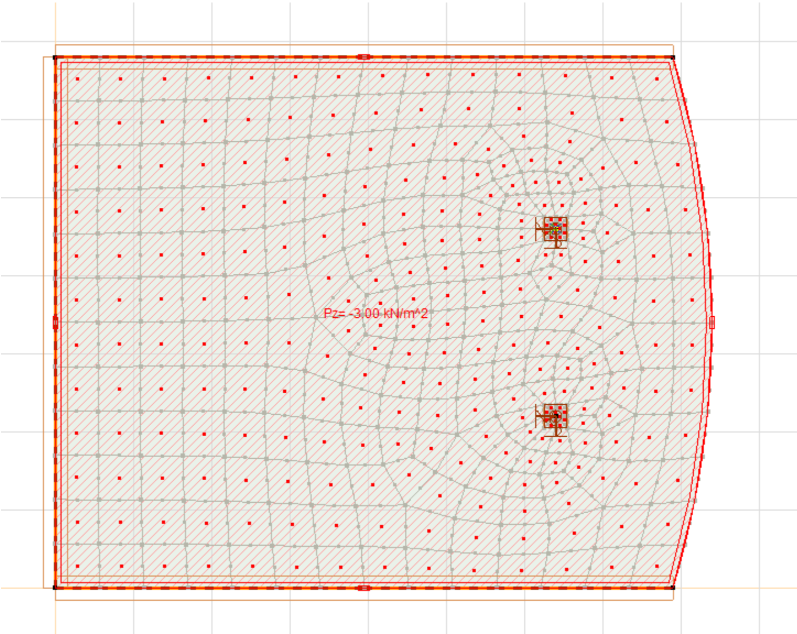


Click on ***OK***, then the automatic mesh generation starts.

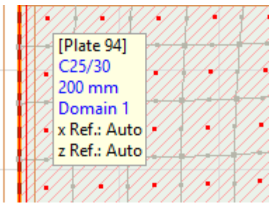
Progress bar shows the progress of meshing:



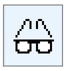
The following will be displayed in the main window after meshing:



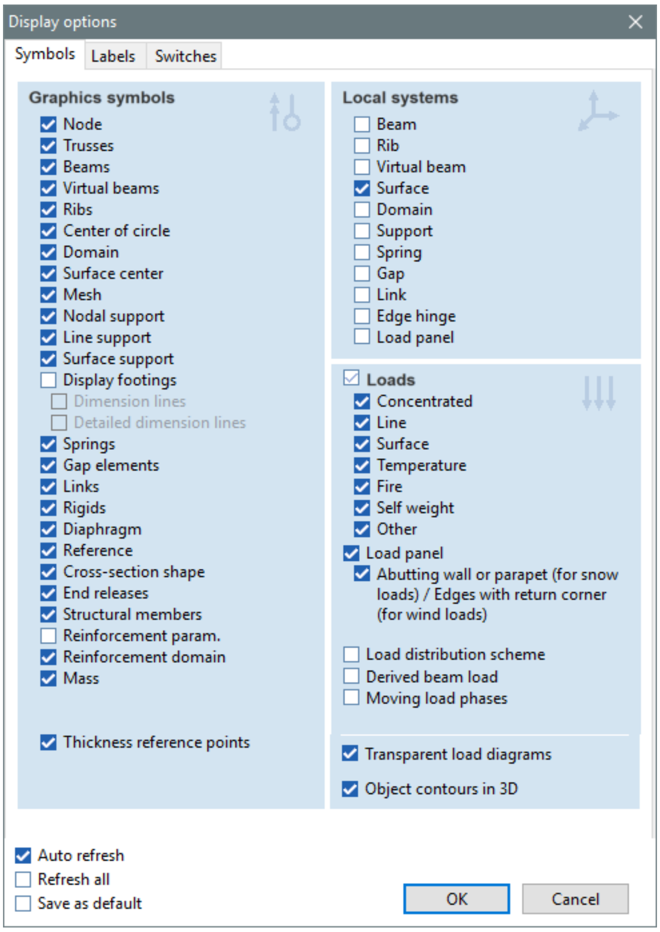
Moving the cursor over the centre of a finite elements (red dot), a hint shows up displaying the properties of the element:



Display options

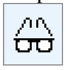


Switch on visibility of local coordinate system of the defined finite elements. Activate ***Display options*** and select ***Symbols*** tab:



In ***Local systems*** group, check ***Surface*** checkbox and close window with ***OK***.
Now, the local system is shown on every finite element: the red line denotes ***x*** direction, yellow for ***y*** direction and green for ***z*** direction.

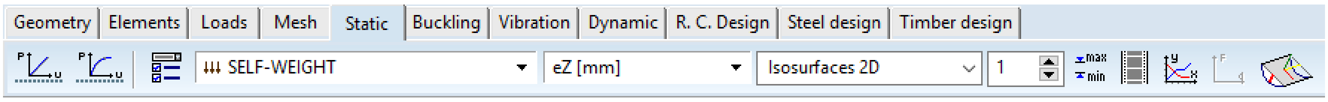
Display options



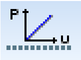
Switch off ***Local system*** of ***Surface*** elements in ***Display options*** window, this will not be necessary now.

Static

Click on ***Static*** tab to analyse the model:



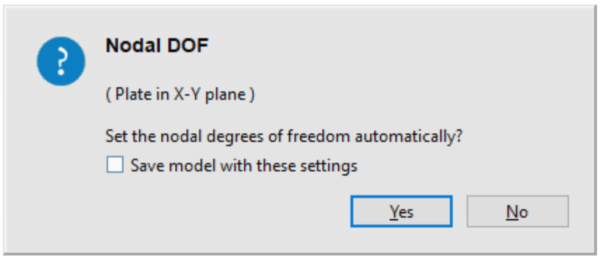
Linear static analysis



Click on ***Linear static analysis*** icon to run analysis.

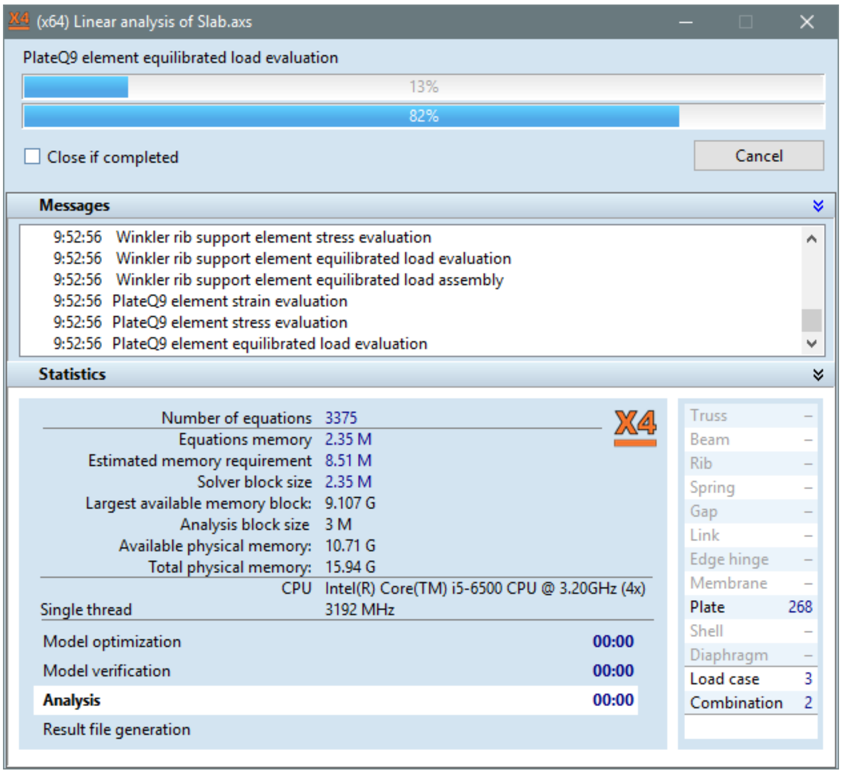
Nodal degrees of freedom

Program checks the model, warns to set nodal degrees of freedom and offers one type in the following dialog box:



Check ***Save model with these settings*** checkbox and degree of freedom settings will be saved. Click on ***Yes*** to accept suggestion (***Plane in X-Y plane***), then program continues analysis.

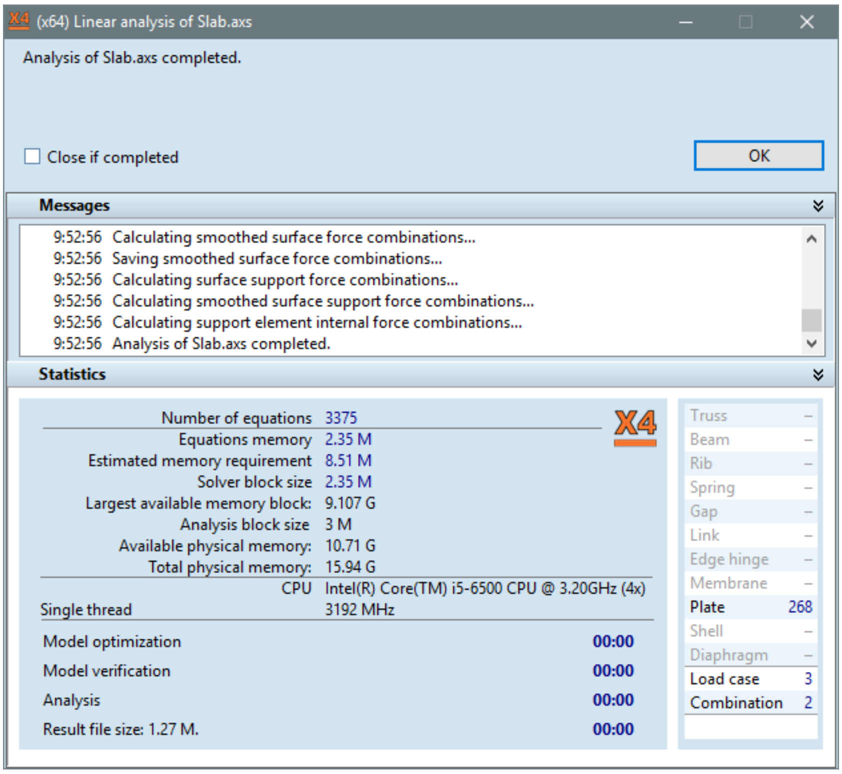
The progress bar shows the calculation process:



Click on ***Statistics*** to see more information about the analysis.

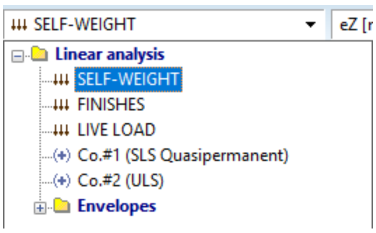
The top progress bar shows progress of the actual task. The progress bar bellow is showing the total progress. ***Estimated Memory Requirement*** shows size of used virtual memory for analysis. If the PC's memory size is less than this, error message regarding size of virtual memory will be displayed.

The calculation closes with the next window, then click on ***OK*** to close window.



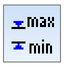
Returning to the main window, the program displays automatically vertical deformations ***ez [mm]*** considering ***SELF-WEIGHT*** load case in ***Isosurfaces 2D*** display mode.

Select load combination ***Co.#1*** (***SLS***) to check serviceability limit states (note: this is only the result of linear analysis):

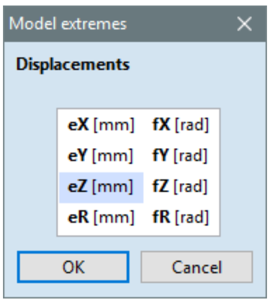


Deformation values are negative because the positive direction of global ***Z*** axis is opposite to the direction of the specified loads.

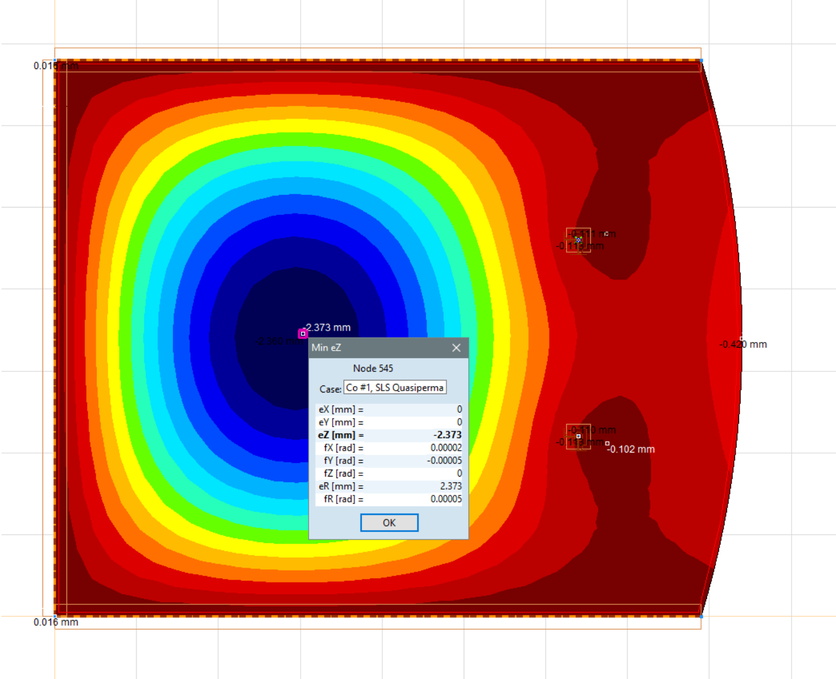
Min, max values



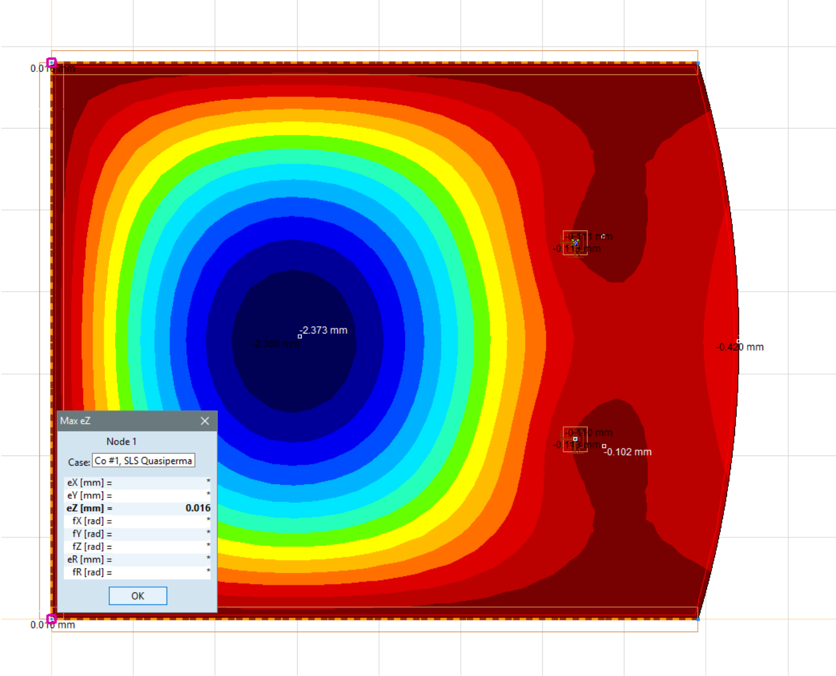
To find location of maximum deformation, use ***Min, max values*** function. By clicking on icon, shows up following window:



Select one of the deformation components. Confirm with ***OK***, then the program shows maximum negative value and its location as well:

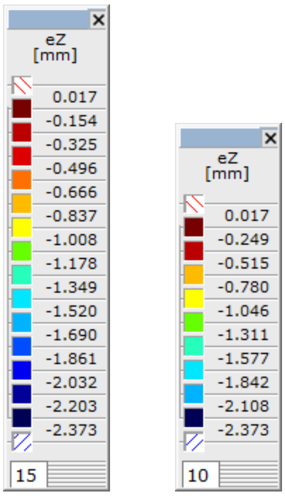


Click on ***OK*** to continue, then the result panel jumps to the maximum positive value:

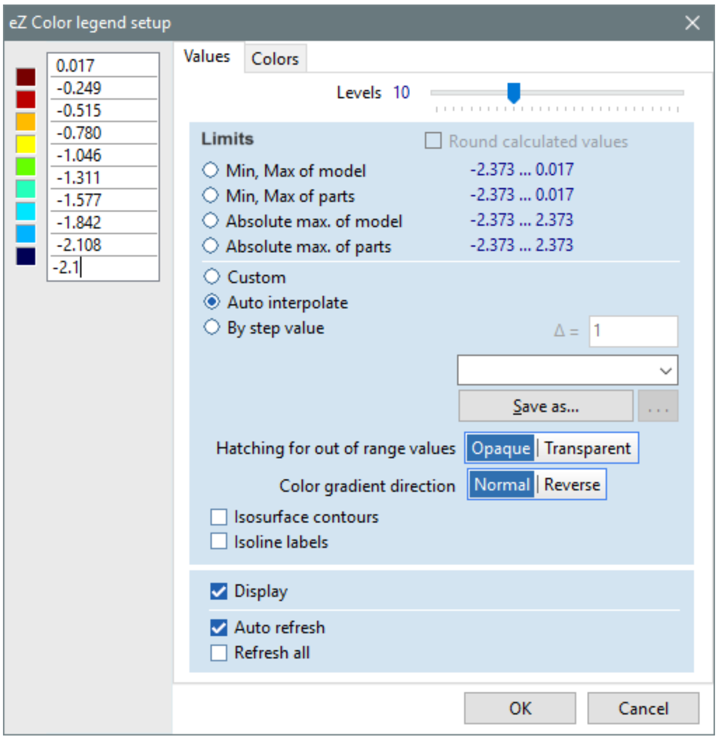


Color legend

The ***Color legend*** shows boundary values of each color. Adjust number of boundary values by dragging the bottom edge of the palette:

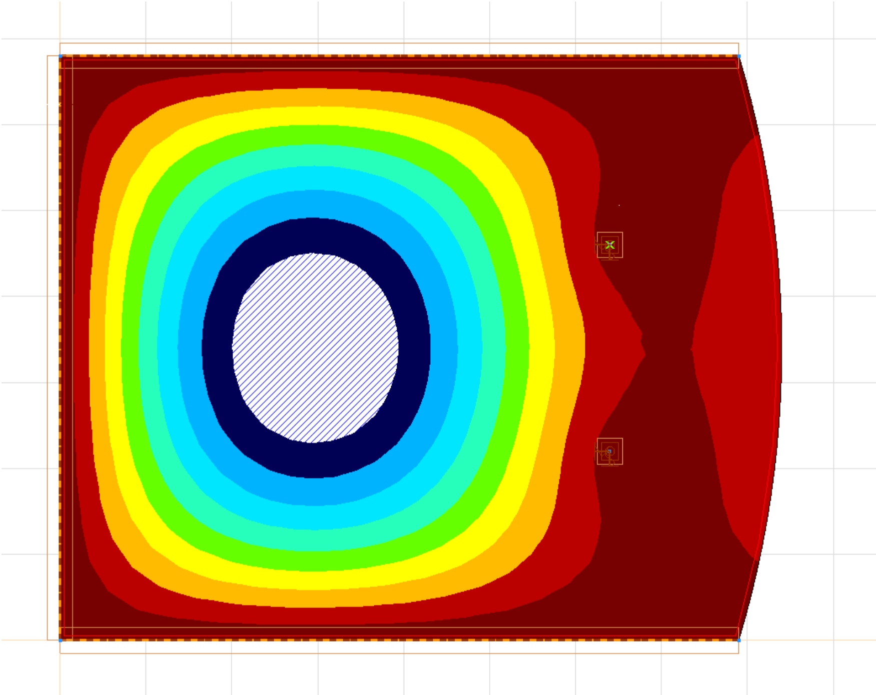


Find areas where the deflection is larger than ***2.1*** mm. Click on ***Color legend***, then a setup window shows up. On the left side, click on the bottom value of the list and change default value (***-2.423***) to ***-2.1***.



Press ***Enter***, then ***Auto interpolate*** function sets the other boundary values.

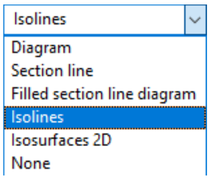
Close window with ***OK***, in the main window the following will be displayed:



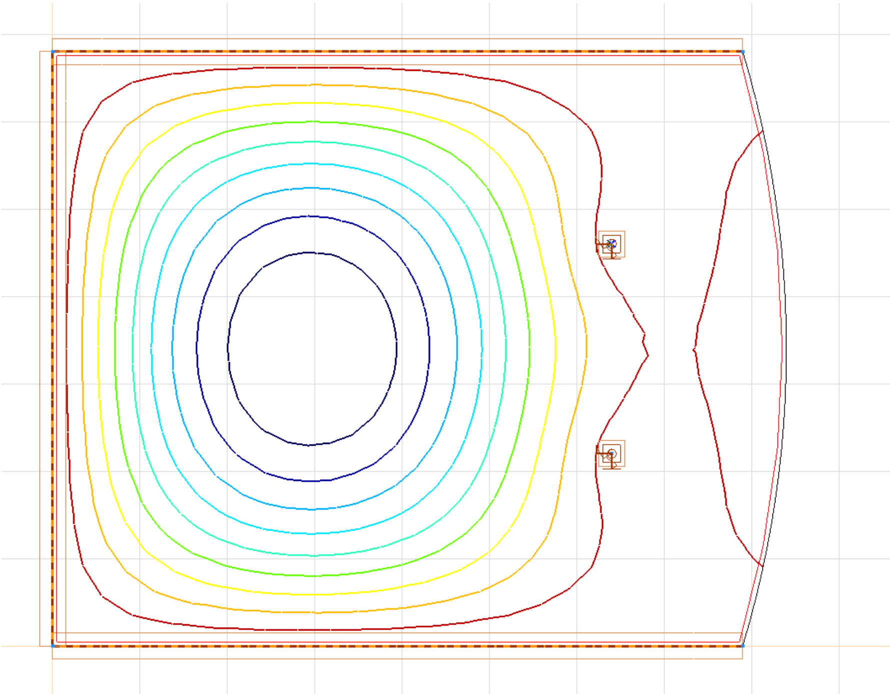
Area with larger than ***2.1*** mm is indicated by the hatched area in blue.

Isolines

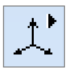
See results in ***Isolines*** display mode as well. Click on arrow right to ***Isosurfaces 2D*** title and select ***Isolines*** in the drop list.



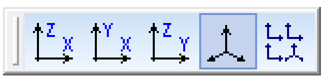
The resulting figure is the next:



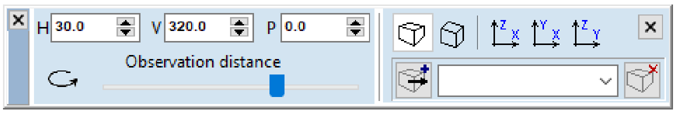
Views



Change view to ***Perspective***.

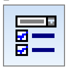


Change ***Perspective settings*** as shown below:

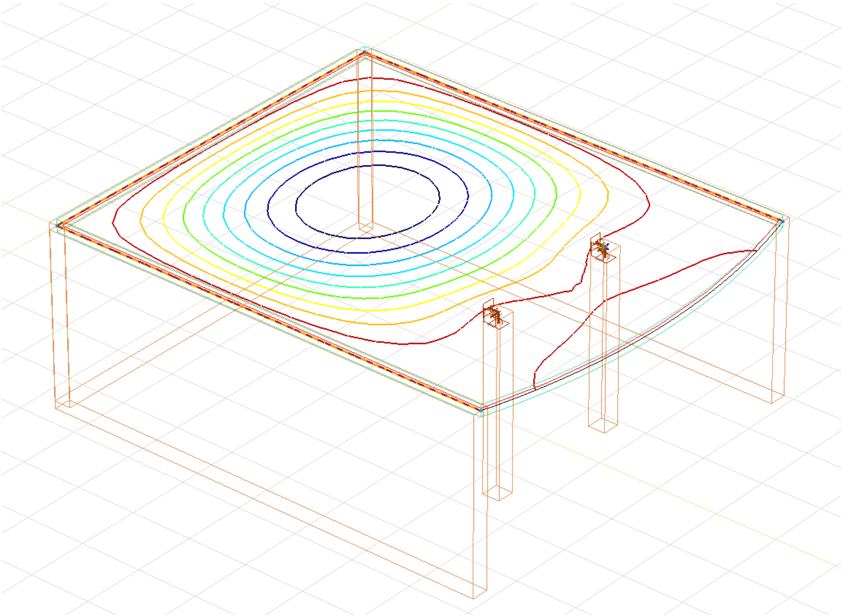


then click on ***X*** in the top right corner to close window.

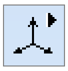
Result display parameters



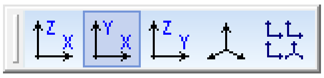
To see the deformed shape, click on ***Result display parameters*** icon. Select ***Deformed Display shape*** in the window and confirm with ***OK***. Now, the deformed shape is displayed, if necessary change scale of the diagram (next to the ***Display mode*** drop down list).



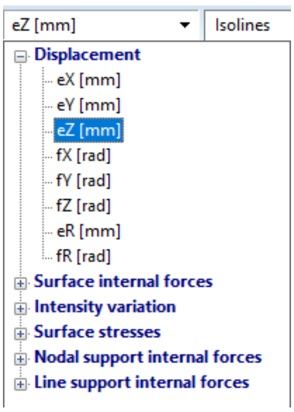
Views



Change view to ***X-Y Top view***!



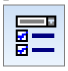
After the deformations component ***ez***, check internal forces of the slab.
For this purpose, firstly select load combination ***Co.#2*** (***ULS***).
Click on arrow next to ***ez [mm]*** title, then the following drop list appears showing different result components:



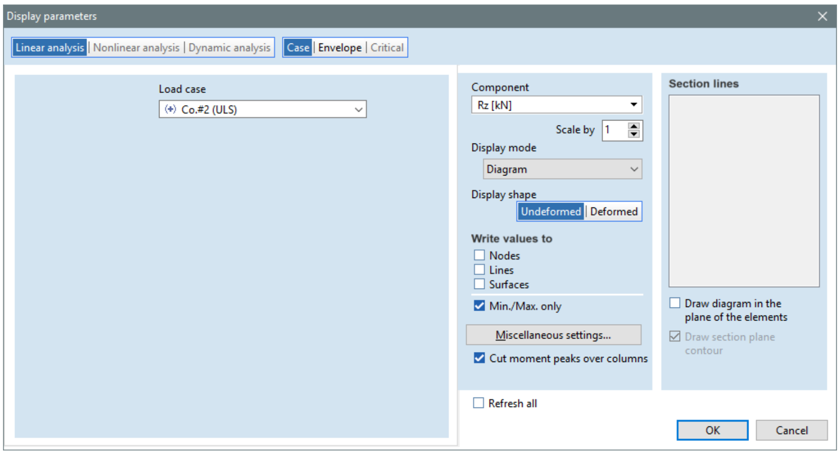
Select ***Surface internal forces*** – ***mx [kNm/m]*** then ***mx*** moments will be displayed in ***Isolines*** mode. The ***mx*** specific moment represents the moment that rotates around local ***y*** axis. With same method, select and see the following result components: ***my***, ***mxy***.

Now, select ***Rz [kN]*** component in ***Nodal support internal forces*** list to check vertical internal force in nodal supports.

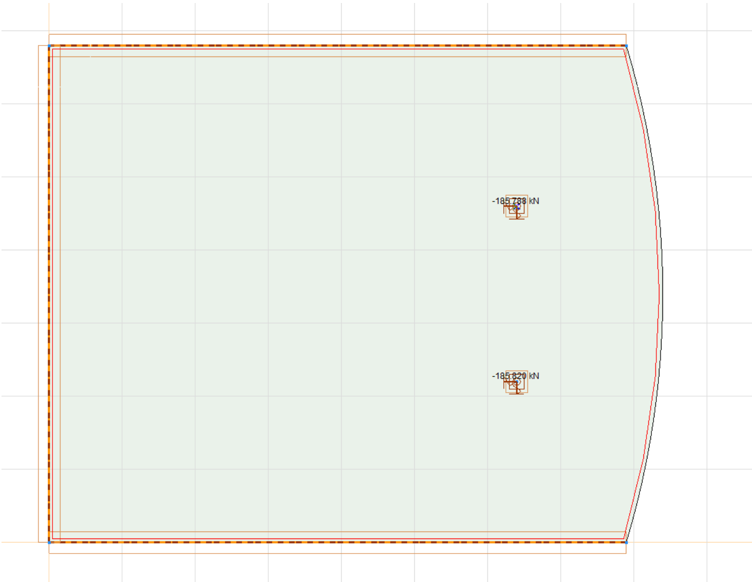
Result display parameters



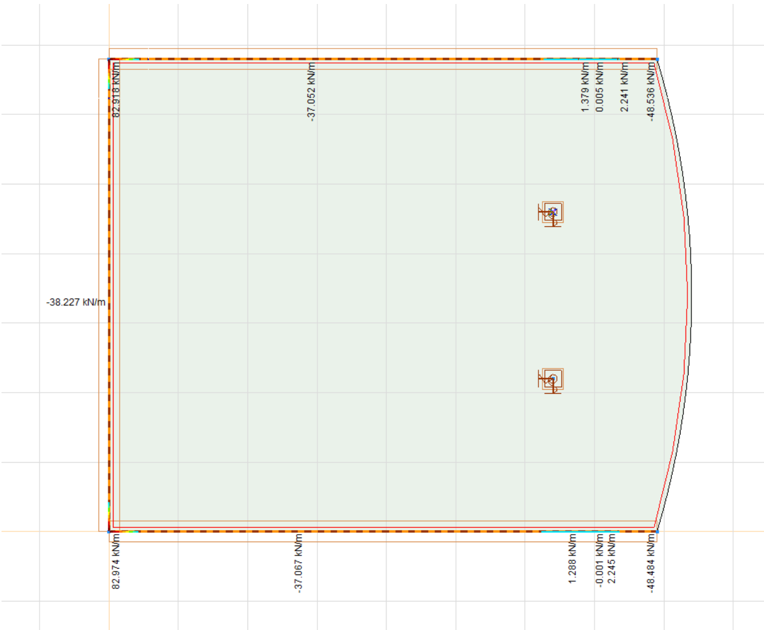
Activate ***Results display parameters*** icon:



In this window select ***Write values to Nodes*** function and close with ***OK*** to see the results:

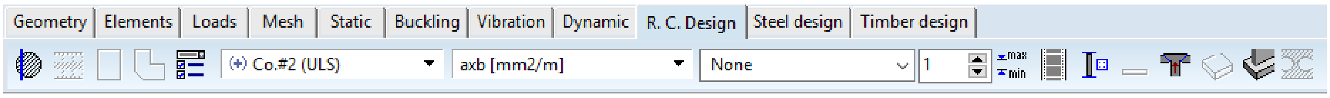


Change result component to ***Rz [kN/m]*** in group ***Line support internal forces***. Activate ***Result display parameters*** window and check ***Write values to Lines*** function to see the result values.

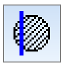


R. C. design

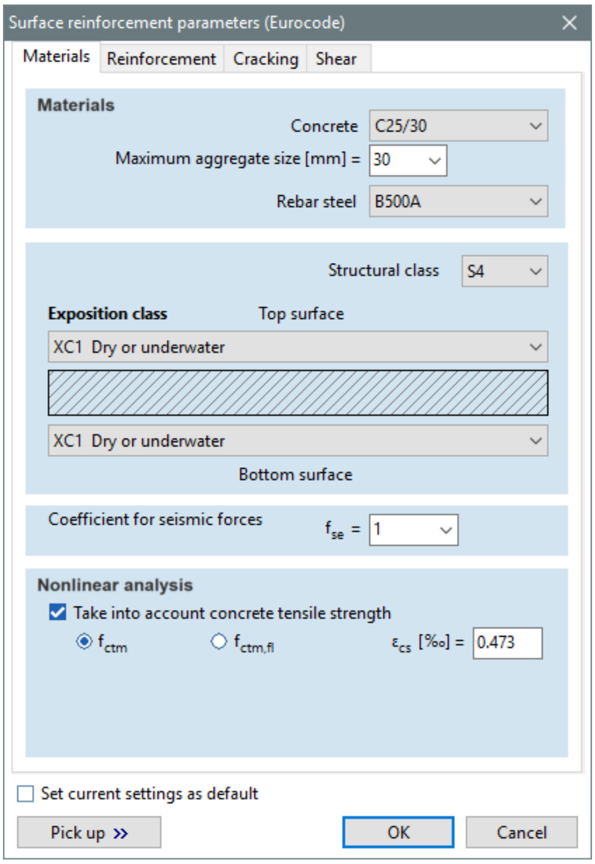
Click on ***R. C. design*** tab to calculate required reinforcement and to apply actual reinforcement.



Reinforcement parameters

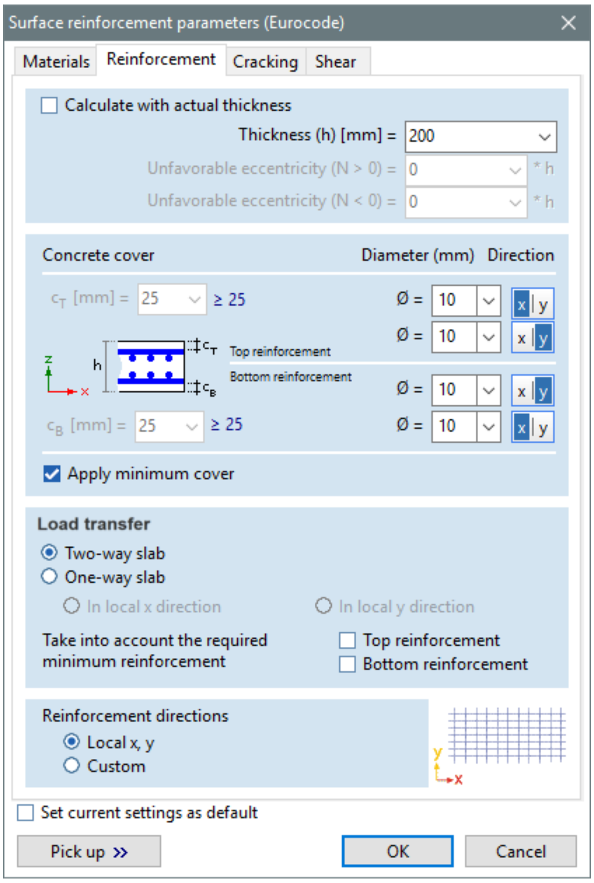


Click on ***Reinforcement parameters*** icon, then select ***All*** (*) and confirm with ***OK***. The following ***Surface reinforcement parameters*** window shows up:



Firstly, specify ***Exposition classes*** on ***Materials*** tab. Set ***XC1*** for ***top*** and ***bottom surface***.

On ***Reinforcement*** tab, set ***Primary direction of reinforcement*** as ***x*** for top and bottom. Check ***Apply minimum cover*** checkbox:



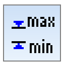
Close window with ***OK***.

Change display mode to ***Isosurfaces 2D***-re!

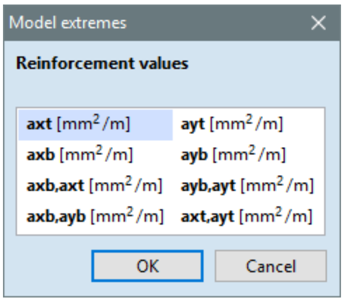
Now ***axb [mm²/m]*** results can be seen, which is the required amount of reinforcement in local ***x*** direction at bottom of the slab. See ***Reinforcement values*** in the list of results where you can also select required area of reinforcement for other directions as well.

By setting the boundary values of ***Color legend***, the areas that can be reinforced with a certain rebar spacing (e.g. for Ø10/160 reinforcement set 491 mm²/m) can be isolated.
Let us check the required top reinforcement in local ***x*** direction, click on ***axt [mm²/m]*** result component.

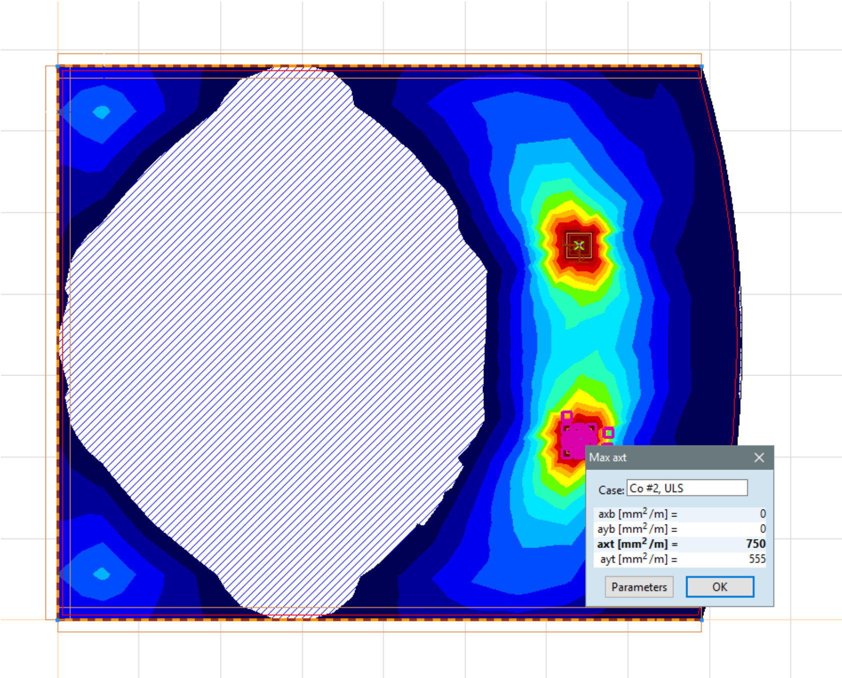
Min, max values



To find location of maximum required area of reinforcement, click on ***Min, max values*** icon. Select the appropriate result component in the next window:



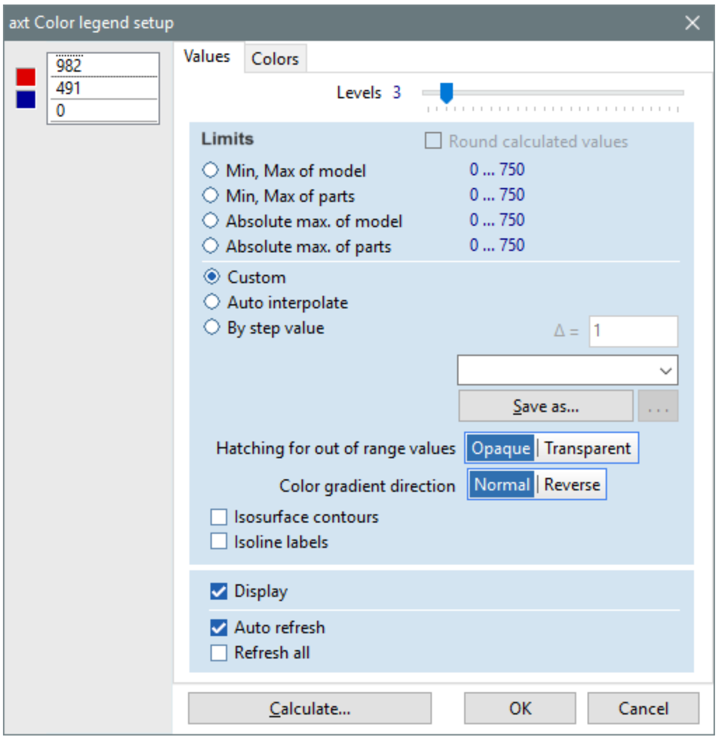
By clicking on ***OK***, the program shows maximum area of required reinforcement and its location.



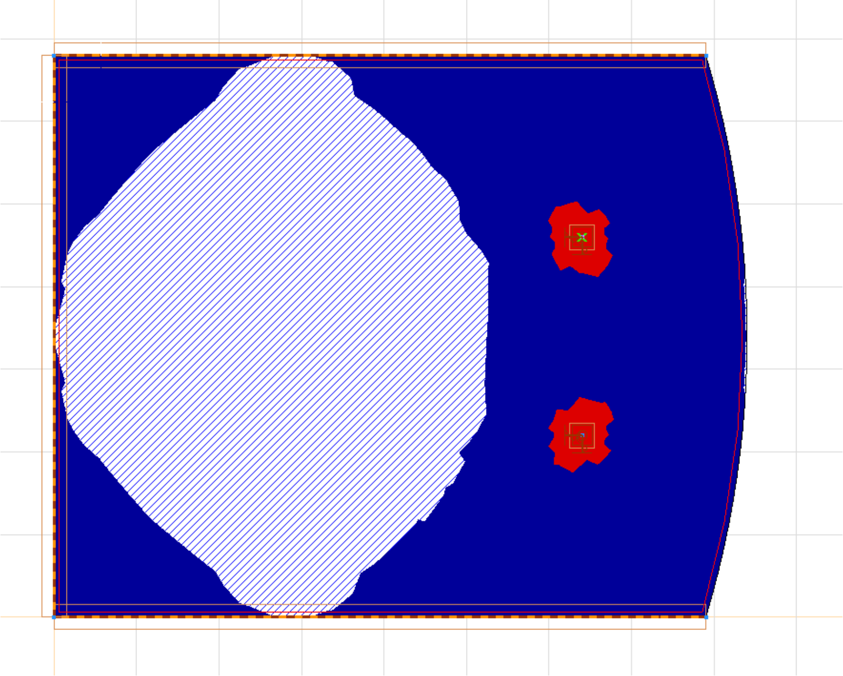
Click on ***OK*** to exit.

Apply actual reinforcement according to the requirements.
As a base reinforcement, use ***Ø10/160 mm (491 mm²/m)*** reinforcement on top and bottom layers in both direction. In the area of maximum moments use ***Ø10/160 mm*** extra reinforcement additionally (in these areas the total area of reinforcement will be ***982 mm²/m***).
The previous figure shows that the maximum required reinforcement (***750 mm²/m***) is less than the area of double-reinforcement (***982 mm²/m***), so the slab can be reinforced by these concepts.
To isolate areas with different reinforcement requirement, set number of levels in ***Color legend*** to ***3***.

Enter the above two values (***491*** and ***982*** mm²/m) as boundary values.



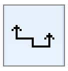
Close settings with ***OK***, then the program separates areas by colors where the specified reinforcement spacing is still sufficient.



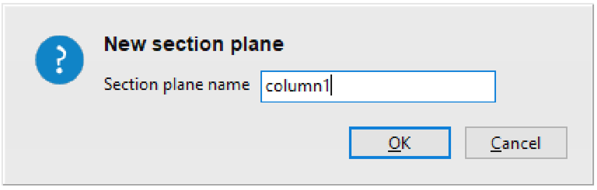
It can be seen that there is no need for upper reinforcement in the middle, hatched area of the slab. In the blue areas ***Ø10/160 mm*** base reinforcement is needed, and in the vicinity of the columns extra reinforcement must be applied (***Ø10/160 mm+ Ø10/160 mm***).

Let us examine again design moments in the vicinity of the columns. Go back to ***Static*** tab and select ***mxD+*** design component in ***Reinforcement design forces*** list. The result component ***mxD+*** considers also ***mxy*** internal forces.

Section lines

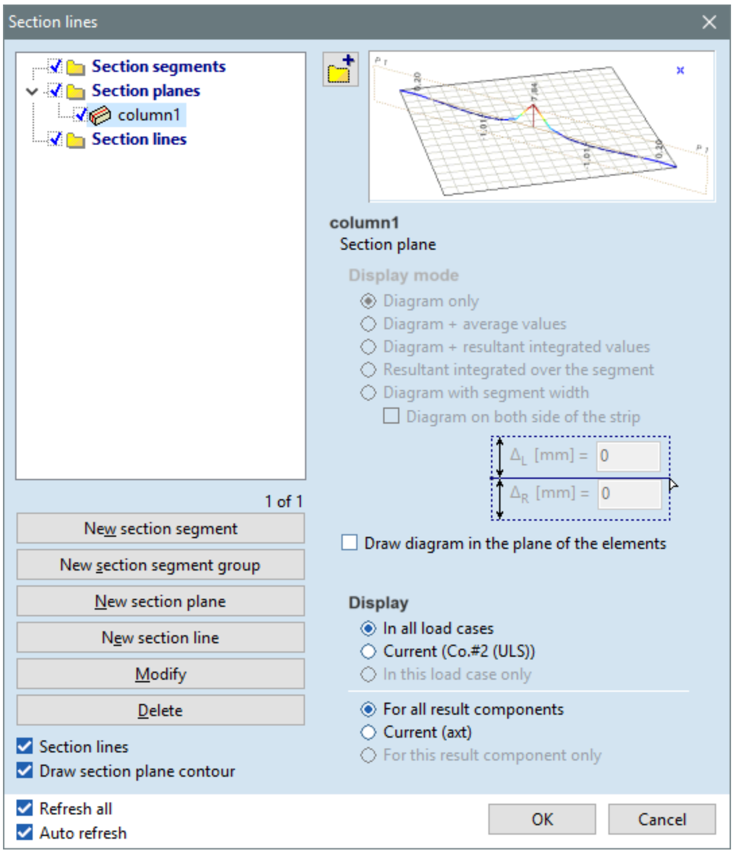


Select ***Section lines*** function from vertical toolbar on the left.
In the popup window click on ***New section plane*** button and in the dialog window specify the name of the section, type in ***column1***.



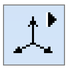
By clicking on ***OK***, the location of the section plane should be defined. First click on top nodal support then on bottom nodal support.

The following window will be displayed:

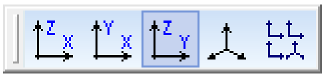


Click on ***OK*** to see moments in ***x*** direction in plane section of ***column1***.

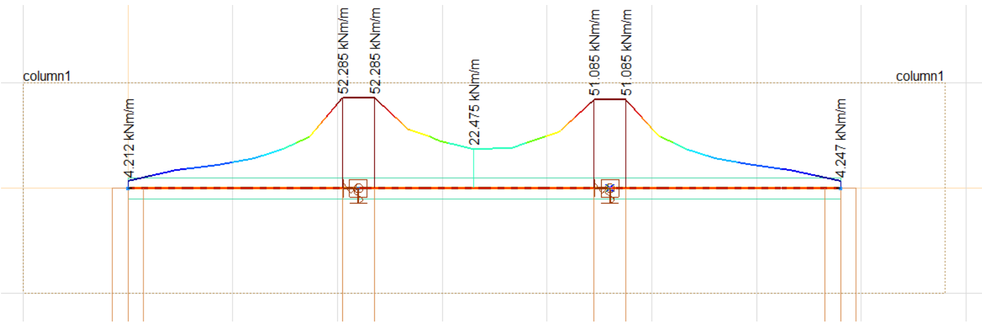
Views



Change view to ***Y-Z*** plane,



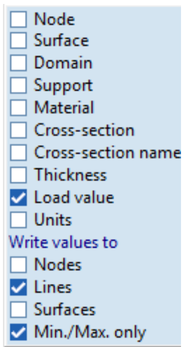
and change results from ***Isosurfaces 2D*** to ***Section line***, then the internal forces are displayed in section defined previously. It can be seen on the diagram, that software cut moment peaks over columns because this function was activated previously when meshing was created (see ***User's manual*** for more information about the rules and operation).



Now, turn off sections. Click on ***Section planes*** speed button (bottom right corner).

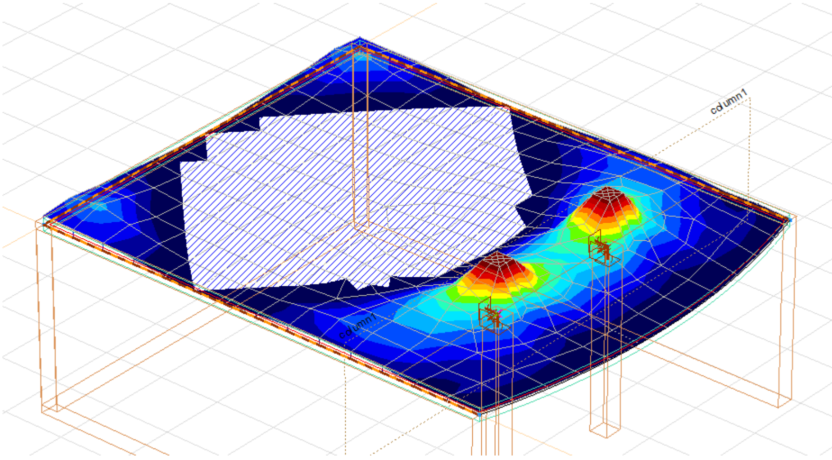
Speed buttons

Click on ***Numbering*** icon (among ***Speed buttons***) and uncheck ***Write values to Lines*** and ***Units*** as well.

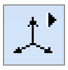


Change view to ***Perspective*** and ***Section line*** display mode to ***Isosurfaces 3D***.

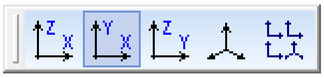
The following will be displayed which shows design moments in local ***x*** direction:



View

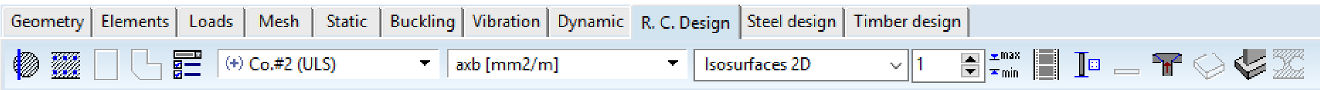


Return to ***X-Y*** view!

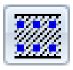


R. C. design

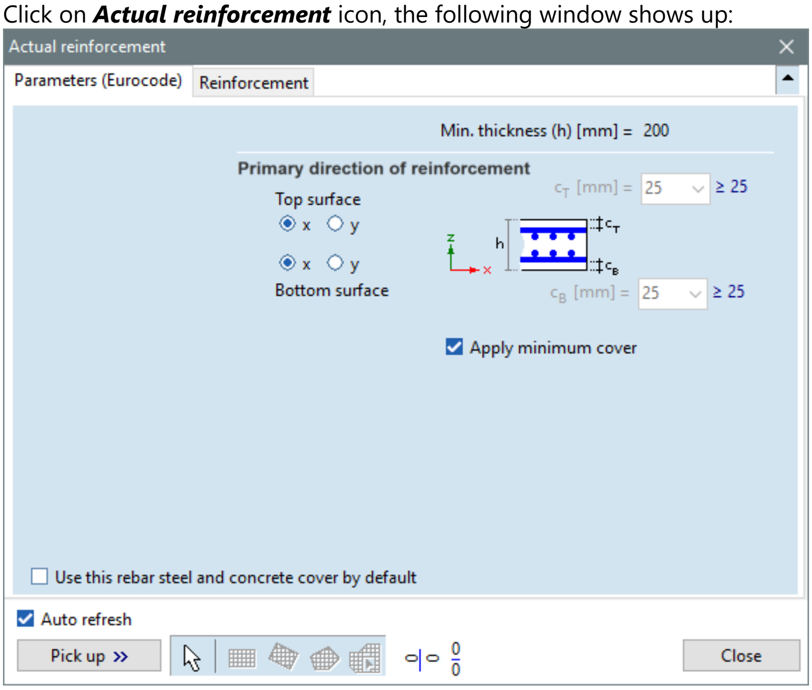
To specify actual reinforcement, change tab to ***R. C. design***.



Actual reinforcement



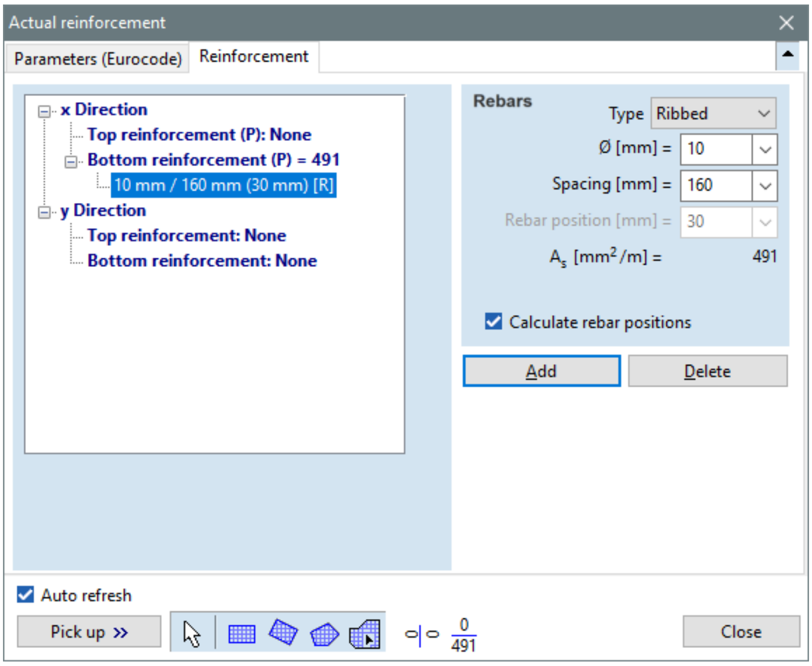
Click on ***Actual reinforcement*** icon, the following window shows up:



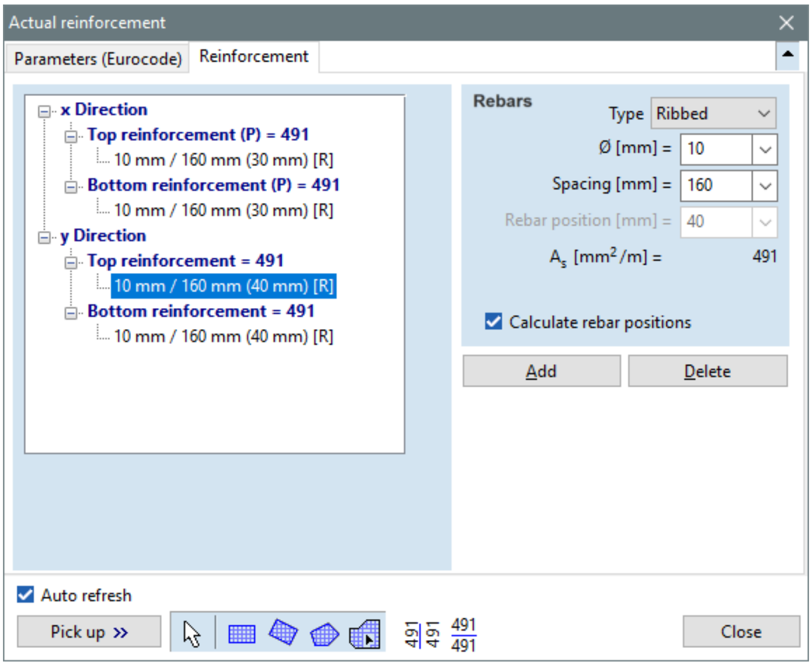
Firstly, the base reinforcement (***Ø10/160 mm***) will be specified in each direction, the extra reinforcement will be added to actual reinforcement later.

On ***Parameters*** tab, set ***x*** direction as ***Primary direction of reinforcement*** on top and bottom surface and check ***Apply minimum cover*** checkbox.

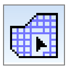
On ***Reinforcement*** tab, set Rebar diameter ***Ø[mm]*** = ***10***, and ***Spacing [mm]*** to ***160***. Check ***Calculation rebar position*** function. After select ***Bottom reinforcement*** in ***x Direction*** and click on ***Add*** button. The next will be displayed in the window:



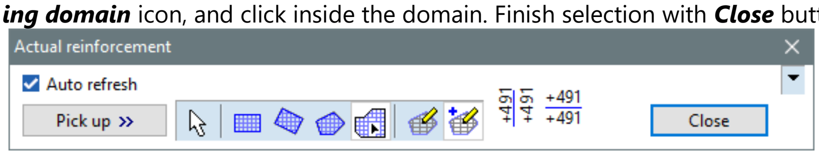
Define the other reinforcement layers in ***x*** and ***y Direction*** in the same way. Performing the task, you will need to see the next:



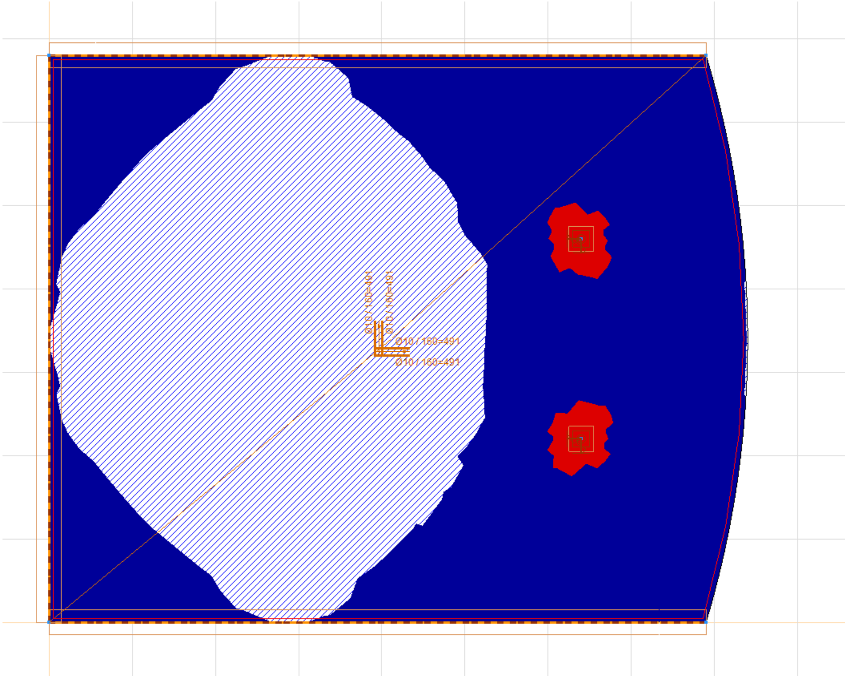
Reinforcement over an existing domain



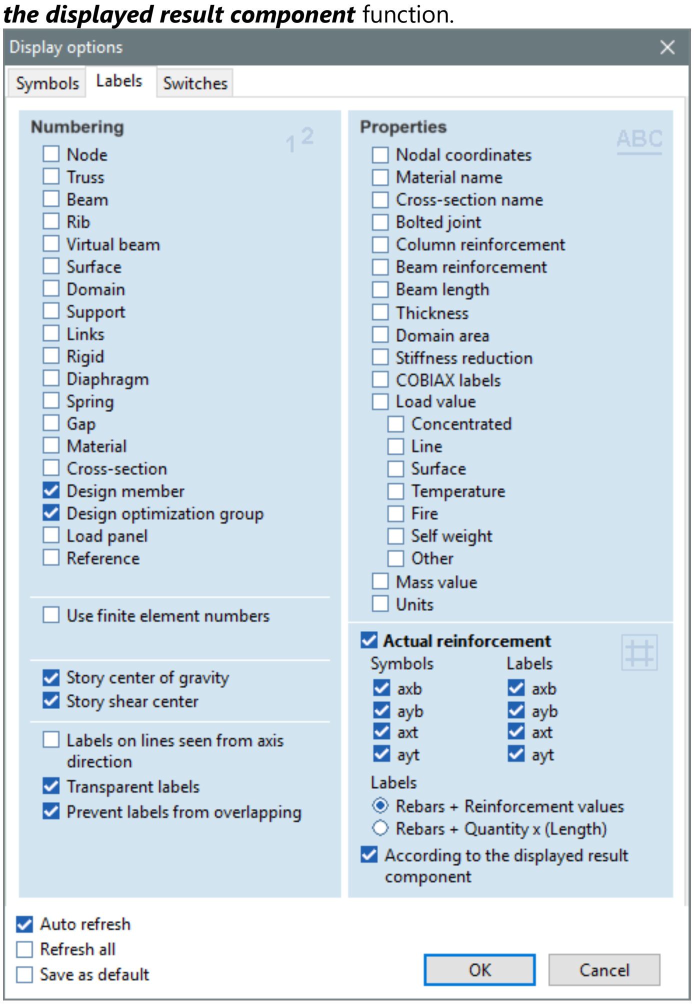
Now, the set reinforcement should be assigned to the domain. Click on ***Reinforcement over an existing domain*** icon, and click inside the domain. Finish selection with ***Close*** button.



Change display mode to ***Isosurfaces 2D***. The specified reinforcements are displayed on the slab:



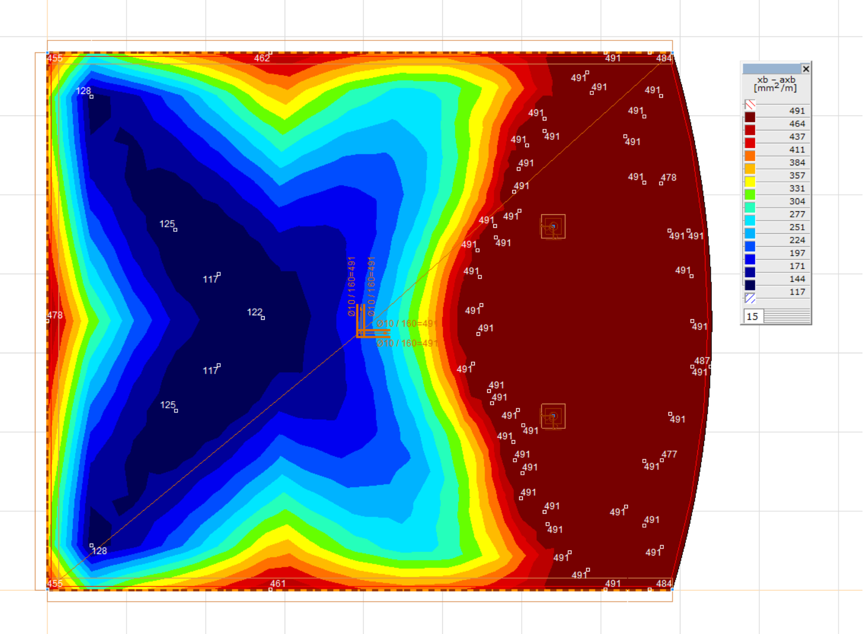
The thick brown lines and the titles indicates the given actual reinforcement. If only one direction of reinforcement is visible on the screen, then open ***Display options***. On ***Label*** tab switch off ***According to the displayed result component*** function.



Reinforcement difference

Select ***xb-axb*** result component in ***Reinforcement difference*** list.

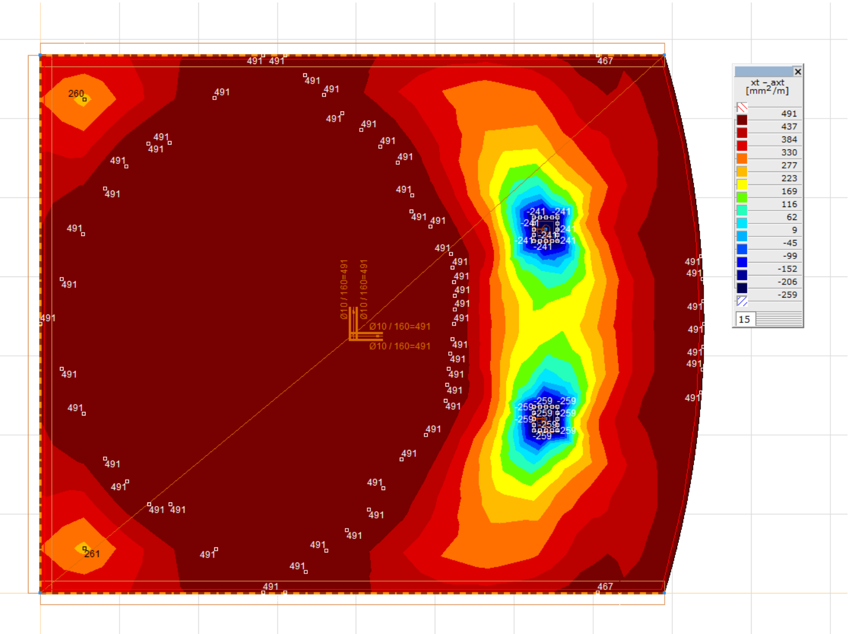
Click on ***Numbering*** icon (among ***Speed buttons***) and uncheck ***Write values to Surfaces*** to see the also the values not only the colors:



The specified actual reinforcement in local ***x*** direction is safe in ***Co.#2 (ULS)*** load combination because any negative value cannot be seen in the figure.

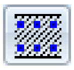
We will come to the same conclusion if ***yb-ayb*** result component is checked.

Now change to ***xt-axt*** component:



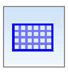
Around the nodal supports negative values can be seen indicating the areas where additional reinforcement must be applied.

Actual reinforcement



Increase reinforcement around the columns in ***2x2 m*** square area. Click on ***Actual reinforcement*** icon and select ***Reinforcement*** tab. Set ***Ø10/160*** reinforcement, select ***Top reinforcement*** in ***x Direction*** and click on ***Add*** button.

Rectangular reinforcement domain

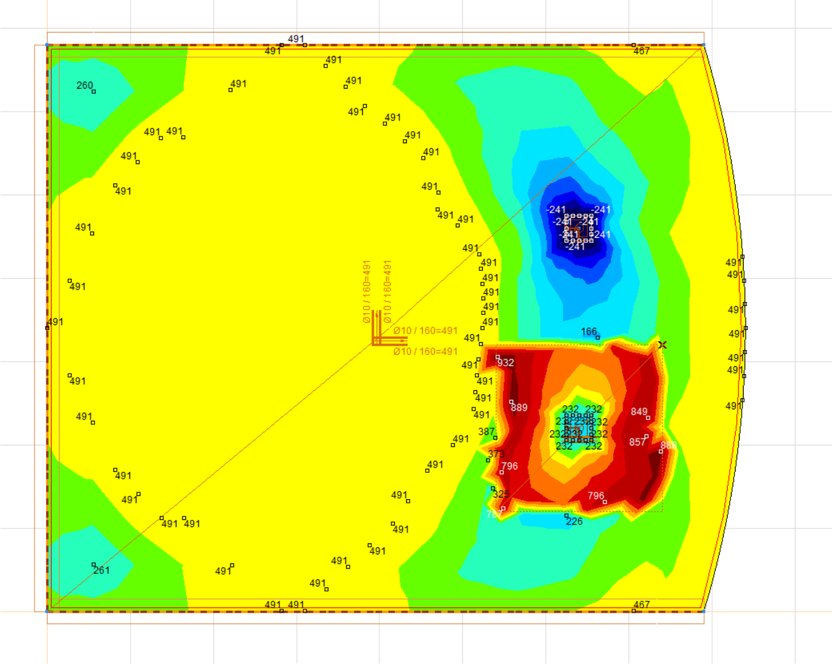


Additional reinforcement can be defined using ***Rectangular reinforcement domain*** function. Click on the icon, then move cursor to the node of the bottom nodal support. Here, press ***Insert*** key to place the origin of the local coordinate system, the first corner point of the square will be defined relative to this point.
Using relative coordinates, enter the following coordinates:

x	-1	y	-1	z	0	<**Enter**>
x	2	y	2	z	0	<**Enter**>

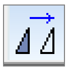
finally press ***Esc*** twice to exit.

The following will be the result:

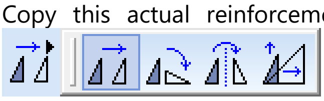


The results of ***xt-axt*** component around the bottom lower nodal support are positive, so the actual reinforcement is sufficient with the additional reinforcement.

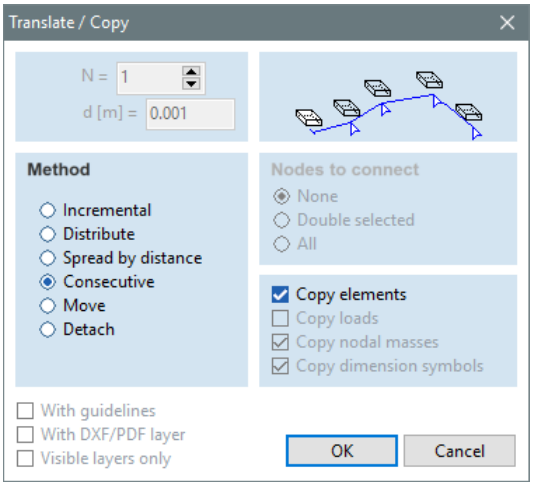
Translate / Copy



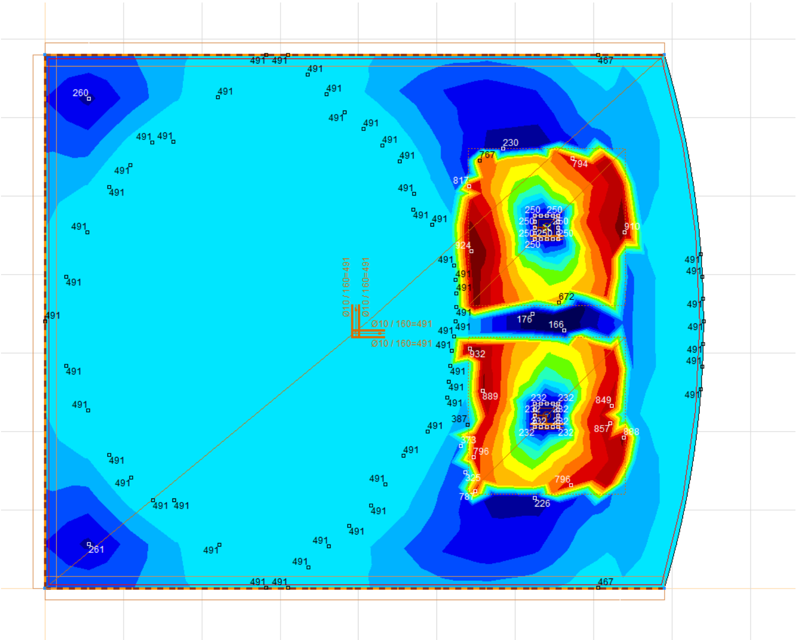
Copy this actual reinforcement domain to the upper nodal support. Click on ***Translate*** icon:



Then move the cursor over the contour of reinforcement domain and click on it to select. Confirm with ***OK*** to finish selection. In the ***Translate*** window select ***Incremental Method*** and set ***N=1***:

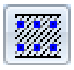


Close window with ***OK*** and define translation vector by clicking on lower and after on upper nodal support. Performing these steps, we have copied the actual **domain reinforcement**, the following will be displayed:

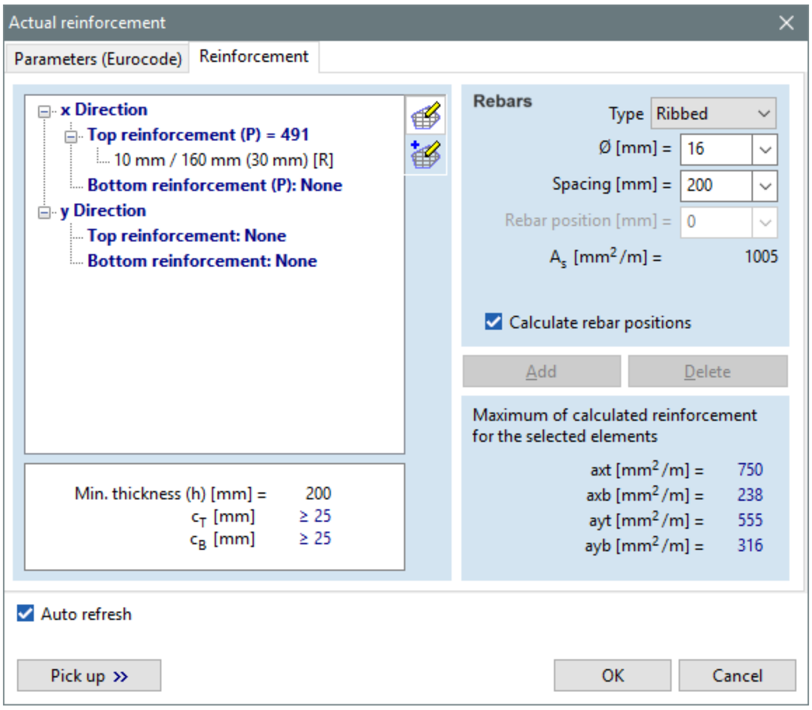


There is not any negative ***xt-axt*** value, so the actual reinforcement is sufficient at top in ***x*** direction. Now select ***yt-ayt*** result component. The negative values show that we also need additional reinforcement at top in **y** direction.

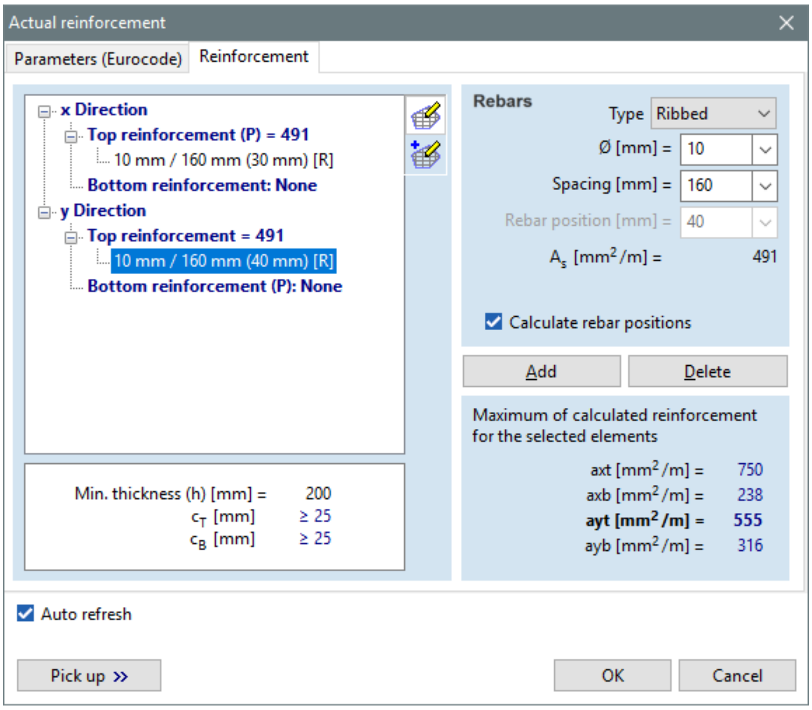
Actual reinforcement



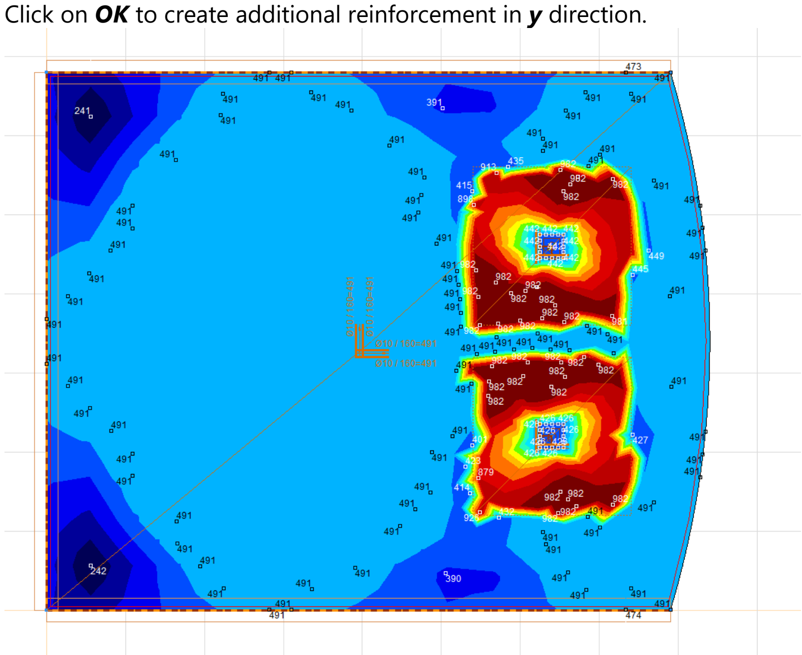
Press and hold ***Shift*** key, select actual reinforcement above columns, then click on ***Actual reinforcement*** icon. In ***Actual reinforcement*** window change to ***Reinforcement*** tab:



Click on the field of given reinforcement at ***Top reinforcement*** in ***x Direction***, after click on ***Top reinforcement*** in ***y Direction***. Finally, add the specified reinforcement to the last selected layer:



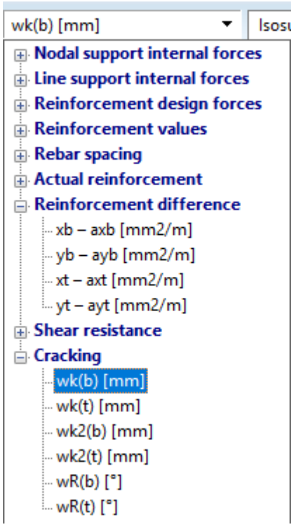
Click on ***OK*** to create additional reinforcement in ***y*** direction.



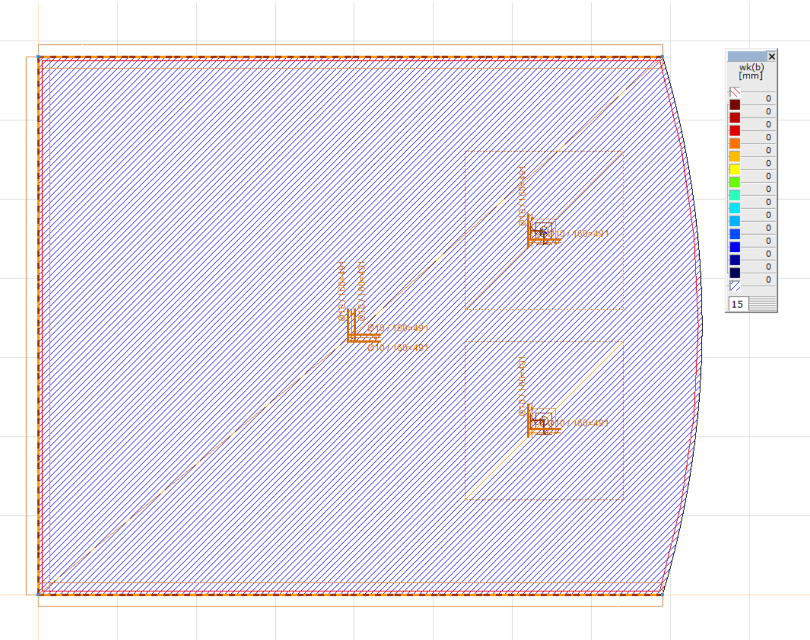
Now, the results of ***yt – ayt*** component are positive, so the actual (defined) reinforcement is sufficient for all reinforcement layers.
Check crack width in ***Co.#1 (SLS)*** load combination.

Crack width

Select load combination ***Co.#1 (SLS)***, then select ***Cracking – wk(b)*** result component which is showing crack widths in local ***x*** direction at bottom surface of the domain (slab).

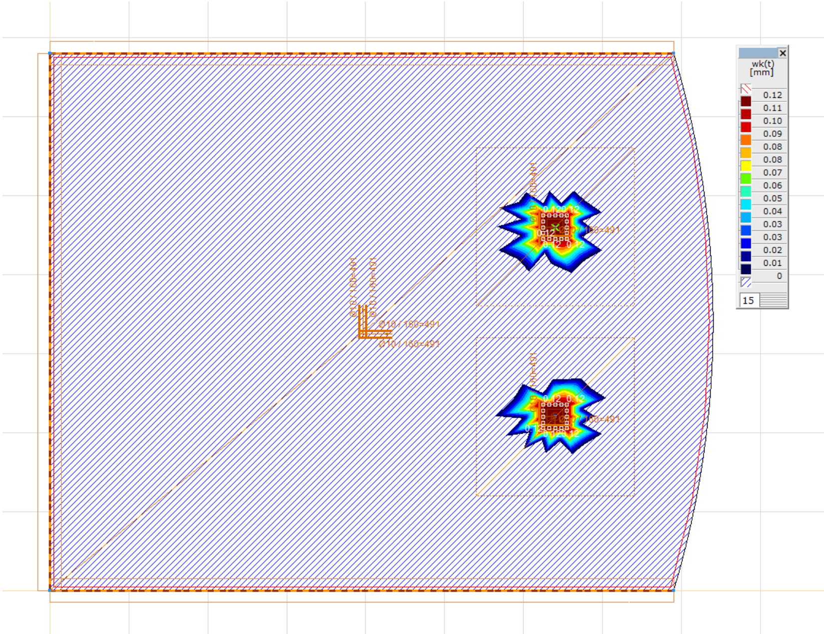


The following will be displayed:



Domain is hatched in blue which means that there are no cracks at bottom surface of domain applying the selected load combination.

Select ***Cracking – wk(t)*** result component which is showing crack widths in local ***x*** direction at top surface of the domain. The maximum crack widths above nodal supports is ***0.12 mm***.

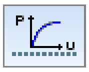


To determine the deflection of the cracked slab you need to run ***Nonlinear static analysis*** considering load combination ***Co.#1 (SLS)***.

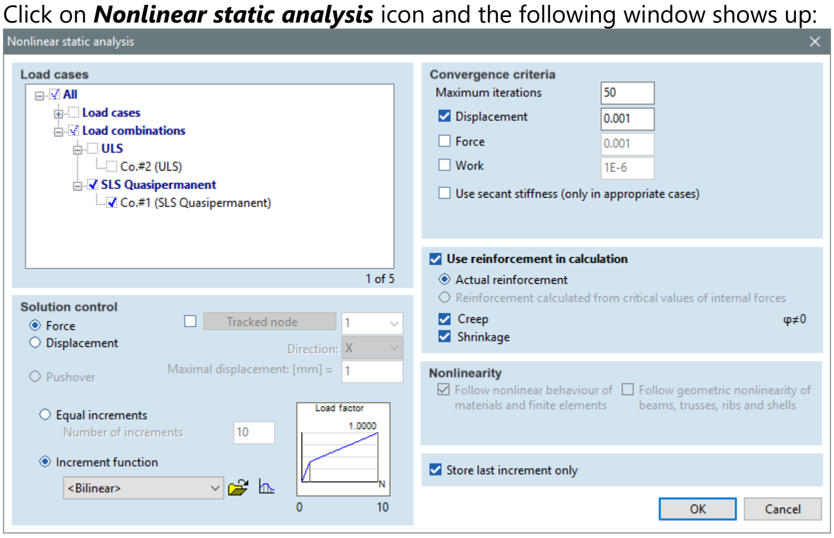
Static

Click on ***Static*** tab to run analysis.

Nonlinear static analysis



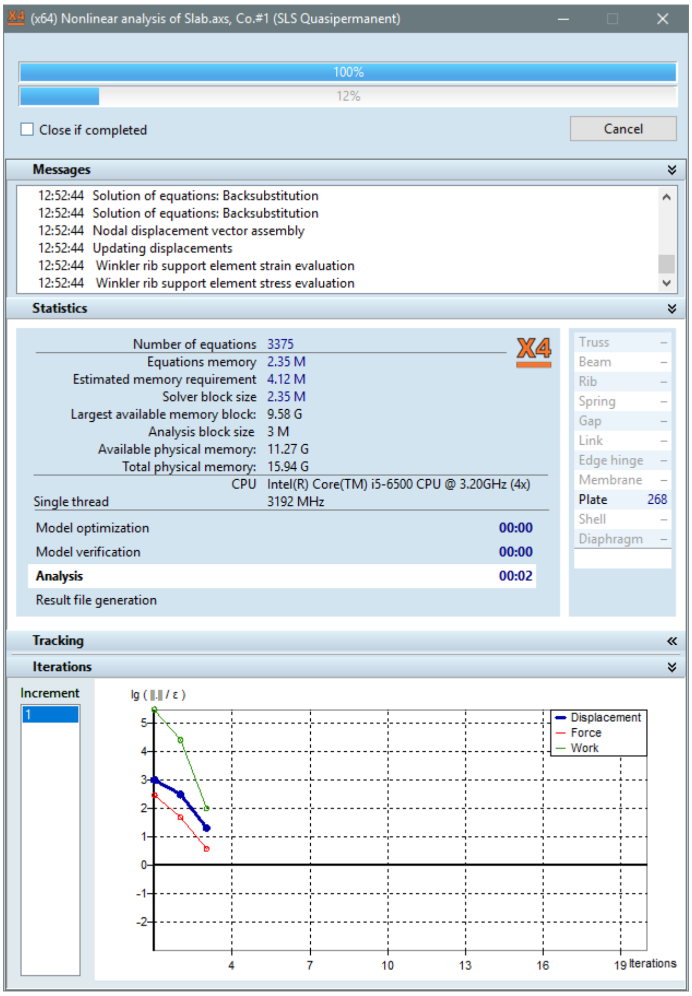
Click on ***Nonlinear static analysis*** icon and the following window shows up:



At ***Load cases*** select ***Co.#1 (SLS)*** load combination and check ***Actual reinforcement*** to use in calculation, and let us consider the effect of ***Creep*** and ***Shrinkage***. By clicking on ***OK***, then the analysis starts:

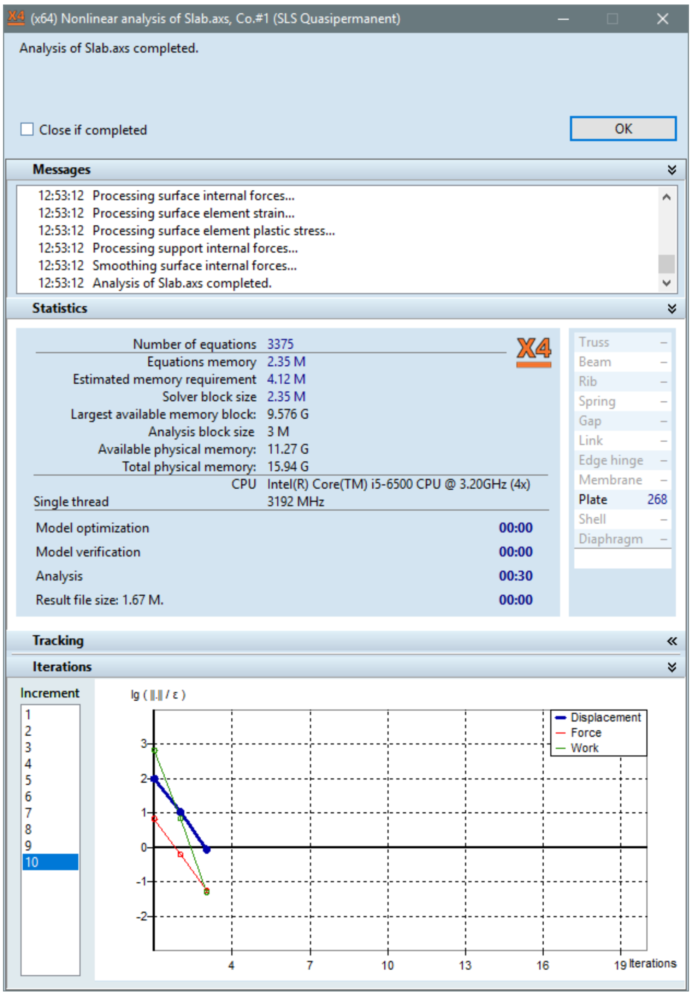
Statistics

Click on ***Messages*** to see more information about the analysis durng calculation:



Iterations

Click on ***Iterations*** to see how the calculations are converging for each increment:



After click on ***OK,*** then program automatically activates vertical deformations ***ez [mm]*** in ***Isosurfaces 2D*** display mode in load combination ***Co.#1*** with nonlinear analysis. Switch of ***Write values*** to ***Nodes*** at ***Numbering*** speed button, then the following will be displayed:

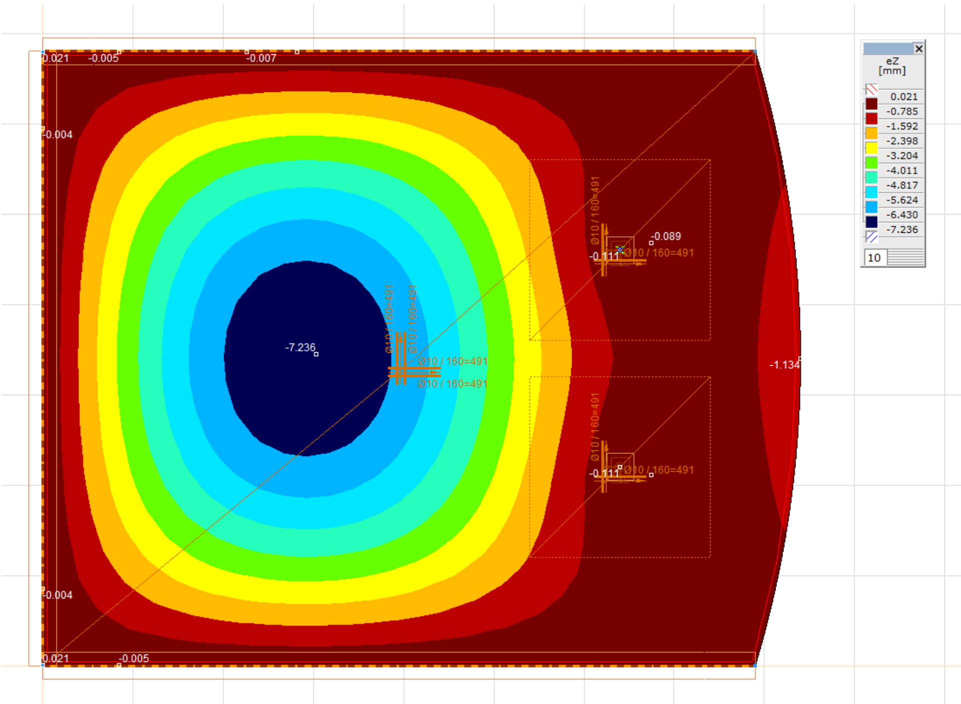
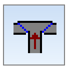
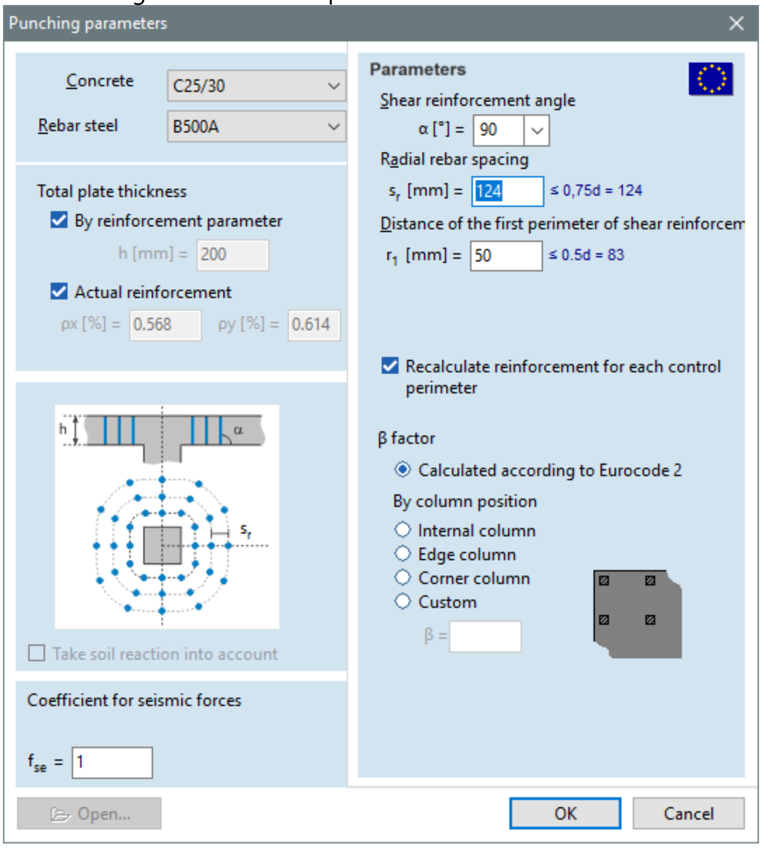


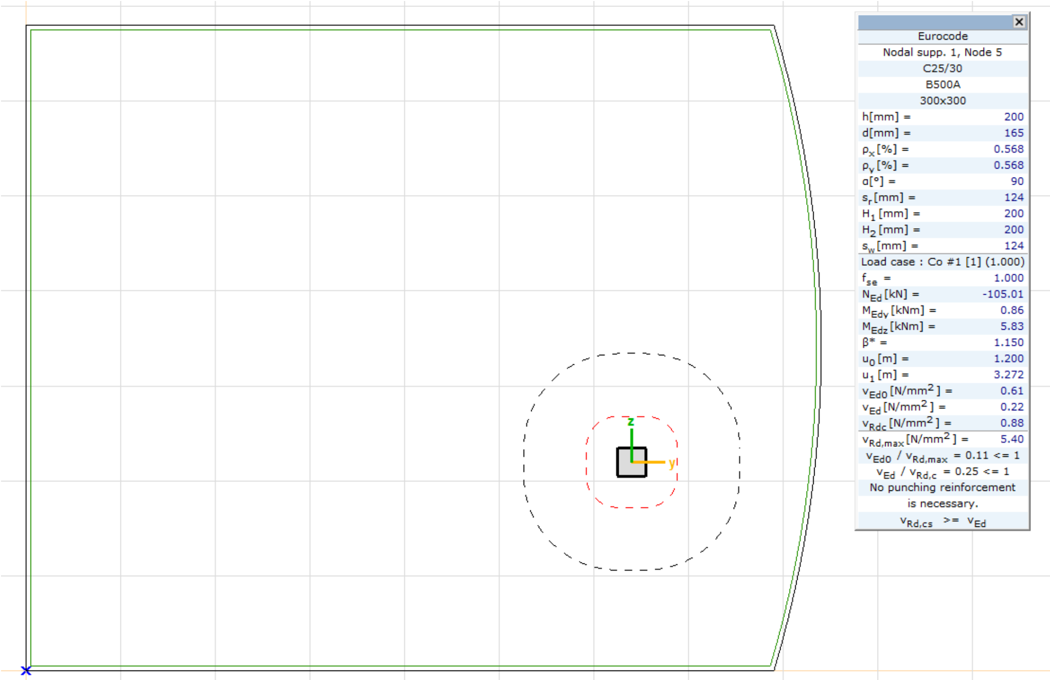
Plate punching analysis



To check shear punching click on ***R.C. Design*** tab and select load combination ***Co.#2 (ULS)***. Click on ***Plate punching analysis*** icon then select upper nodal support and click on ***OK***.
The following window shows up:

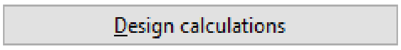


The material types (for concrete or rebar) are set automatically applying the parameters set before. Programs applies the current design code and national specifications. Set the parameters shown above and click on ***OK*** to close the window. We get the following results:

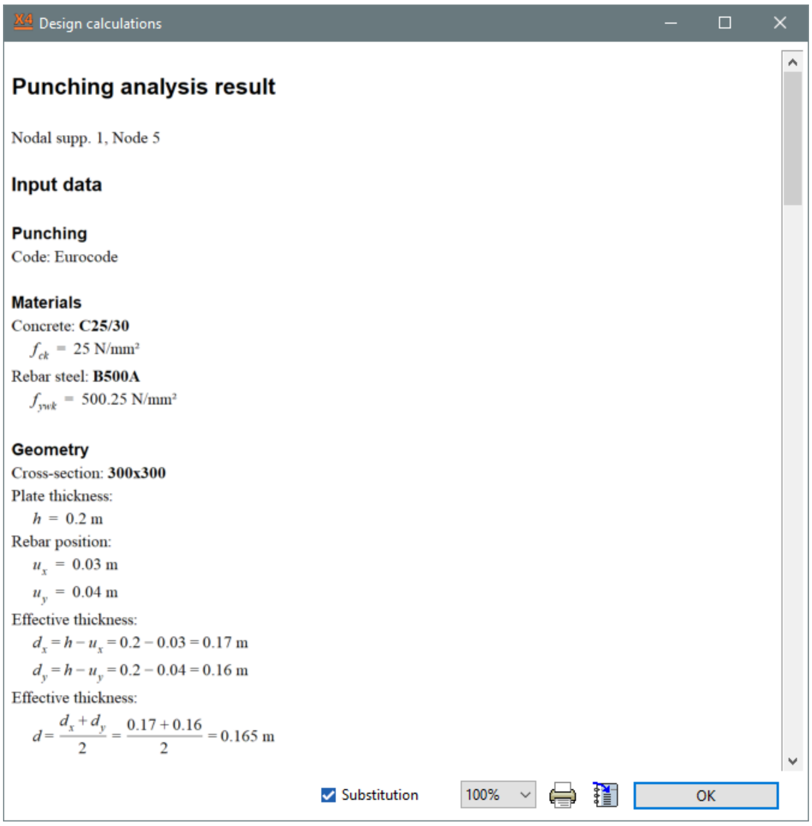


According to the check, ***No punching reinforcement*** is needed.

Design calculations



To see detailed calculations about the punching analysis, click on ***Design calculations*** button:



Click on ***OK*** to exit from ***Design calculations*** and click on ***Close*** to exit from ***Plate punching analysis*** window.

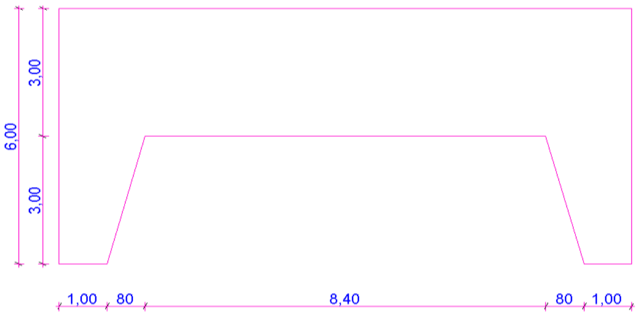
Intentionally blank page

4. MEMBRANE MODEL

4.1. Geometry definition using parametric mesh

Objective

The objective of the analysis is to determine the internal forces and reinforcements of the following wall structure. The loads and support conditions will be determined later.

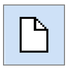


The wall thickness is 200 mm, assume material C25/30 for concrete and B500A for the reinforcement. Analyse the structure according to the Eurocode 2 standard.

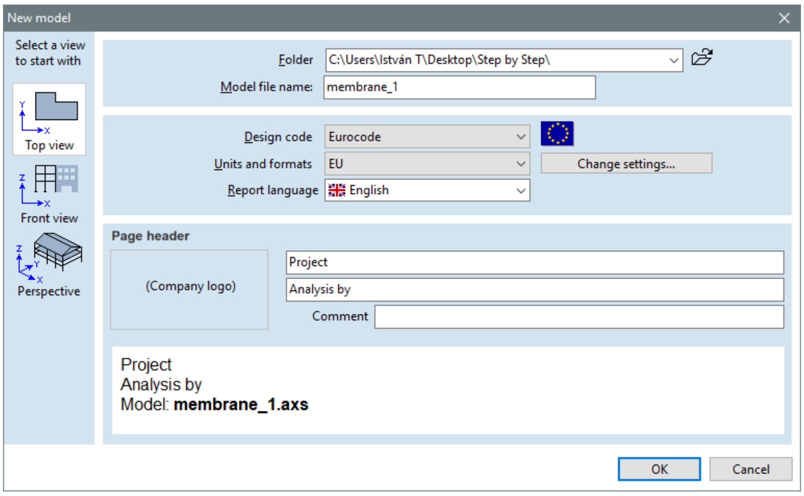
Start

Start ***AxisVMX4*** by double-clicking on ***AxisVMX4*** icon in its installation folder, found in ***Start*** – ***Programs*** menu.

New

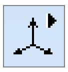


Create a new model by clicking on ***New*** icon. In the dialogue window replace ***Model file name*** with '***membran_1***', select ***Eurocode*** from ***Design code***s and set ***Unit and formats*** to ***EU***.

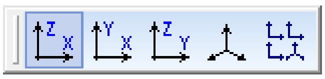


(Starting workplane (***X-Z*** plane/front view) can also be set in this window if selected from the list in the left side.)
Click ***OK*** to close the dialog window.
In the following the geometry of the wall structure is created by editing toolbar.

View

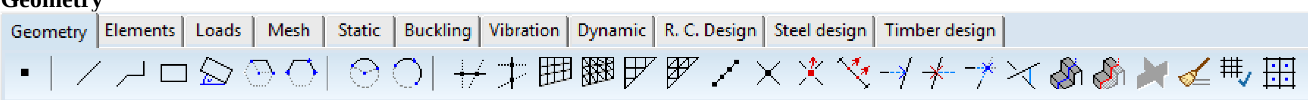


Check view (workplane) of the model when starting a new model. On the left side of the main window find ***Views*** icon, open it with moving the cursor over the icon and select ***X-Z*** view. The actual view is presented by the global coordinate system sign at the left bottom corner of the main window.

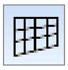


Define of geometry - Geometry

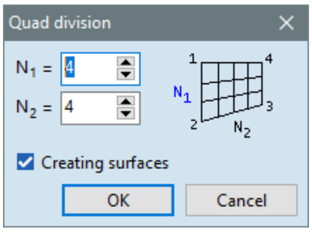
Select ***Geometry*** tab to define the geometry and structural properties of the beam. On the tab the icons of the available functions are displayed.



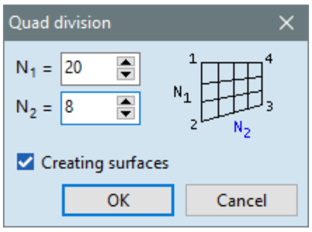
Quad division



Create the geometry by ***Quad division*** function. On ***Geometry*** tab click on its icon, the following shows up:



To define the top part of the wall type in ***20*** for the value of ***N_1***A and ***8*** for ***N_2***:

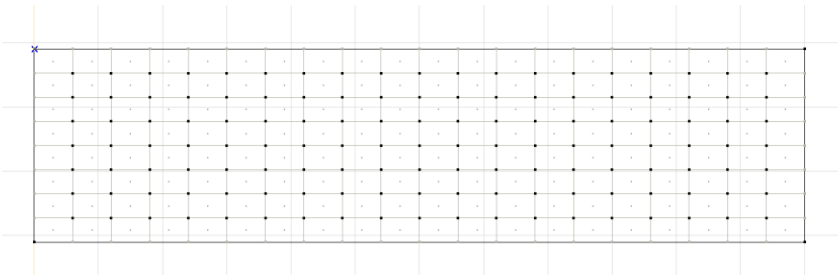


Close panel with ***OK*** and specify the corner points of the top 'body' of the wall. The nodes can be placed by editing tools or could be determined by their coordinates.

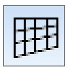
Geometry definition using coordinates:
To determine the starting point of the line (***X=0***, ***Y=0***, ***Z=3.00***), press ***x*** button, then the cursor jumps to the field of ***x*** coordinate on the ***Coordinates*** panel, then enter ***0***. After press ***y*** button and enter ***0***. Similarly, specify ***z*** value (***3.00***), finally close the input with enter key.
The other coordinates are specified as follows:

x	12	y	0	z	0	<**Enter**>
x	0	y	0	z	3	<**Enter**>
x	-12	y	0	z	0	<**Enter**>

Press mouse right button and from the quick menu select ***Cancel*** to finish the drawing process, then the following result can be seen:

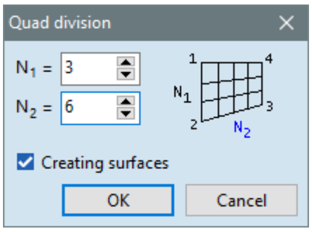


Quad division



To define the 'legs' of the wall structure activate again the function of ***Quad division***. To define the top part of the wall type in ***20*** for the value of ***N_1*** and ***8*** for ***N_2***:

In the window set value of ***3*** for ***N_1*** and ***6*** for ***N_2***:



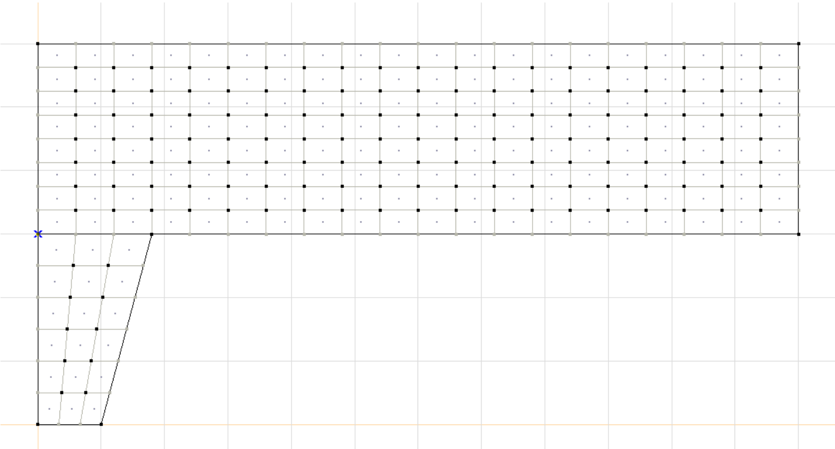
Close the panel with ***OK*** and specify the four corner nodes of the leg on the left side.

The coordinates are specified as follows:

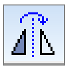
x	0	y	0	z	-6	<**Enter**>
x	1	y	0	z	0	<**Enter**>
x	0.8	y	0	z	3	<**Enter**>
x	-1.8	y	0	z	0	<**Enter**>

Press ***Esc*** to finish data input.

The following geometry can be seen as a result:

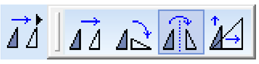


Mirror



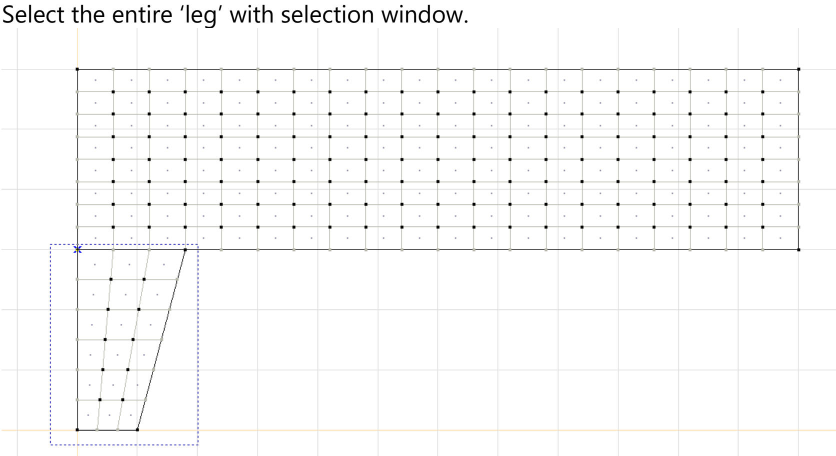
Mirror the previously drawn 'leg' to the symmetry axis of the wall (***X=6 m***).

The mirror function can be found on the vertical toolbar on the left side.

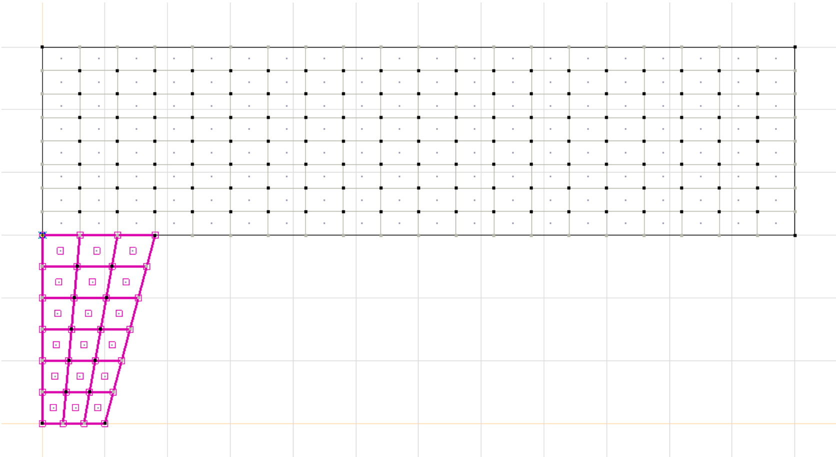


By clicking on the icon, the selecting panel activates:

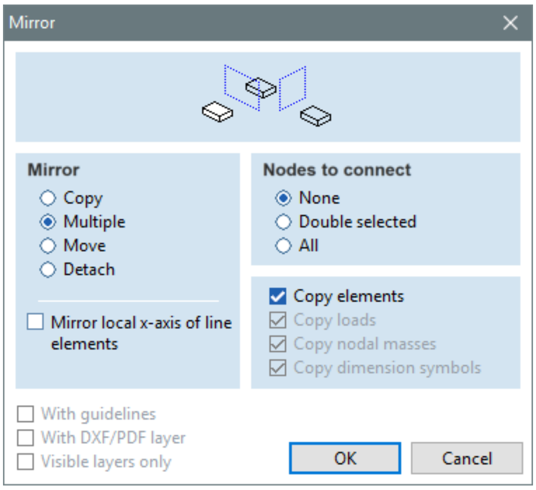
Select the entire 'leg' with selection window.



The selected elements will be highlighted:

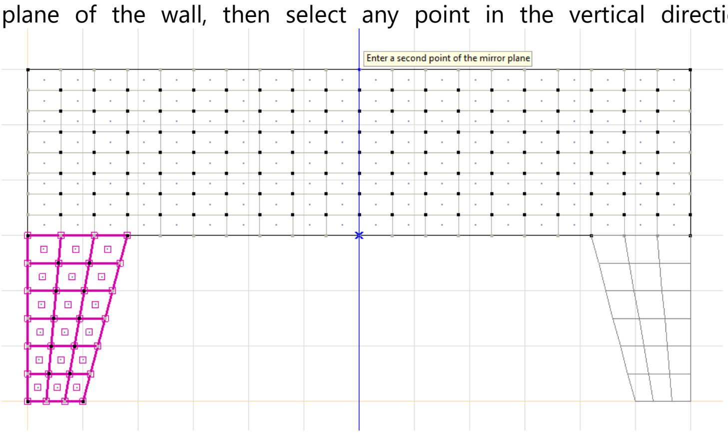


Finish the selection with ***OK*** and the following dialogue panel will be displayed:

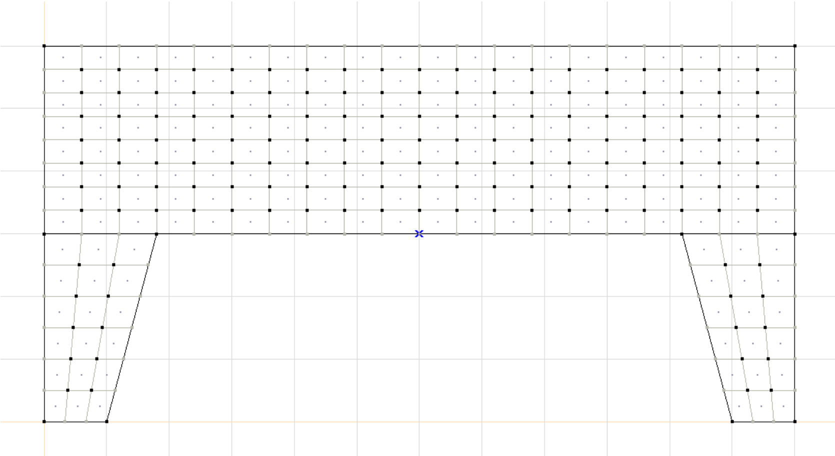


Select ***Multiple Mirror*** type and set ***None*** for ***Nodes to connect***.

After clicking on ***OK***, the mirror plane should be specified. First select any point on the symmetric plane of the wall, then select any point in the vertical direction above or below that point.

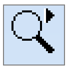


As a result of mirroring the following can be seen:

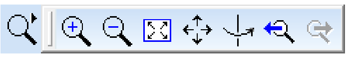


The geometry of the wall has been successfully created.

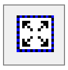
Zoom



Let's zoom to the structure. Move the cursor over the **Zoom** icon and the zoom icon bar pops up:

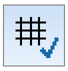


Zoom to fit

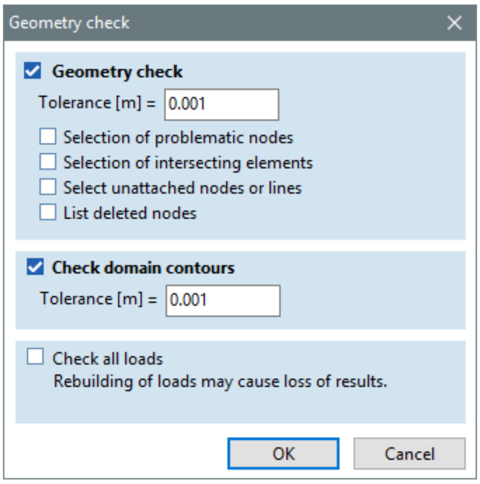


Click on ***Zoom to fit*** icon for better view.

Geometry check

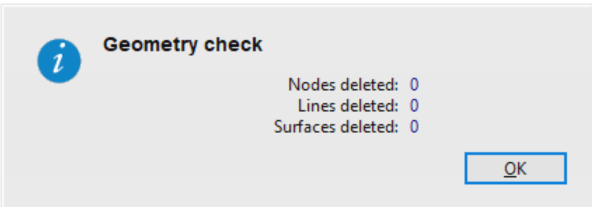


Click on ***Geometry check*** icon on the ***Geometry*** tab to filter the errors in geometry. In the ***Geometry check*** panel, the maximum tolerance (distance) can be specified to merge nodes.



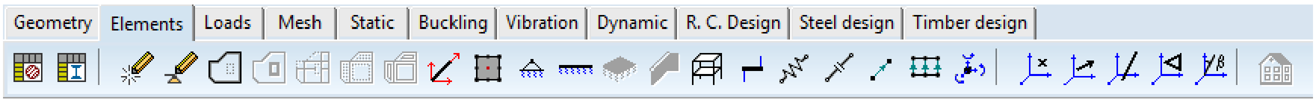
Switch off the option of ***Select unattached nodes or lines***, then close the window with ***OK***.

After the geometry check a summary of actions shows:

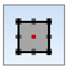


Elements

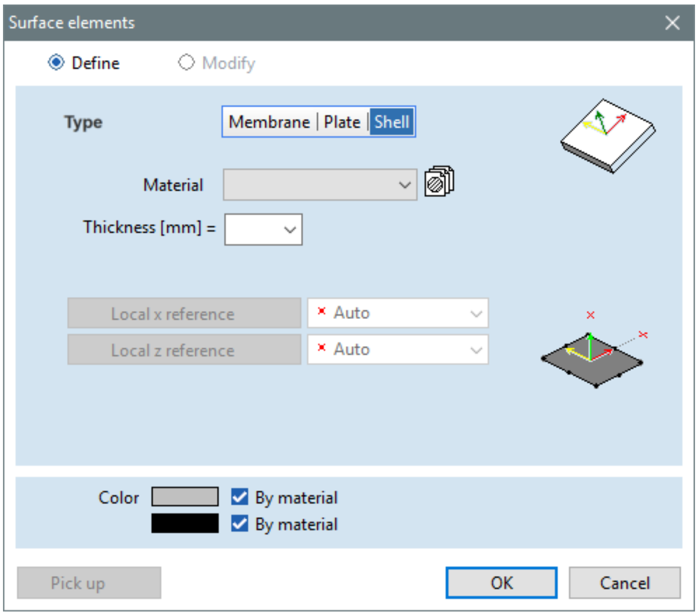
To define ***Surface elements,*** change tab to ***Elements***:



Surface elements

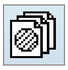


Click on ***Surface elements*** icon and on selecting toolbar, click on ***All*** (*), finally close the panel with ***OK***. After the ***Surface elements*** window shows up:

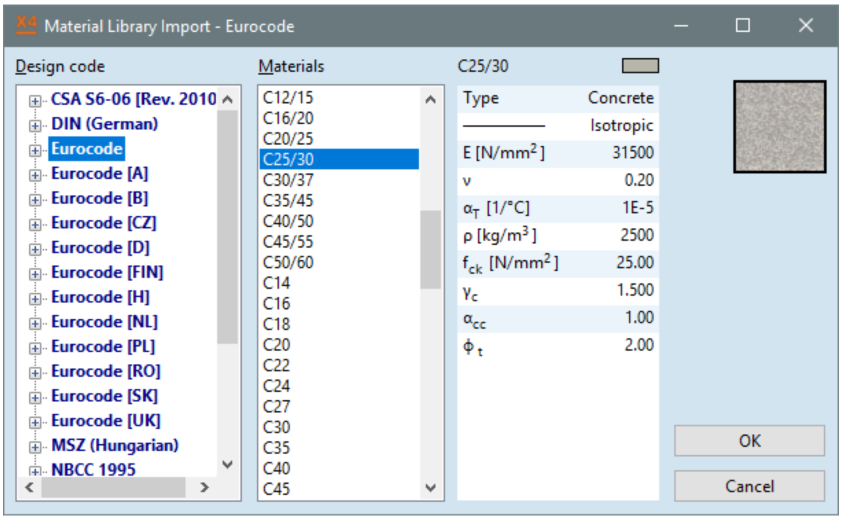


Set ***Type*** of the element to ***Membrane (plane stress)***.

Material library import

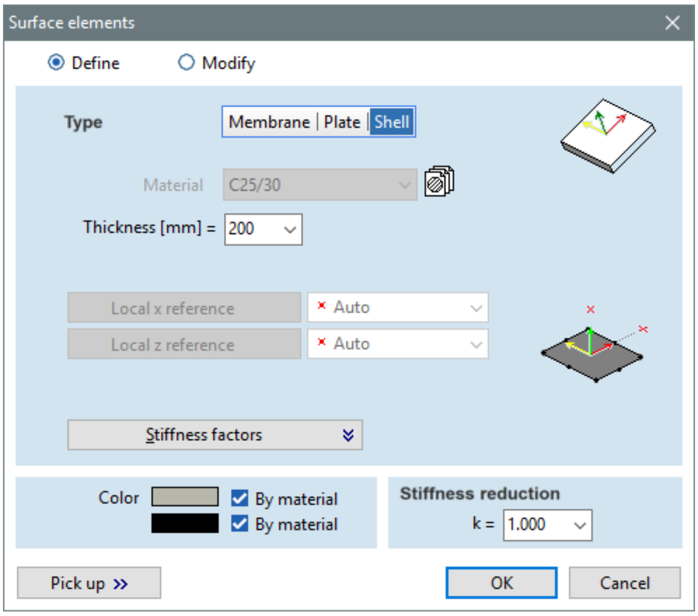


Next to the label of ***Material***, click on ***Material library import*** icon, then the following window shows up:



Select ***C25/30*** from the material list, then confirm with ***OK***.

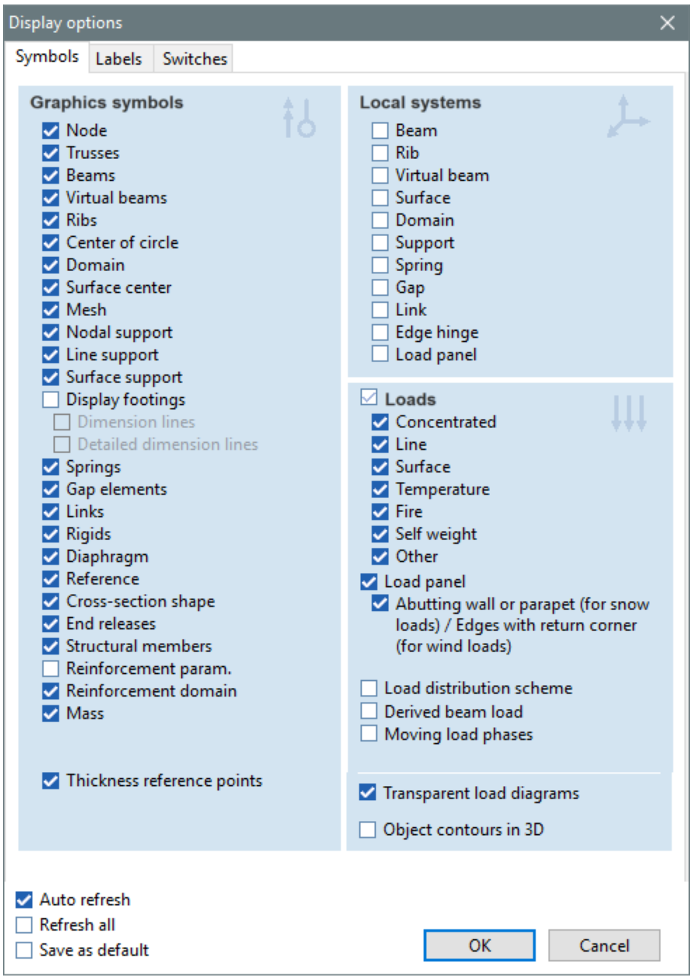
Thickness



Type in the ***Thickness*** edit box ***200*** [mm], then close the dialog with ***OK***.

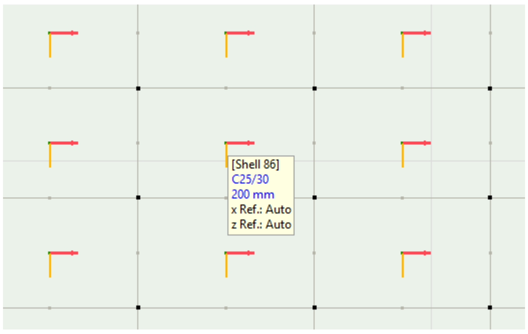
Display options

To view the local coordinate system of the ***Surface elements,*** click on ***Display Options*** icon in the icons menu bar on the left side. The following dialogue window is displayed:



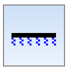
Check ***Surface*** checkbox in the ***Local Systems*** panel group. Accept the change with ***OK***.

If the ***Mesh***, ***Node***, ***Surface centre*** is switched on among the ***Graphics Symbols*** in the ***Display Options***, you can see that the program uses 9-node membrane elements. These 9 nodes are the 4 corners, 4 midpoints and the centre point of surface element. If you move the cursor over the surface centre symbol (a filled square), a hint window is displayed with the properties of the surface element: its tag, material, thickness, mass and references (***x***, ***z***), as shown on the next picture:

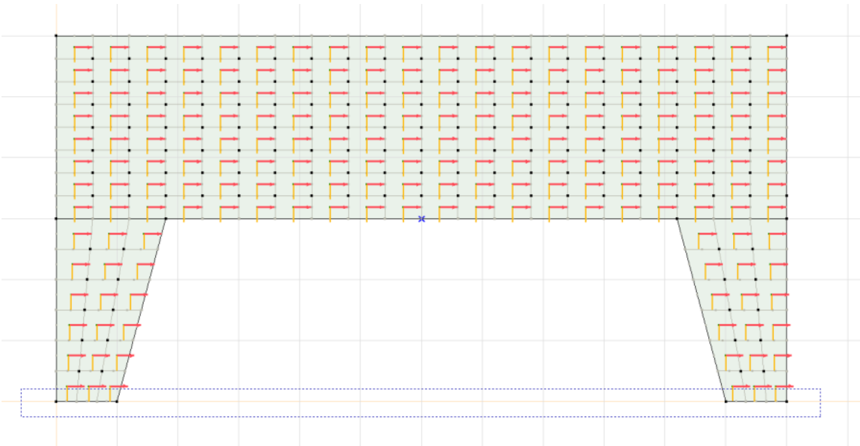


The red line shows the ***x*** axis of the local coordinate system, the yellow one the ***y*** axis and the green one the ***z*** axis.

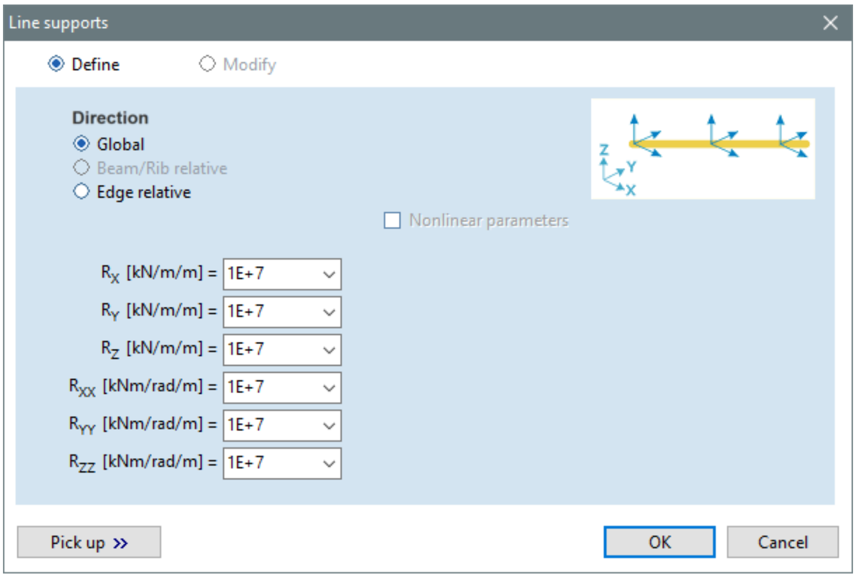
Line support



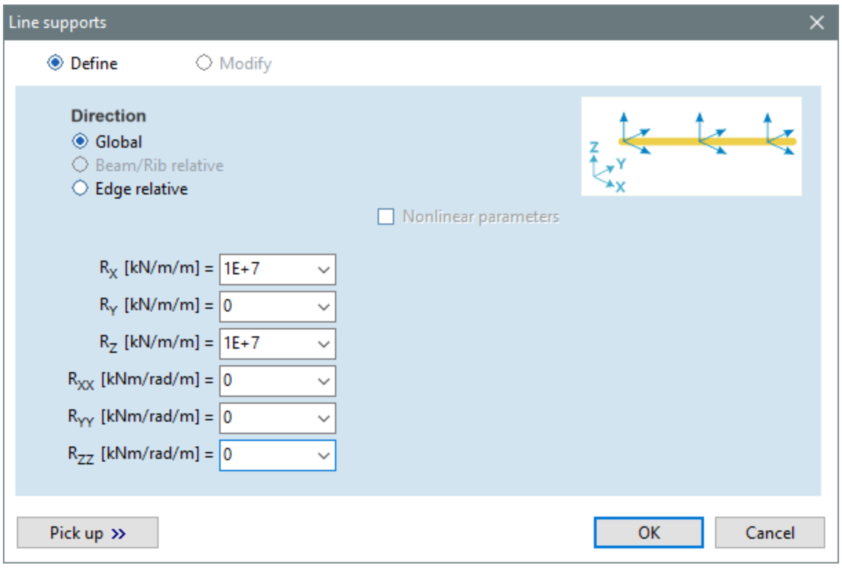
To create supports, click on ***Line support*** icon and select the bottom lines of the 'legs' with a selection box. We suppose pinned line support along these edges.



Press ***OK*** to go on, the following window shows up:



To define a pinned support, use the following settings:

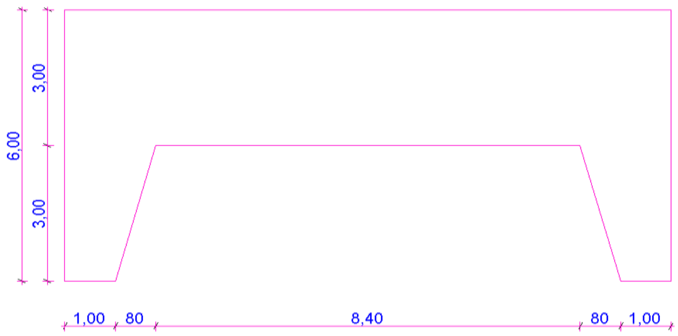


With this final step, the finite element definition of the wall structure has been completed.

4.2. Geometry definition using domains

Objective

The objective of the analysis is to determine the internal forces and reinforcements of the following wall structure. The loads and support conditions will be determined later.

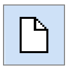


The wall thickness is 200 mm, assume material C25/30 for concrete and B500A for the reinforcement. Analyse the structure according to the Eurocode 2 standard.

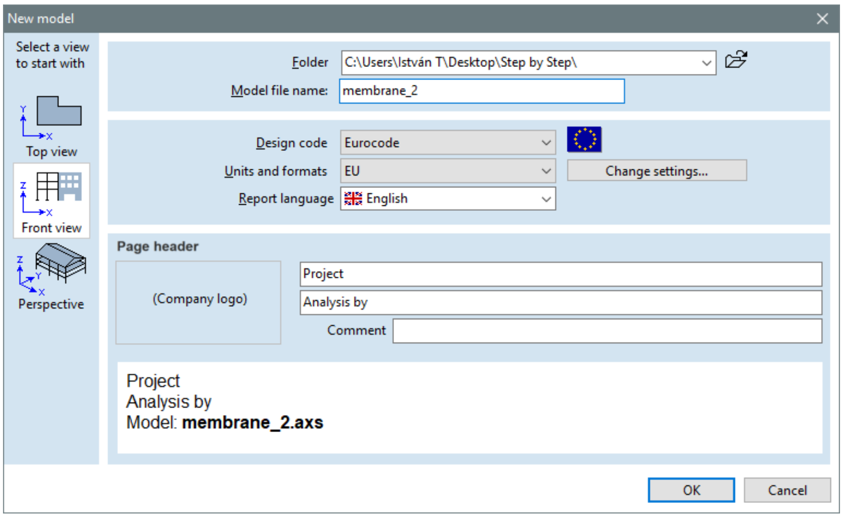
Start

Start ***AxisVMX4*** by double-clicking on ***AxisVMX4*** icon in its installation folder, found in ***Start*** – ***Programs*** menu.

New



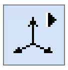
Create a new model by clicking on ***New*** icon. In the dialogue window replace ***Model file name*** with '***membran_2***', select ***Eurocode*** from ***Design code***s and set ***Unit and formats*** to ***EU***.



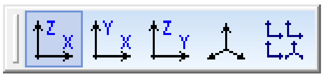
(Starting workplane (***X-Z*** plane/front view) can also be set in this window if selected from the list in the left side.)
Click ***OK*** to close the dialog window.

In the following the geometry of the wall structure is created by editing toolbar.

View

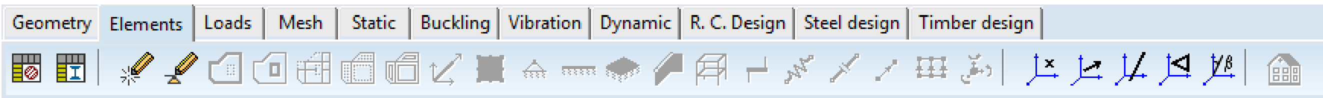


Check the view (workplane) of the model when starting a new model. On the left side of the main window find ***Views*** icon, open it with moving the cursor over the icon and select ***X-Z*** view. The actual view is presented by the global coordinate system sign at the left bottom corner of the main window.



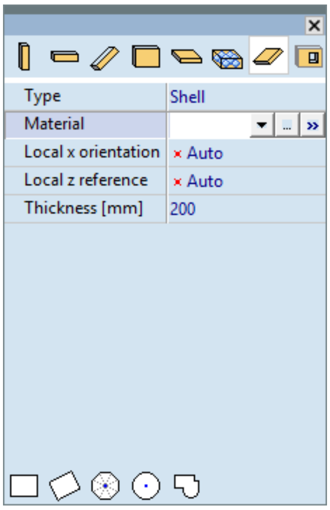
Define of geometry - Elements

Select ***Geometry*** tab to define the geometry and structural properties of the beam. On this tab the icons of the available functions are actives.

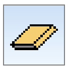


Draw objects directly

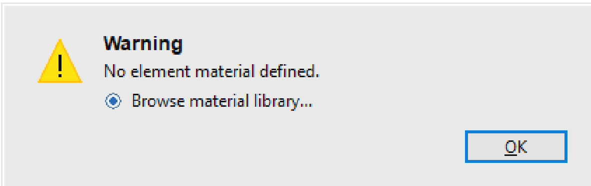
By clicking on ***Draw objects directly*** icon brings up the following window:



Domain

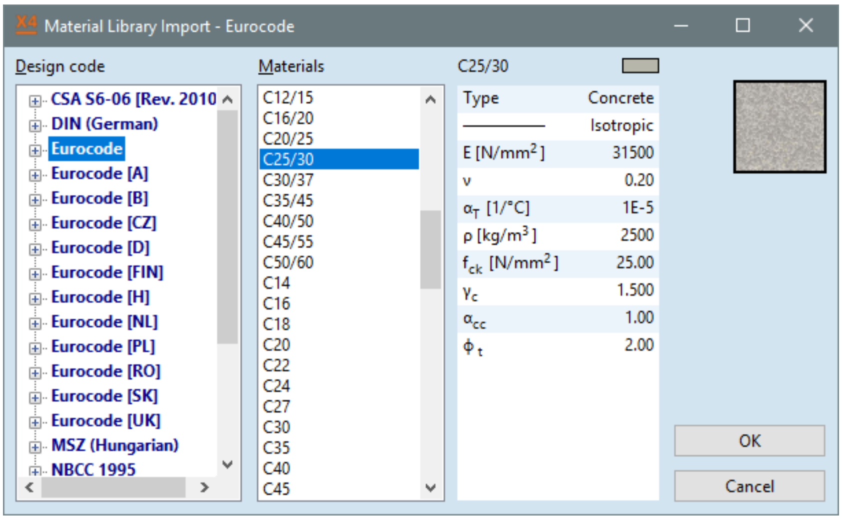


Click on ***Domain*** icon (the second one on the right among the top row icons). Choose it even if this is the active icon because of the sequences of steps. The following dialogue panel shows after clicking:



Material library import

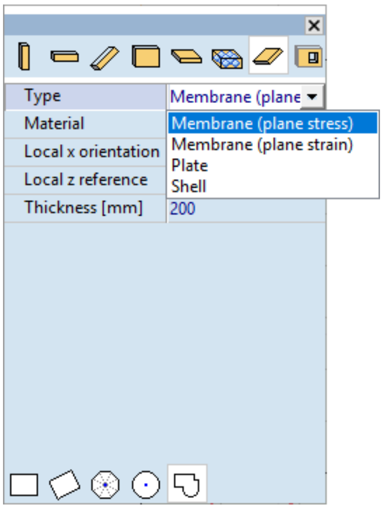
The following, ***Cross-section editor*** window shows after clicking.



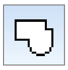
Select ***C25/30*** from the ***Materials*** list, then confirm with ***OK***.

Type

On the following panel, set ***Type*** to ***Membrane (plane stress)***:

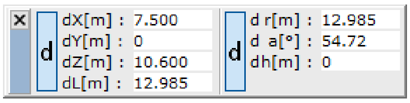


Complex slab



Click on **Complex slab** icon. User can draw the domain directly or define its coordinates.

Geometry definition using coordinates:
To determine the starting point of the polygon (***X=0***, ***Y=0***, ***Z=0***), press ***x*** button, then the cursor jumps to the field of ***x*** coordinate on the ***Coordinates*** panel, then enter ***0***. After press ***y*** button and enter ***0***. Similarly, specify ***z*** value, finally close the input with ***Enter*** key.
In the following, use relative coordinates to define the nodes of the domain. Press button ***d*** on the ***Coordinates*** panel. If it is pressed on the ***Coordinates*** panel the values can be specified relative to local origin (***dX***, ***dY***, etc...).

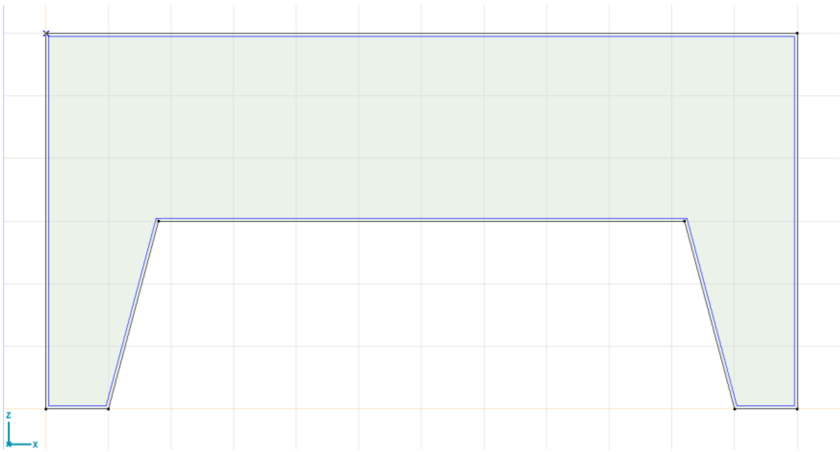


Continue defining next points of the domain with relative coordinates:

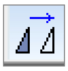
x	1	y	0	z	0	<**Enter**>
x	0.8	y	0	z	3	<**Enter**>
x	8.4	y	0	z	0	<**Enter**>
x	0.8	y	0	z	-3	<**Enter**>
x	1	y	0	z	0	<**Enter**>
x	0	y	0	z	6	<**Enter**>
x	-12	y	0	z	0	<**Enter**>
x	0	y	0	z	-6	<**Enter**>

Press ***Esc*** twice to exit from the command of object drawing (***Draw objects directly***).

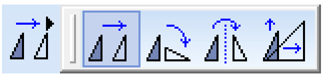
The following will be displayed in the main window:



Translate / Copy

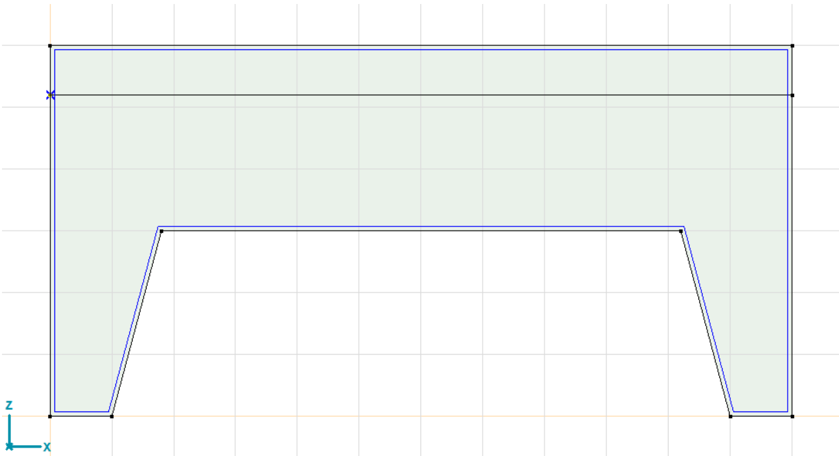


Click on **Translate** icon!

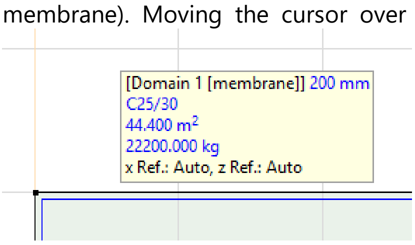


Select the top horizontal line and finish the selection with ***OK***. Choose ***Incremental*** from the ***Method*** panel, ***N=1***, ***Nodes to Connect***: ***None***, then close dialog window with ***OK***. Now specify the translation vector. Click on any empty place in the graphics area, then type in the following sequence:
x 0 y 0 z -0.8 , after press ***Enter***.

The following will be displayed in the main window:

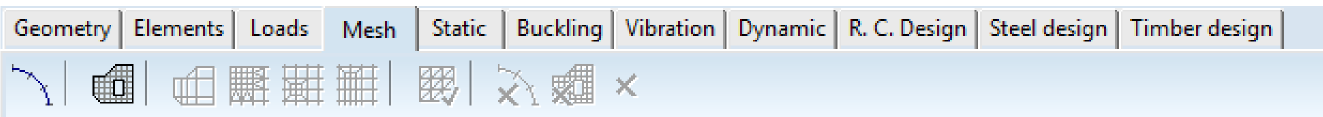


The blue line on the inner contour of the domain indicates the type of the domain (in our case it is membrane). Moving the cursor over it shows a hint window with the properties of the domain:

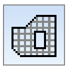


Mesh

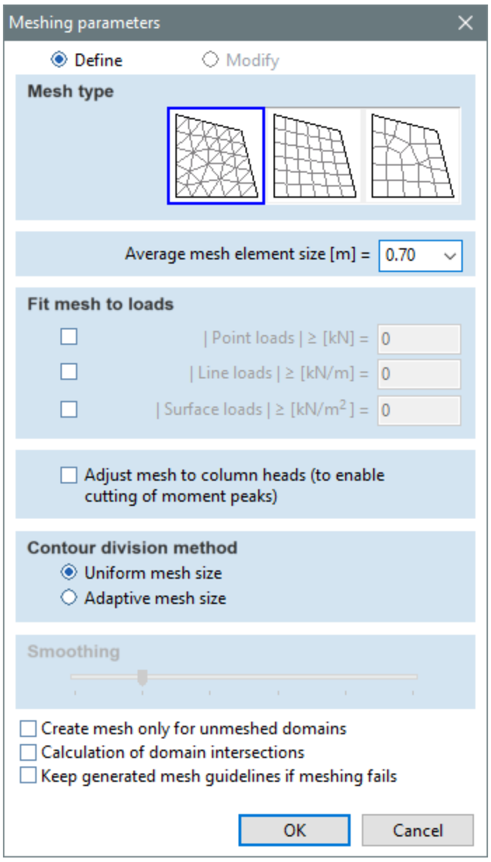
Click on ***Mesh*** tab for domain meshing.



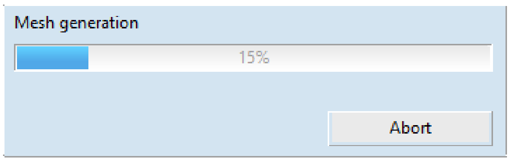
Domain meshing



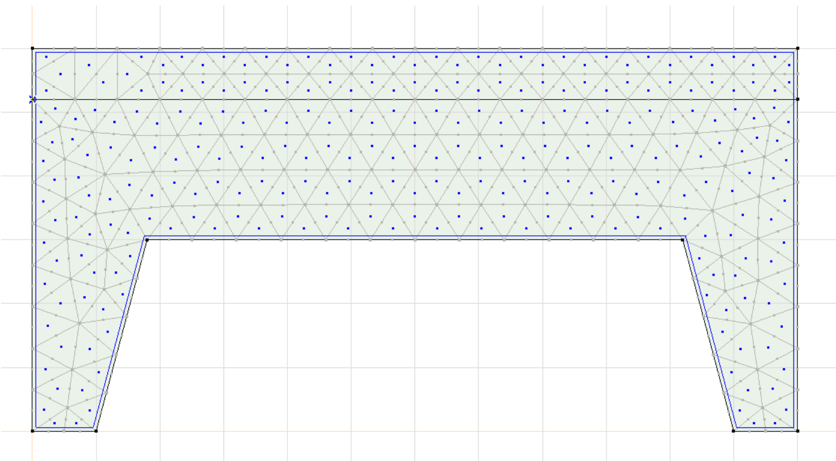
Click on ***Domain Meshing*** icon. Select the domain with the ***All*** (*) button and finish selection with ***OK***. The following dialogue window will be displayed:



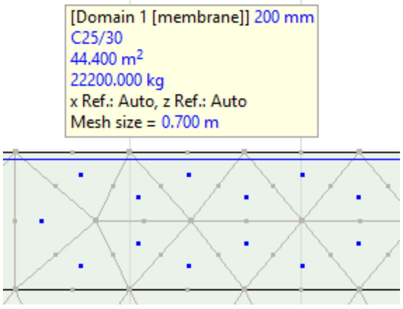
Type in **0.7 [m]** for the ***Average mesh element size***. After closing the dialog with ***OK***, the automatic mesh generation starts. The progress of ***Mesh generation*** is shown in the window:



When mesh generation is completed, the following will be displayed:

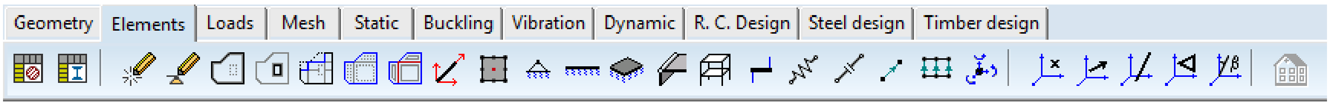


If you move the cursor on the symbol of the surface centre (a filled square), a hint window is displayed with the properties of the selected surface element: its tag, material, thickness, mass and references as shown in the next figure:

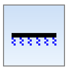


Elements

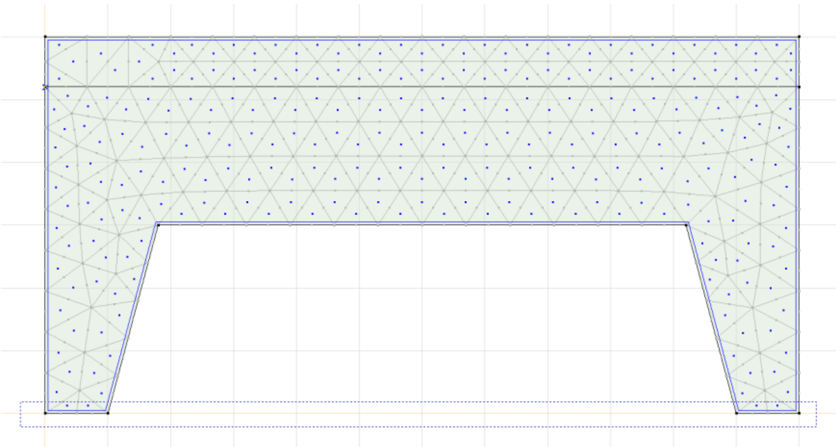
Return to ***Elements*** tab:



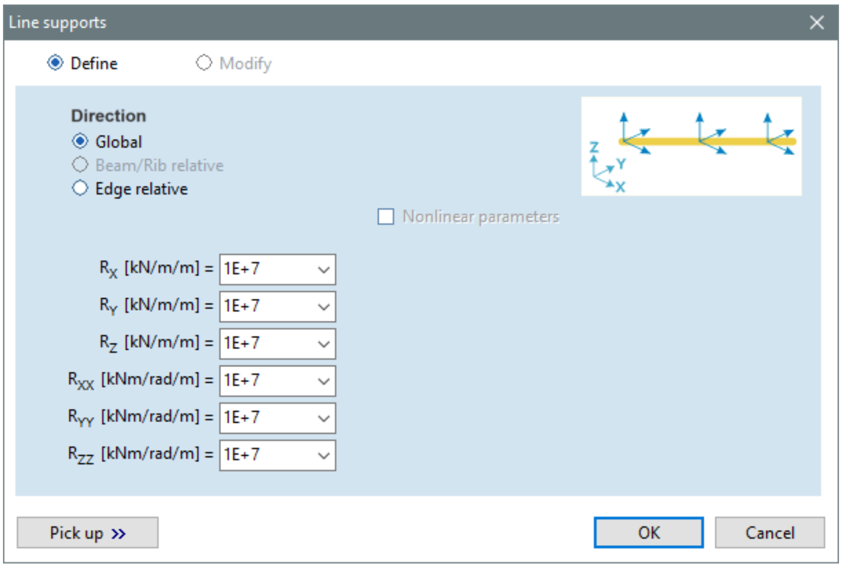
Line support



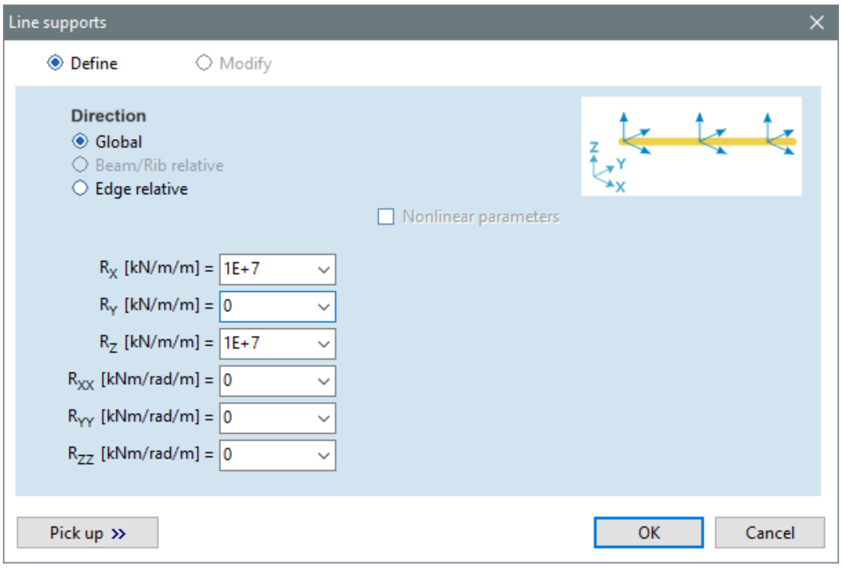
To create supports, click on ***Line support*** icon and select the bottom lines of the 'legs' with a selection box. We suppose pinned line support along these edges.



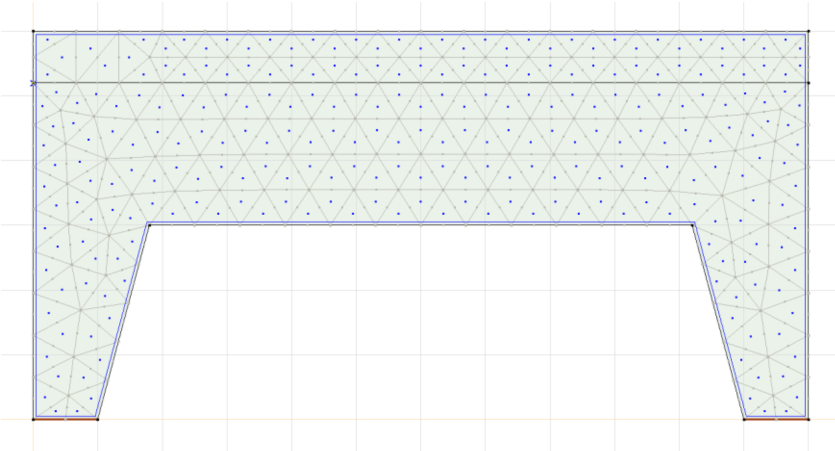
Press ***OK*** to go on, the following window shows up:



To define a pinned support, use the following settings:



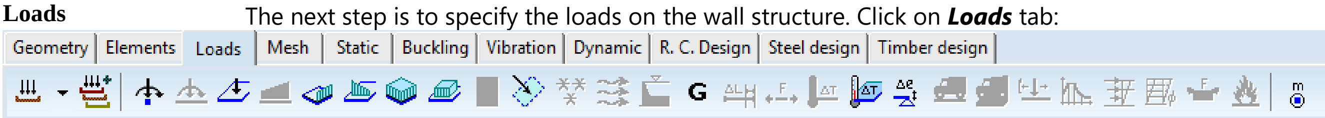
Close window with ***OK***, the following result can be seen:



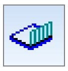
With this final step, the finite element definition of the wall structure has been completed. In the following, we proceed the analysis with this model (*Geometry definition using domains*).

Loads

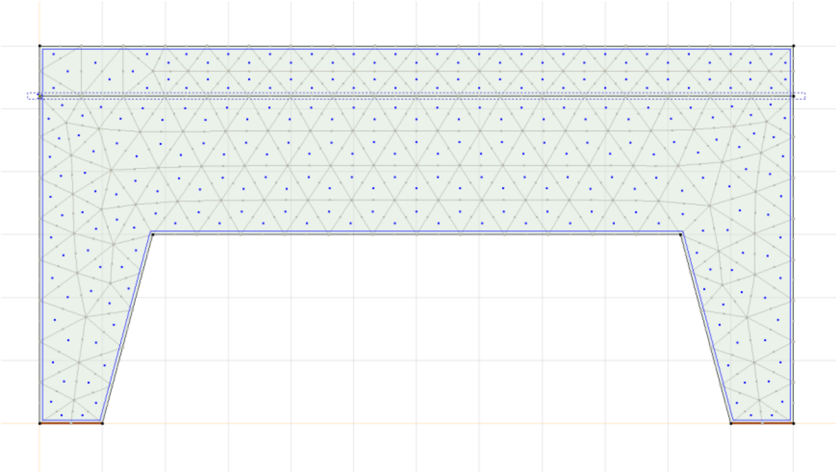
The next step is to specify the loads on the wall structure. Click on ***Loads*** tab:



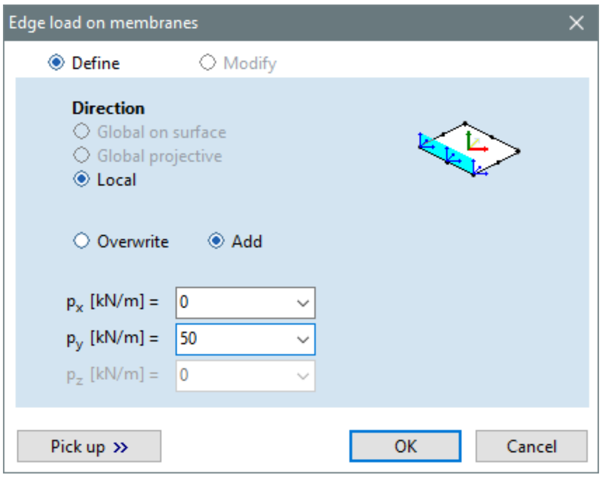
Surface edge loads



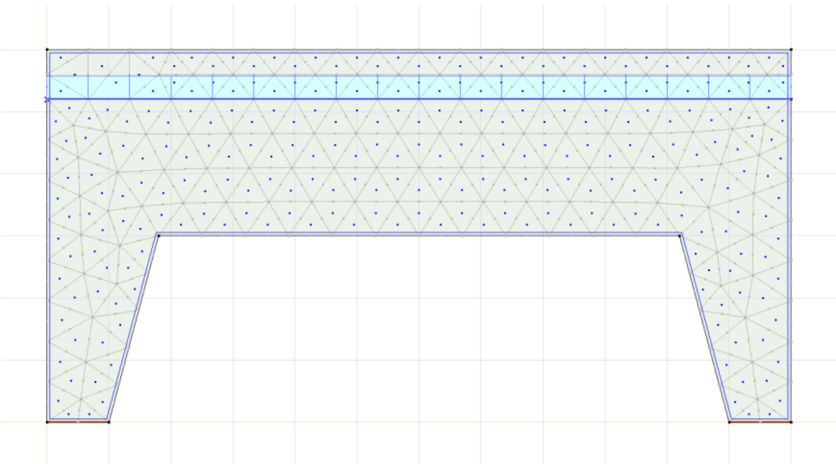
Assume a ***50 kN/m*** vertical distributed load. Click on ***Surface Edge Load*** icon, then select the second line from top (created with the translate command before):



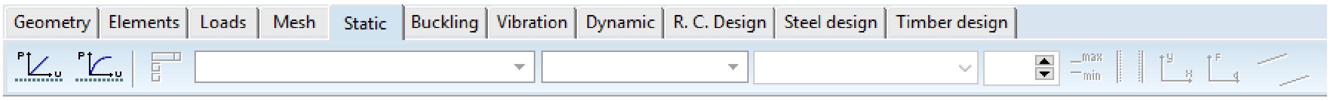
Finish the selection with ***OK*** and type in ***p_y [kN/m]*** to ***50***:



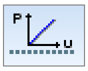
Press ***OK*** and the load is applied.
The following result is displayed:



Static The next step is the analysis and post processing. Click on ***Static*** tab:



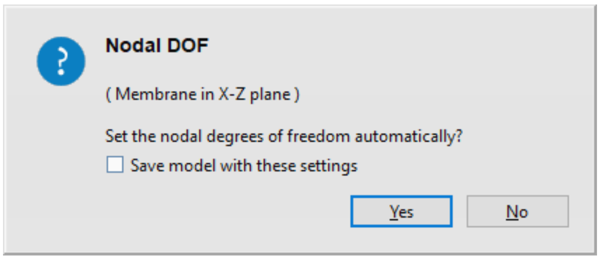
Linear static analysis



Click on ***Linear static analysis*** icon to start analysis.

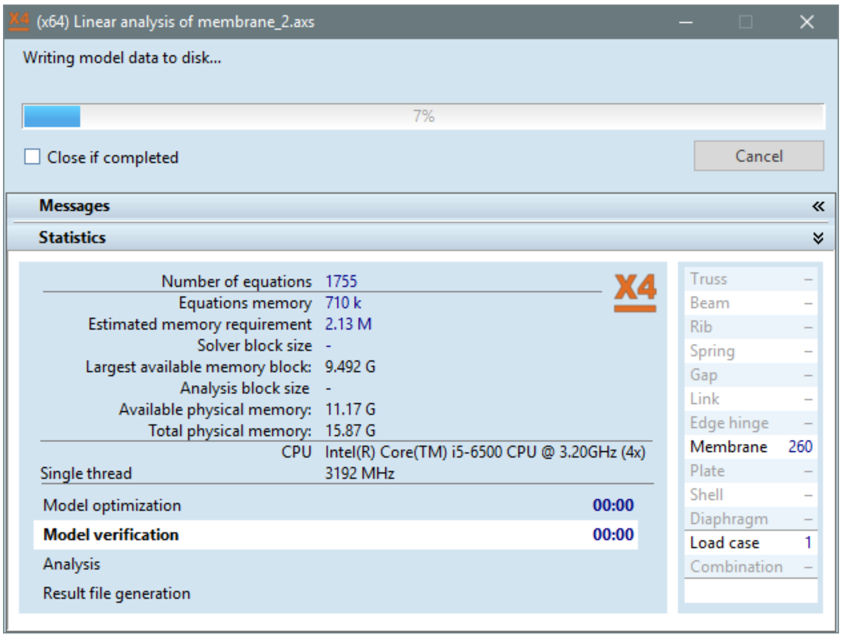
Nodal degrees of freedom

To proceed with the analysis, you must set degrees of freedom (DOF). The program checks the model and offers a setting:



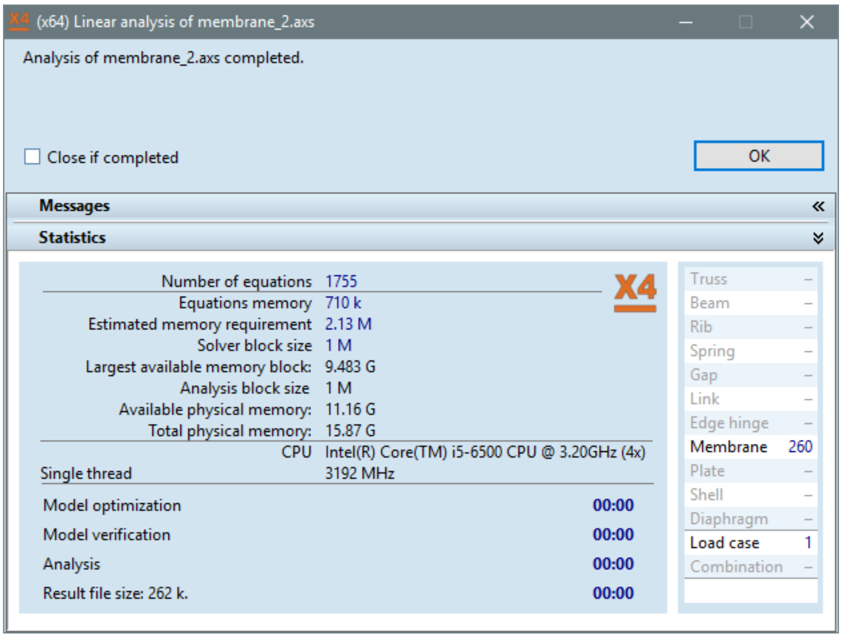
Check the ***Save model with these settings*** checkbox and degree of freedom settings will be saved. Accept ***Membrane in X-Z plane*** setting and close with ***Yes*** button.

Analysis continuous and the following progress bar shows up:



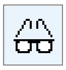
Messages, statistics

Click on ***Messages*** and ***Statistics*** to see more information about the analysis.
The following window will be displayed after analysis:



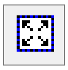
After click on ***OK***, the program automatically activates vertical deformations ***ez [mm]*** on ***Static*** tab in ***Isosurfaces 2D*** view.

Display options

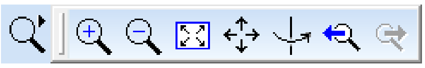


Switch off ***Loads*** in ***Display options*** on ***Symbols tab*** for cleaner view.

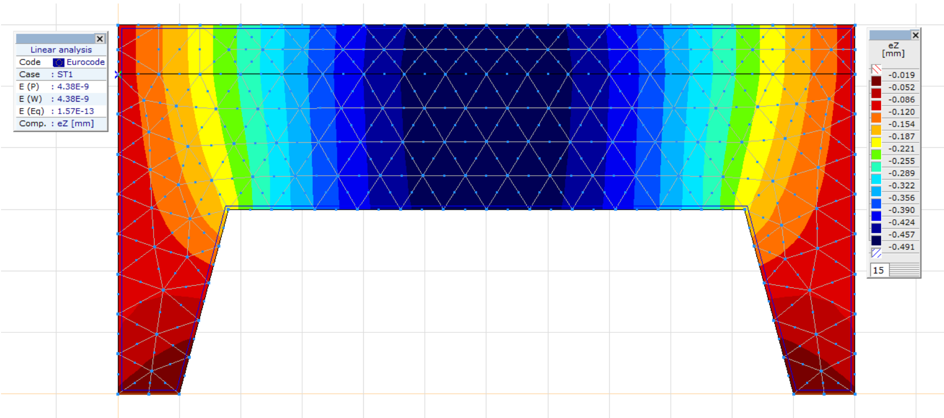
Zoom to fit



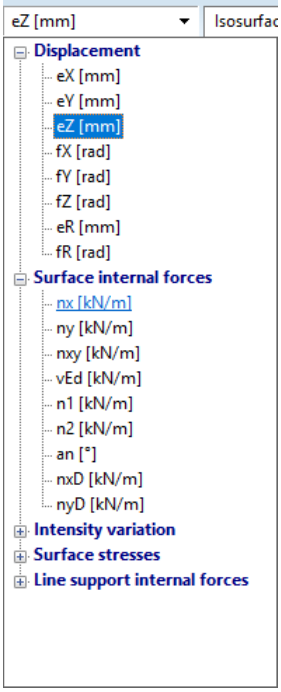
Click on ***Zoom to fit*** function for better view.



The following will be displayed:

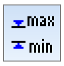


Select ***Surface internal forces – nx*** by clicking on arrow next to the text ***ez [mm]*** on ***Static*** tab:

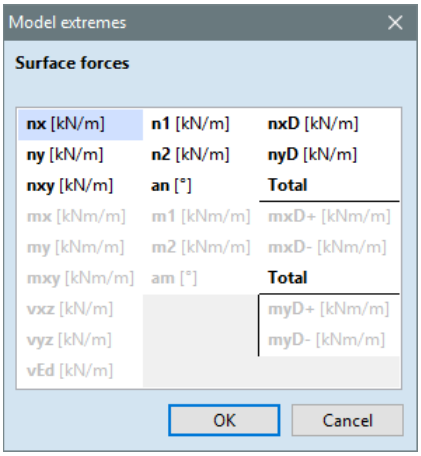


The ***nx*** diagram shows the membrane force in local ***x*** direction.

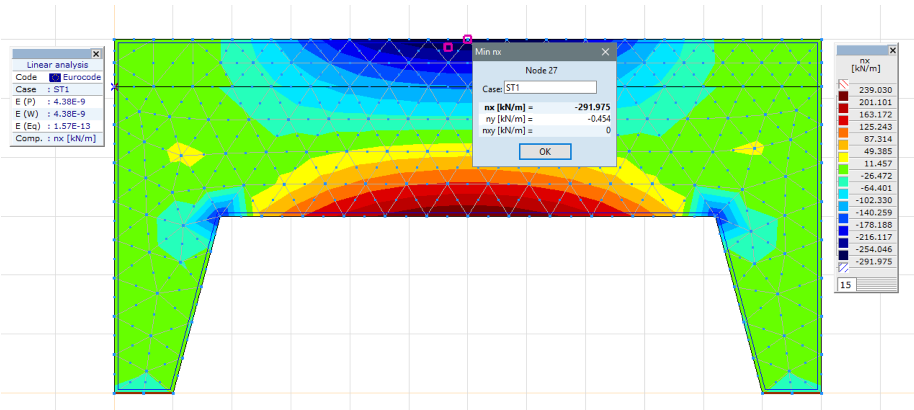
Min, max values



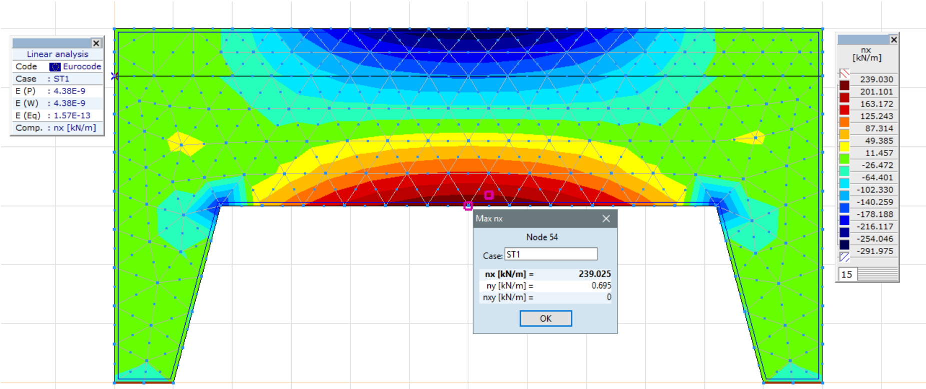
To find location of minimum and maximum values use ***Min, Max values*** function. By clicking on its icon shows following window:



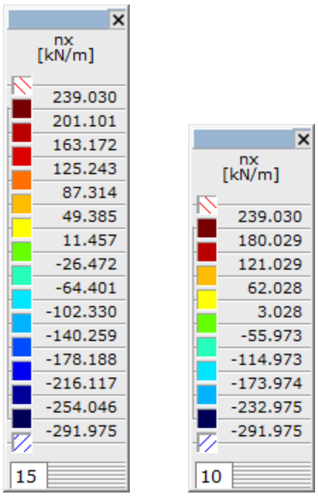
Select one of the ***Surface force*** components. Confirm with ***OK***, and the program shows the negative maximum value and its location as well.



After click on ***OK***, then the maximum positive value will be showed and its location will be marked.

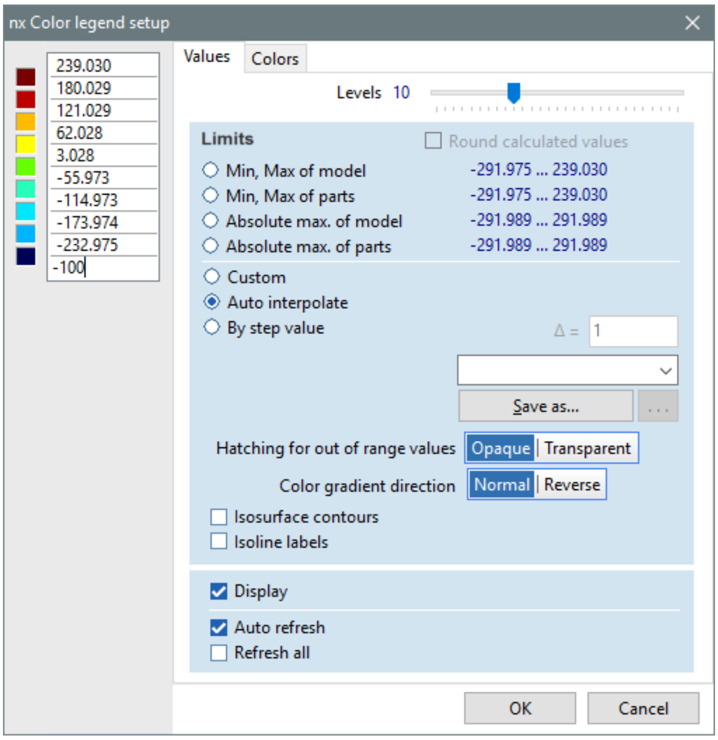


The ***Color legend*** shows boundary values of each colour. Adjust number of colors and boundary values by dragging up or down the bottom edge of the palette.



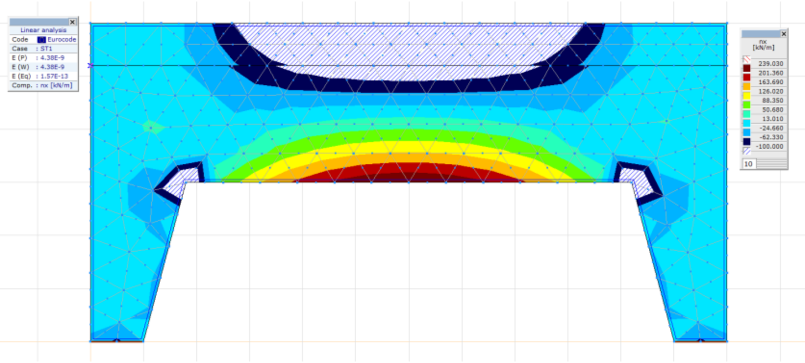
Color legend

Find areas where the ***nx*** internal force exceed value of ***-100 kN/m***. The boundary values can be set by clicking on ***Color legend***. In ***Color legend setup*** the values (next to the colors) can be edited. Click on the last value and change the default minimum maximum value (***-291.975***) to ***-100***:



Press ***Enter*** to finish data entry and switch on ***Auto interpolate.*** This option will recalculate the inner boundary values depending on the number of boundary levels.

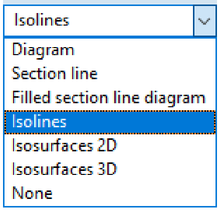
By clicking on ***OK***, the following will be displayed:



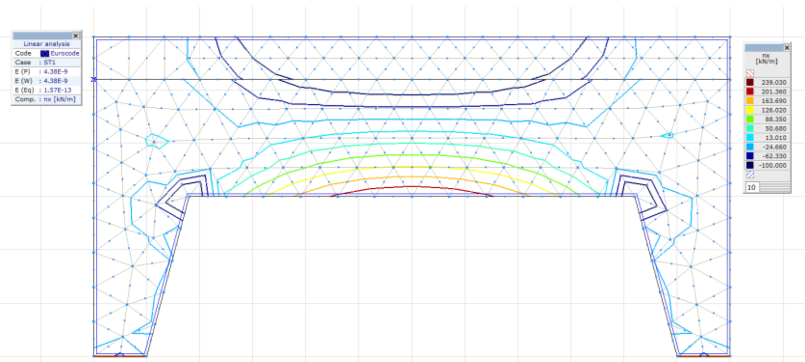
Areas with ***nx surface force*** exceeding ***-100 kN/m*** are hatched in blue.

Isolines

After, let us have a look at the internal forces in ***Isolines Display mode***. Click on arrow right to ***Isosurfaces 2D*** text and select ***Isolines*** from the list.

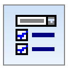


After selecting ***Isolines*** the following will be displayed:

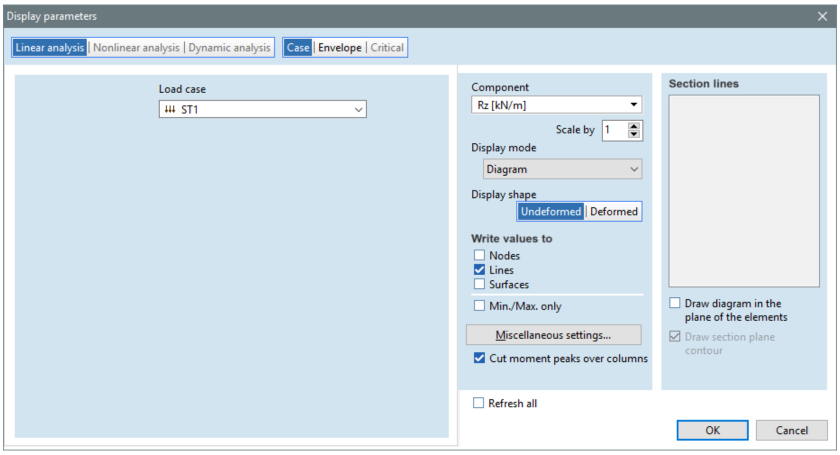


View internal forces of the supports. Select ***Rz*** component between ***Line support internal forces*** in the ***Result component*** combo box.

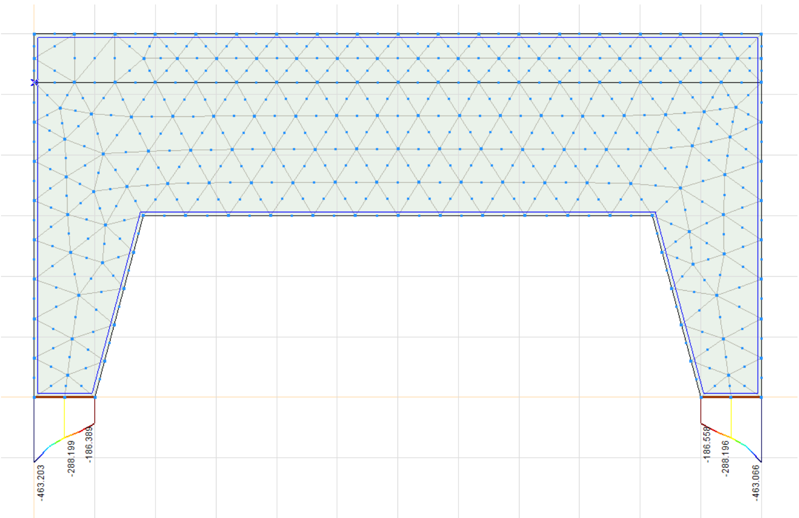
Result display parameters



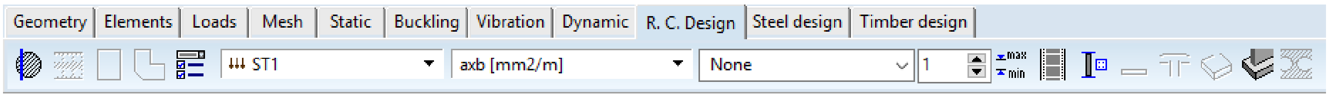
Click on ***Result Display Parameters*** icon, and check the ***Lines*** checkbox in the ***Write values to*** panel and set the ***Display mode*** to ***Diagram***:



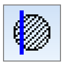
Close the dialog window with ***OK*** and the values of vertical support forces will be displayed on the screen:



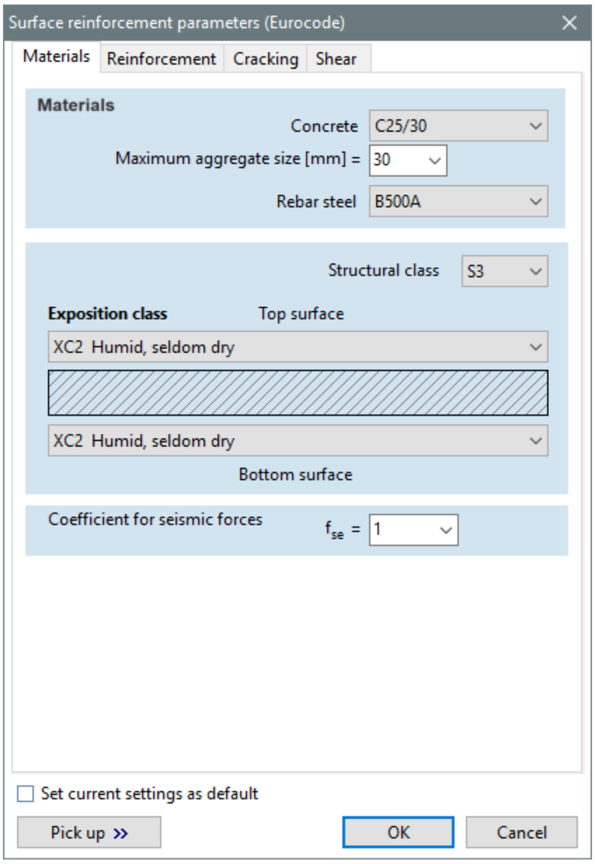
R. C. design The next step is to calculate the required reinforcement of the wall structure. Click on ***R. C. design*** tab:



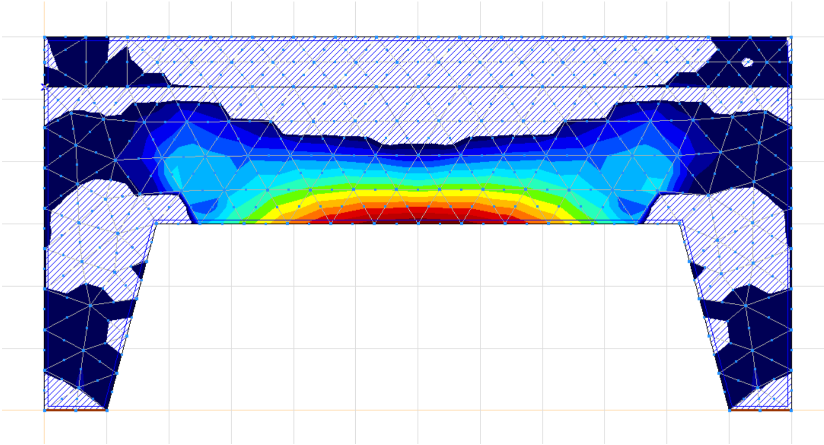
Reinforcement parameters



Click on ***Reinforcement Parameters*** icon, and select all surface elements with ***All (*)*** button. Complete selection with ***OK***, and set the followings on ***Materials*** tab:



Close dialog window with ***OK*** and the ***axb [mm^2/m]*** diagram is displayed in ***Isosurfaces 2D*** view:

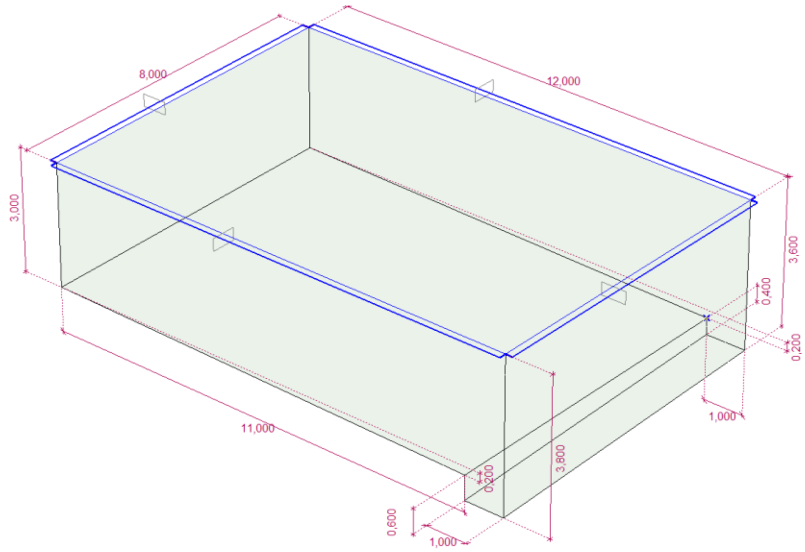


The required specific reinforcement in the ***x*** direction is the sum of ***axt*** and ***axb*** values.

5. SHELL MODEL

Objective

Determine the specific internal forces and the amount of reinforcement for the water reservoir shown below.



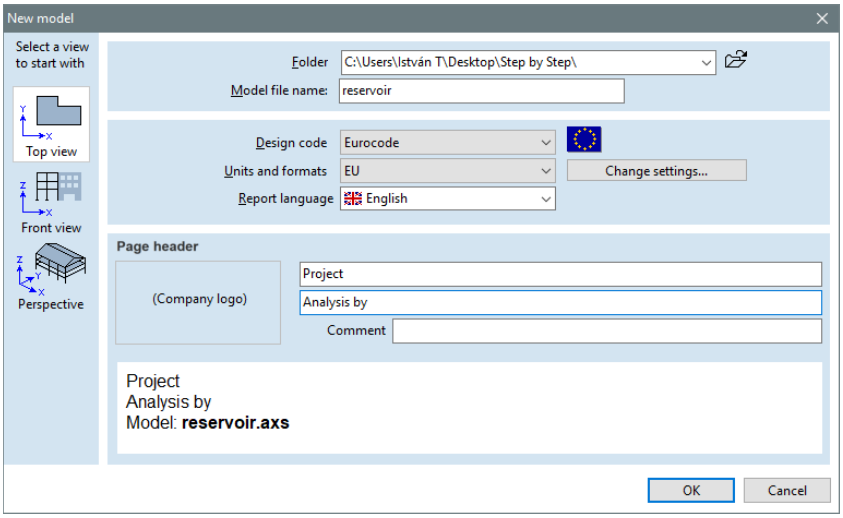
The thickness of the walls and the bottom slab is 250 mmm, the cross section of the ribs on the upper edges is 300x600 mm. The reservoir is made of concrete C25/30, the type of rebar is B500B. Use Eurocode 2 for design.

Start

Start ***AxisVMX4*** by double-clicking on ***AxisVMX4*** icon in its installation folder, found in ***Start*** – ***Programs*** menu.

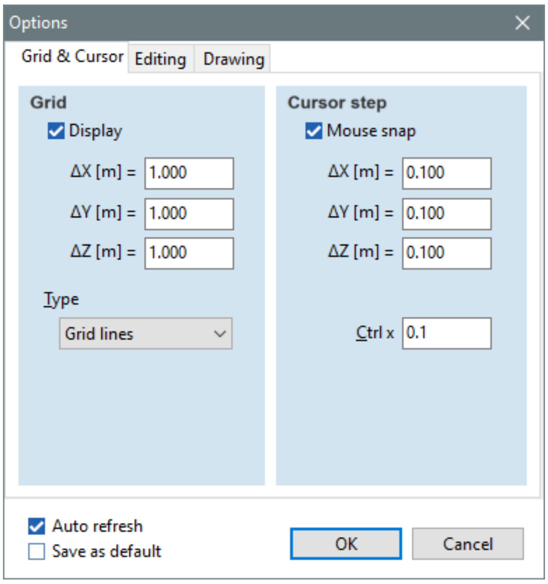
New

Create a new model by clicking on ***New*** icon. In the dialogue window replace ***Model file name*** with '***reservoir***', select ***Eurocode*** from ***Design code***s and set ***Unit and formats*** to ***EU***.

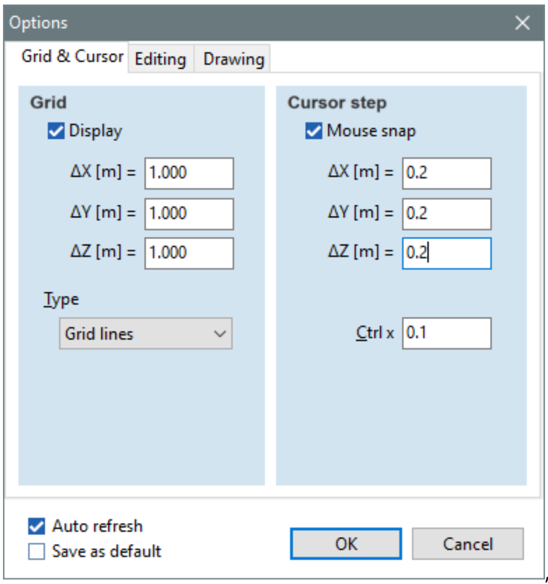


Options

Change settings for grid and cursor step. Click ***Settings***/ ***Options***/ ***Grid & cursor*** and set the followings in the window:



Replace each value under ***Cursor step*** to ***0.2***.

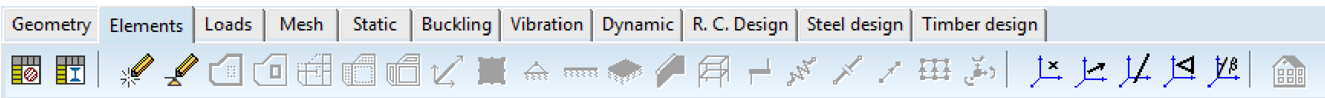


With these settings, the mouse cursor moves in ***0.2 m*** steps (geometric imperfection or editing error can be avoided while drawing the model).

Now, create the geometry using enhanced editing functions.

Define of geometry – Elements

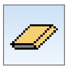
Select ***Elements*** tab to bring up ***Elements toolbar***.



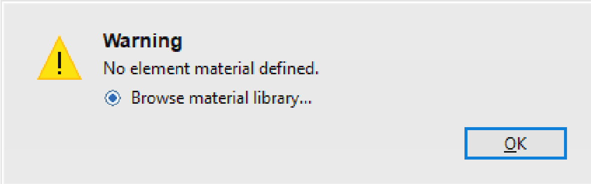
Draw objects directly

By clicking on ***Draw objects directly*** icon shows following window:

Domain

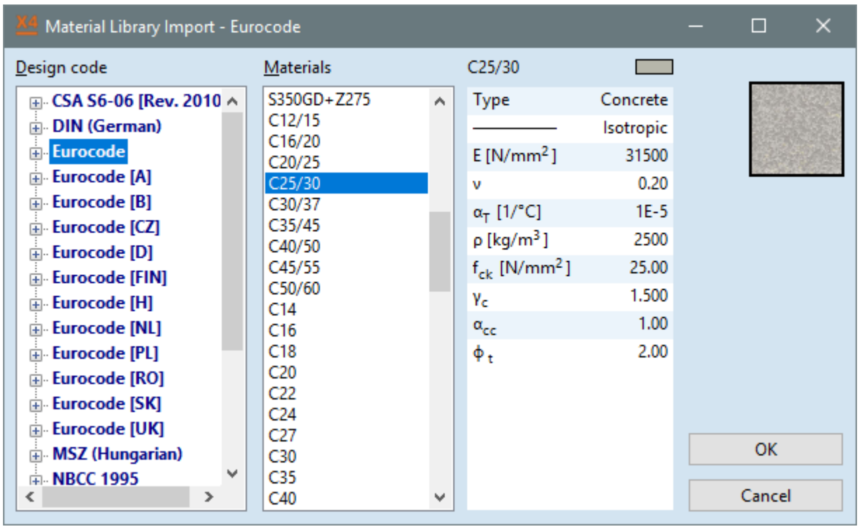


Click on ***Domain*** icon, the following window shows up:



Material library import

The following window shows after clicking on ***OK***:

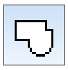


Select ***C25/30*** concrete in ***Materials*** list and confirm with ***OK***.

Thickness

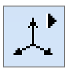
Set ***Thickness [mm]*** to ***250***.

Complex slab

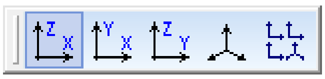


Click on ***Complex slab*** icon right to the first. The contour line can be drawn directly on the screen or can be defined by coordinates.
Firstly, let us draw the side wall of the reservoir in the **X-Z** plane.

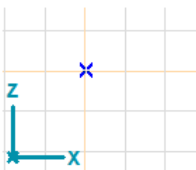
Views



Change view to ***Front view*** (***X-Z*** plane):

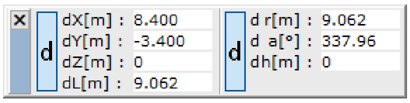


Choose the global origin as the first point of the polygon. It is at the bottom left, where the horizontal and vertical brown lines (representing the global ***X*** and ***Z*** axes) intersect. The blue cross (rotated 45°) shows the current origin of the actual coordinate system.

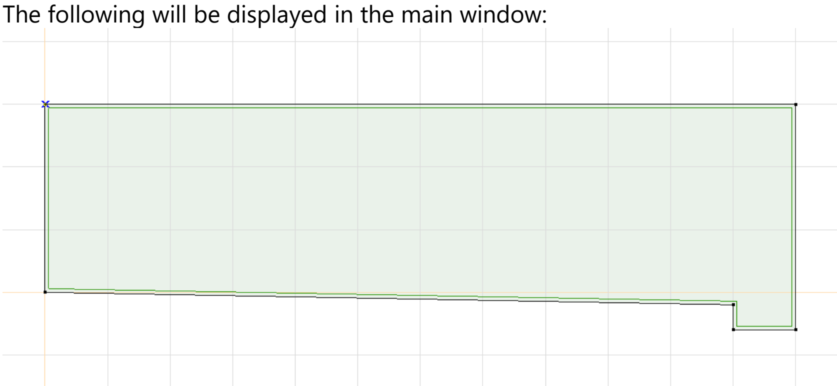


Nodes in relative coordinate system

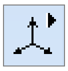
To enter additional nodes, select the relative coordinate system input. Press ***d*** button on the left of the ***Coordinates*** window. If ***d*** button is down (pressed) it denotes relative coordinates and the coordinate texts are marked by ***d*** (***dX***, ***dY***, ***dZ***, ...)



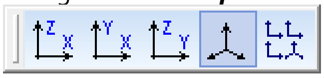
Move the mouse cursor to the following locations and click once to enter each vertex: ***11.0*** [m] right and ***0.2*** down, down ***0.4***, right ***1.0***, up ***3.6***, left ***12.0***, down ***3.0*** (or by keyboard: x 11 z –0.2 [***Enter***] z –0.4 [***Enter***] x 1 [***Enter***] z 3.6 [***Enter***] x –12 [***Enter***] z –3 [***Enter***]).
Press ***Esc*** twice to quit the drawing function, when the polygon has been created.
The following will be displayed in the main window:



Views

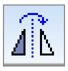


Change view to ***Perspective***:

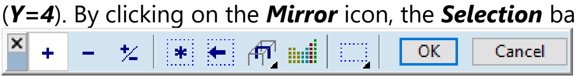


Specify the following values in the ***Perspective settings*** window:
H 30.0; ***V 320.0***; ***P 0.0***.
Click on **X** button at top right corner to close window.

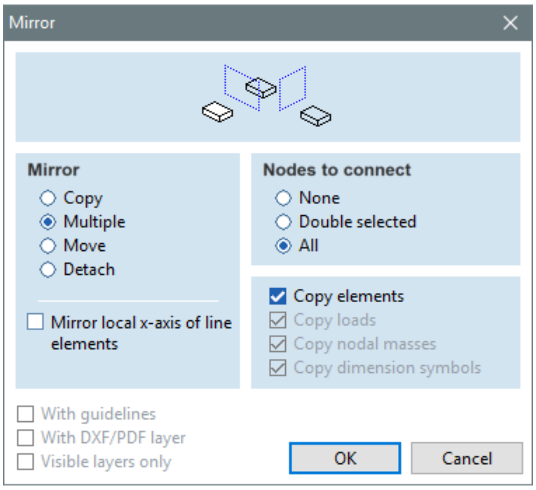
Mirror



Create the parallel wall on other side by mirroring the first one with respect to the centre of structure (***Y=4***). By clicking on the ***Mirror*** icon, the ***Selection*** bar is displayed:



Select the domain with ***All*** (*******) button, the color of the selected elements changes, become pink. Finally, finish selection with ***OK***.

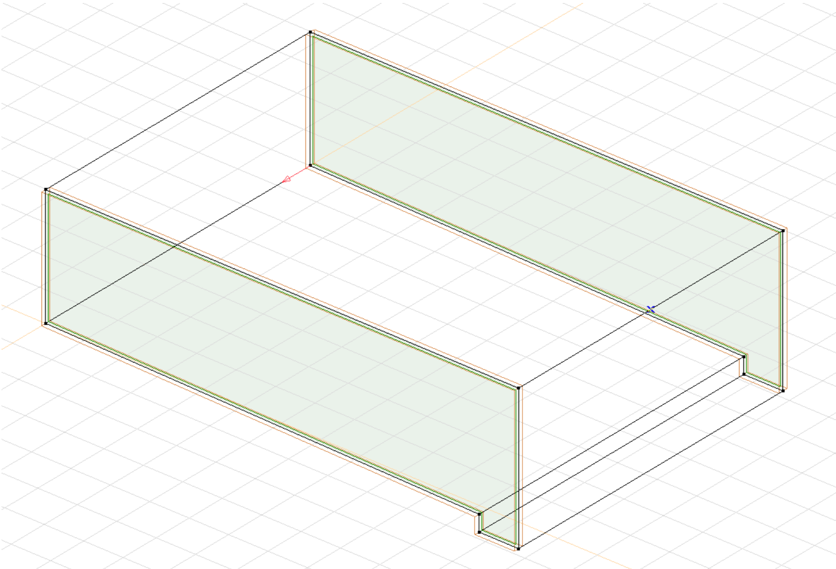


Select ***Multiple*** mirror type and ***All*** for ***Nodes to connect***, then confirm with ***OK***.
Define the mirror plane with two points, type in the following values using relative coordinates.

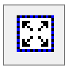
x	12	y	4	z	0	<**Enter**>
x	1	y	0	z	0	<**Enter**>,

Press ***Esc*** once to close the command.

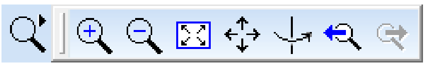
The following will be displayed:



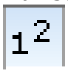
Zoom to fit



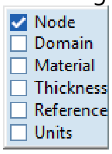
Click on ***Zoom to fit*** icon for better view.



Numbering



Move the cursor to the bottom right corner, find ***Numbering*** icon and click on it (it is the second one on the right), the next checkbox will be displayed:



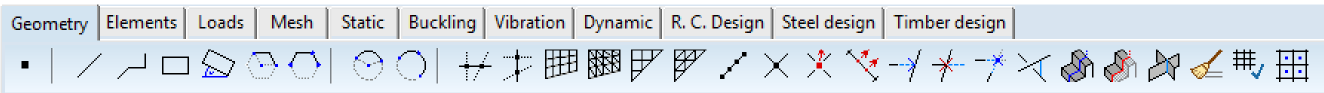
Here numbering function can be turn on or off. Turn on the checkbox at ***Node***, then the node numbers appear immediately next to the nodes.

Translate / Copy

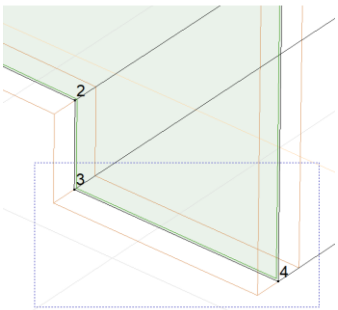
To specify a slope for the watercourse, move the line between ***node 3*** and ***4*** down by ***0.2 m***.

Geometry

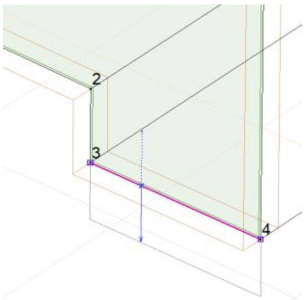
Change tab to ***Geometry***:



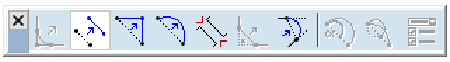
Select the line between ***nodes 3*** and ***4*** by a selection rectangle:



All the elements within the rectangle will be selected (***node 3***, ***4*** and the line between them). To move the line, move the cursor over the selected line, press and hold the left mouse button and drag the cursor down.



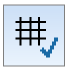
Select the ***Parallel move of line*** icon from the icon bar on the left and choose the first among the active icons:



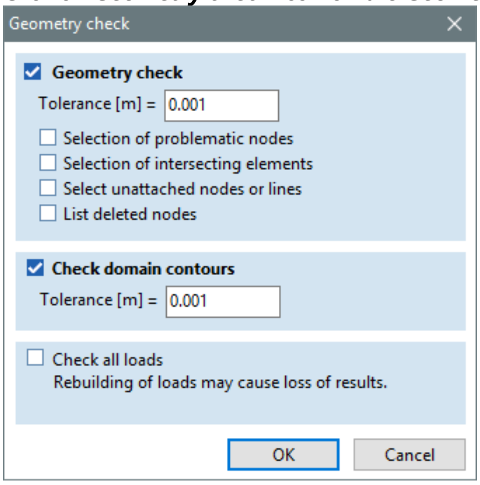
Firstly, the translation vector should be specified, type in the following to define the exact distance:

x 0 y 0 z -0.2 **<Enter>**

Geometry check

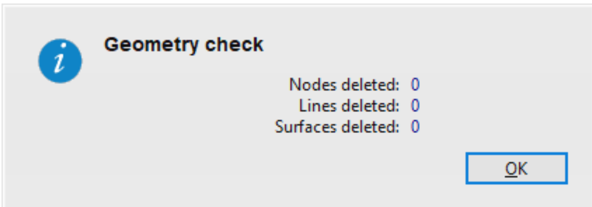


Click on ***Geometry check*** icon on the ***Geometry*** tab, the following window shows up:



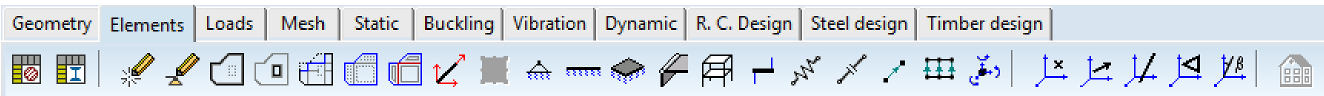
In the window, the maximum ***Tolerance*** (distance) for checking points can be set. Check the ***Select unattached nodes or lines*** checkbox and start the analysis with pressing ***OK*** button.

After the analysis, the following message report will be displayed:

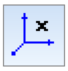


Elements

By clicking ***Elements*** tab types of the elements, their material properties, cross-sections and references can be defined:



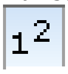
Reference point



The local system of finite elements can be set by references. In this example a reference point is used to define the orientation of the local ***Z*** direction of the domains and a reference plane to define the in-plane ***X*** and ***Y*** axes.

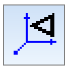
Click on ***Reference point*** icon then click the centre point of the line between ***node 5*** and ***11***. To locate the centre point move the cursor along the line and check if the cursor shape changes from ***/*** (line) to **½**. Press ***Esc*** to exit from function.

Numbering



Move the cursor over the ***Numbering*** icon on the ***Speed buttons*** toolbar. Turn on the ***Reference*** checkbox. Now, an ***R2*** label appears beside the reference symbol.

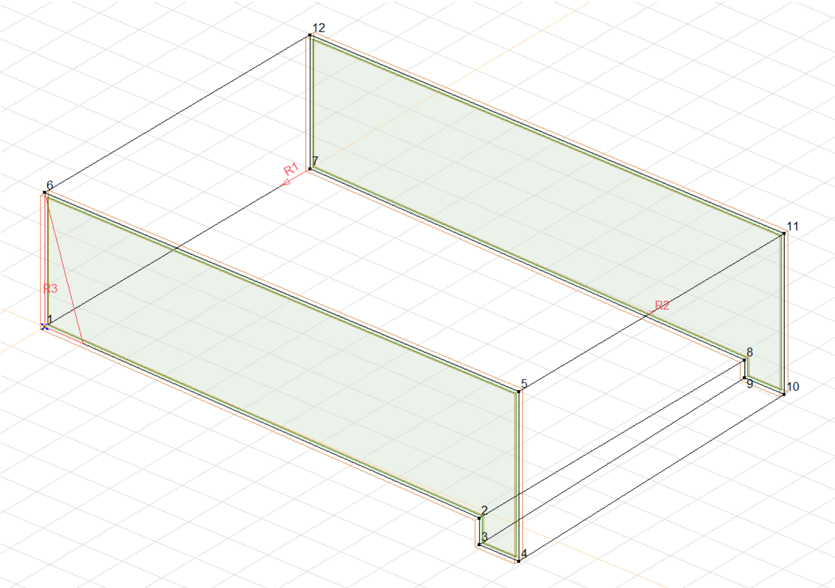
Reference plane



To set the local system of domains create a reference plane. Click on ***Reference plane*** icon on the ***Elements*** tab. You need three points to define a plane.

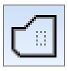
Click on ***node 6***, then click anywhere on the line between ***node 1*** and ***2***, finally click on node ***2***.

The following will be displayed:



Press ***Esc*** to exit from function.

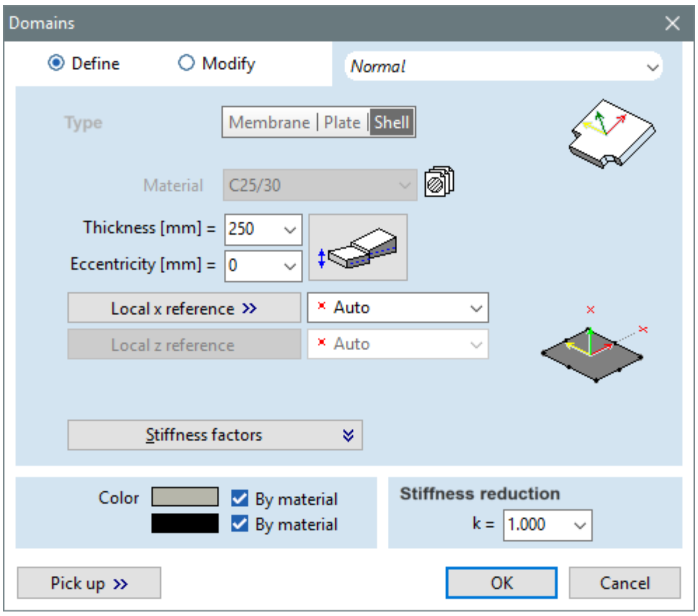
Domain



Define a domain to create structural surface elements. Click ***Domain*** icon, then ***Selection palette*** appears. Click on the following lines between the given nodes to select domain contours:

12 - 6	6 - 1	1 - 7	7 - 12
1 - 7	1 - 2	2 - 8	7 - 8
5 - 11	5 - 4	4 - 10	10 - 11

Click ***OK*** on ***Selection palette***, then ***Domains*** dialog window shows up:

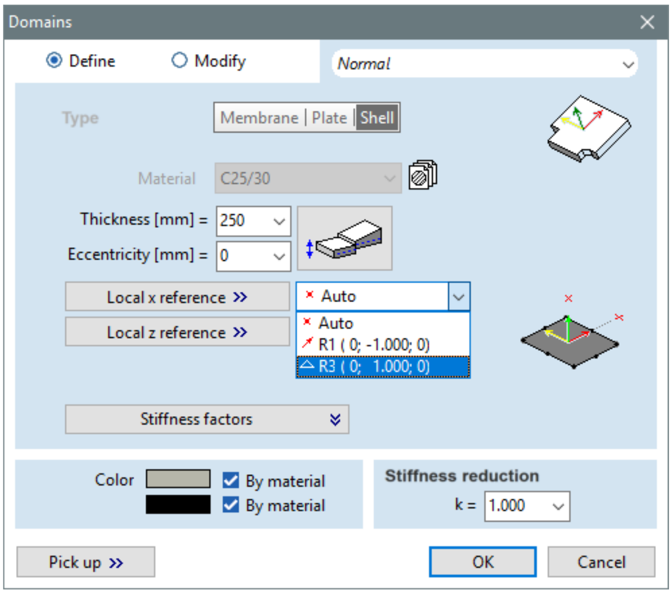


Thickness

Enter ***250*** mm into the edit field of ***Thickness[mm]***.

Reference

Set the ***Local x reference*** to ***R3***:



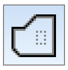
then close window with ***OK***.

A green contour can be seen along the domain boundary showing the shape of the domain. The colour depends on the element type, shell domains always have a green contour.

Local systems

Turn on the display of the local systems of ***Domains*** by clicking on the ***Local systems*** speed button in the bottom right ***Speed buttons toolbar*** (fourth icon on the right).

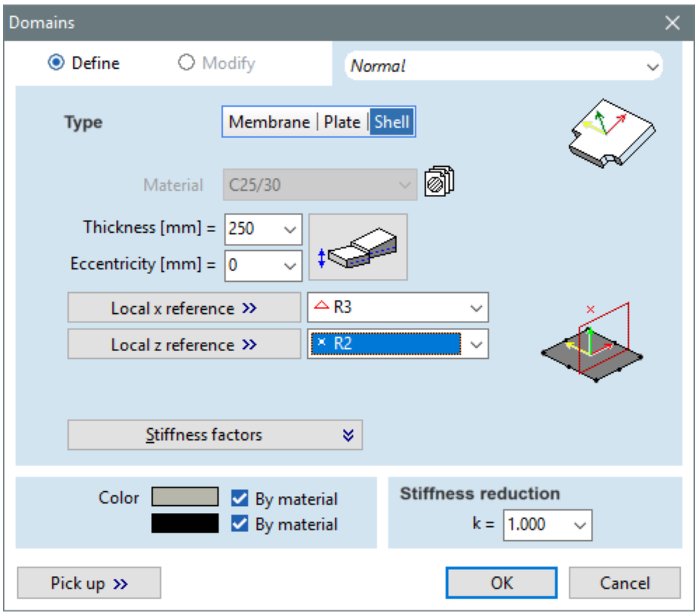
Domain



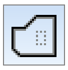
Define the other wall elements. Click on ***Domain*** icon, then ***Selection palette*** appears and select lines between the following nodes:

2 - 8   8 - 9   3 - 9   2 – 3,

then finish selection with ***OK***. Choose ***Shell*** element type and ***R3*** for ***Local x reference*** and ***R2*** for ***Local z reference***, finally press ***OK***.



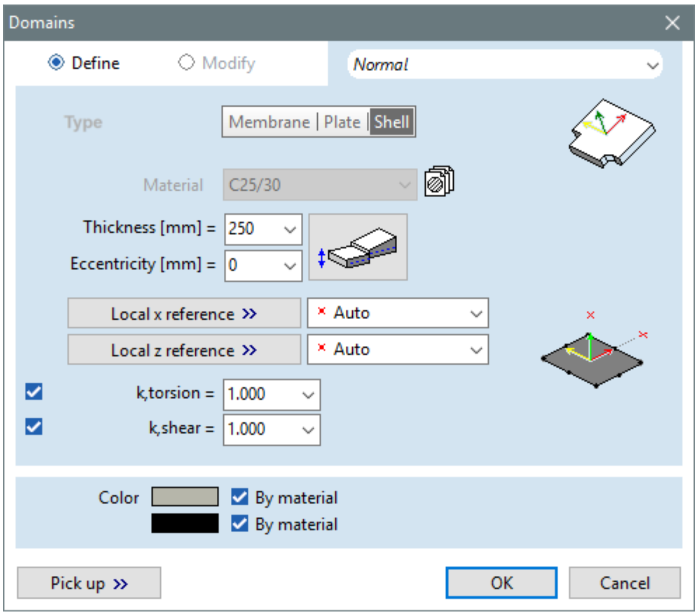
Domain



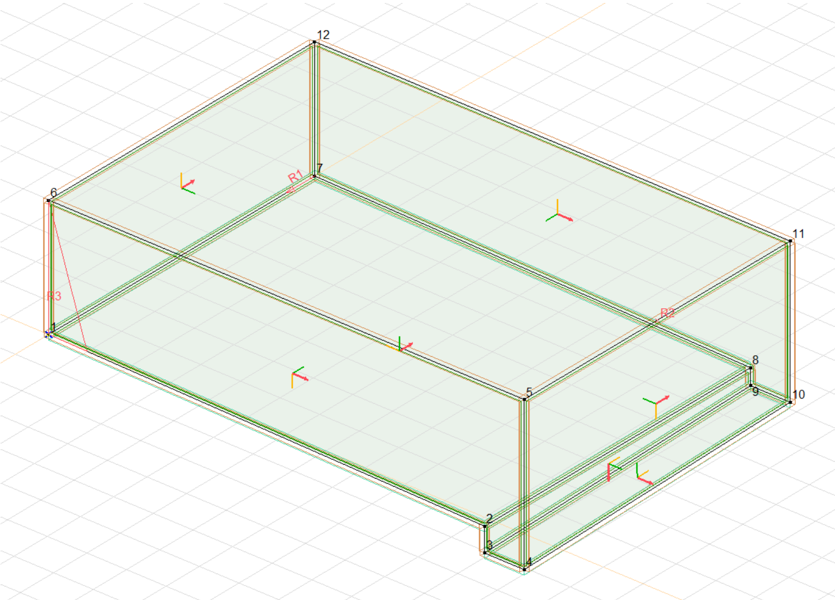
Repeat creating shell elements, activate again ***Domain*** function, then select lines between the given nodes:

3 - 9   3 - 4   4 - 10   9 – 10,

then press ***OK***, then set ***250*** mm for ***Thickness***, and leave references on ***Auto***.



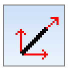
The following will be displayed:



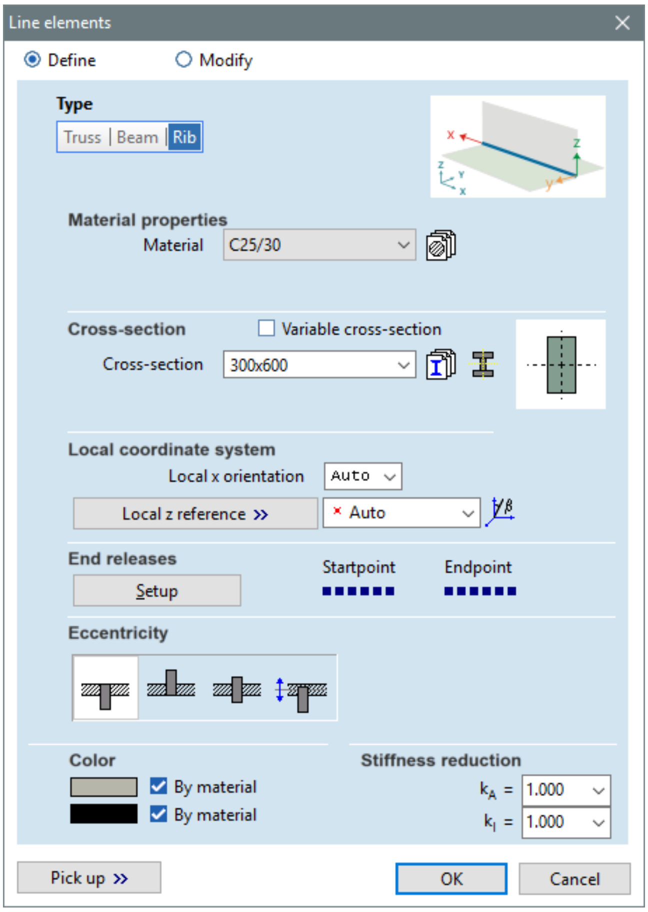
Speed buttons

Turn off ***Numbering*** of ***Nodes*** and ***Local systems*** using ***Speed buttons***.

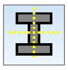
Line elements



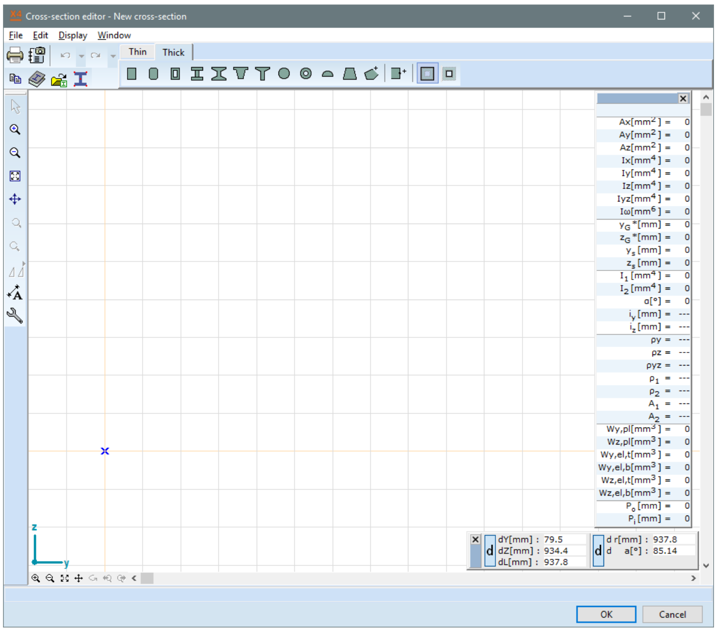
To define ribs on the upper edges, click on ***Line elements*** on the ***Elements*** tab. The ***Selection palette*** appears, click on the 4 edges and close the selection with ***OK***, then ***Line elements*** dialog window appears:



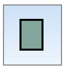
Cross-section editor



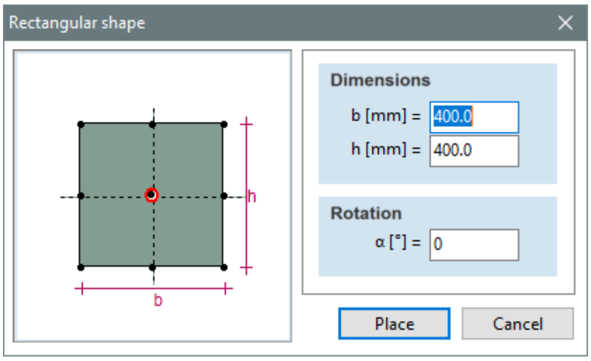
To define the cross-section of the rib, click on ***Cross-section editor*** icon. The following window shows up:



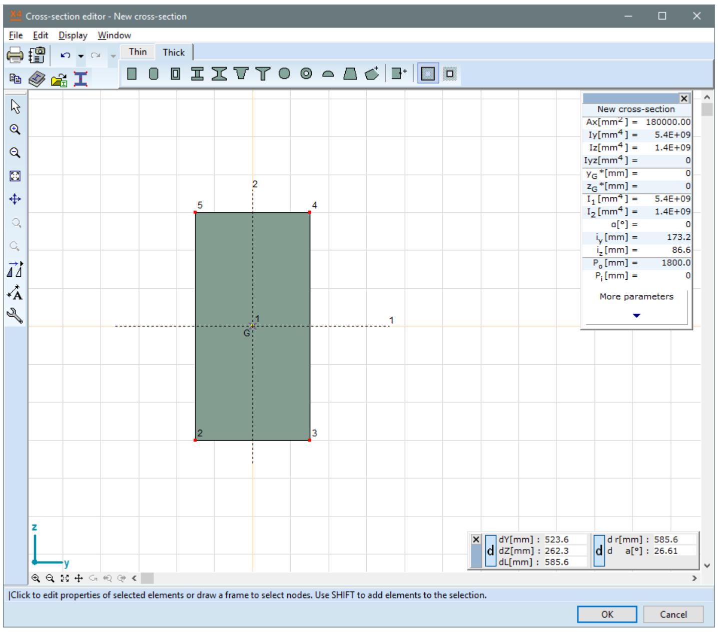
Rectangular shape



To define a ***300x600 mm*** rectangular shape click on **Rectangular shape** icon, and type in the right values for ***b*** and ***h:***

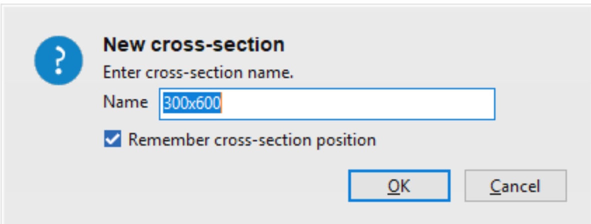


Finishing the data input, press ***Place*** button to place the cross-section in the editor window. Click anywhere in the window. We get the next result:

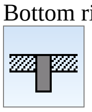


The 1st and 2nd principal direction of the cross section indicated by the crossing dashed lines on the cross-section, marked with numbers 1 and 2. The centre of gravity is marked with G. The other cross-section parameters are displayed within the property window. Click ***More*** parameters if you want to see all parameters calculated automatically by a finite element analysis of the shape.

Click ***OK*** to close ***Cross-section editor***, in the dialog window change default name of cross-section if required. Click on ***OK*** again to confirm.

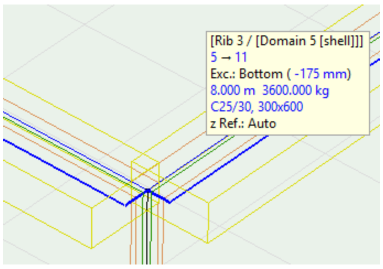


Bottom rib



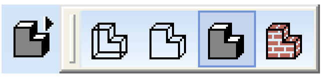
Set eccentricity to ***Bottom rib*** then finish settings with ***OK***. Rib centre lines are displayed in blue and the contour of the rib is shown in yellow.

Move the cursor over a rib and wait for a tooltip displaying element properties:

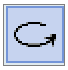


Rendered view

Move cursor over the ***View mode*** icon on the left icon bar. A fly out toolbar appears, then select the third icon to choose ***Rendered view***. In this view mode check the elements already defined.



Rotate

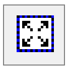


Click ***Rotate view*** icon on the ***Zoom*** toolbar at the bottom left corner of the main window. Drag the model to rotate it. A special ***Rotation*** toolbar appears to control the rotation process. You can also hold ***Alt*** + mouse wheel and rotate the view anytime.

Restores the previous view

Restore previous view with icon between ***Zoom*** toolbar.

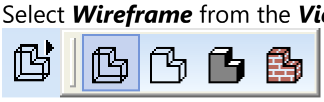
Zoom to fit



Click on ***Zoom to fit*** for better view.

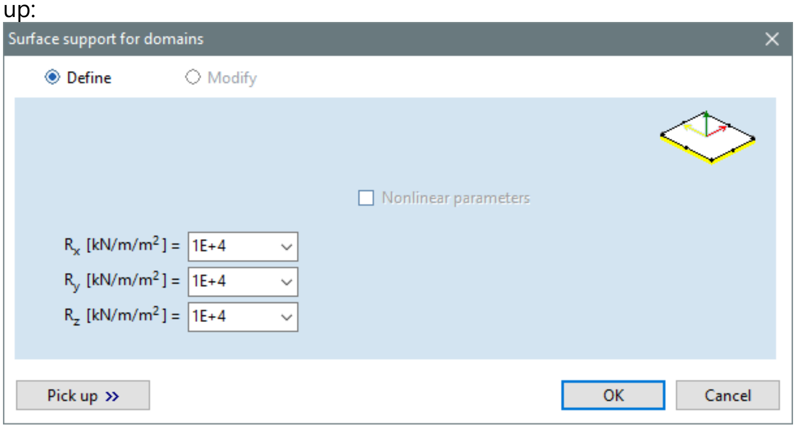
Wireframe

Select ***Wireframe*** from the ***View mode*** fly out toolbar.

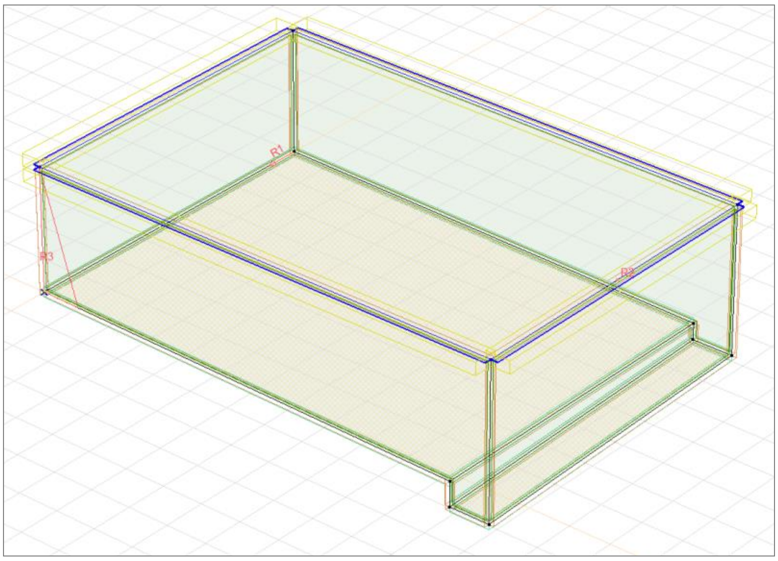


Surface support

To define supports for the structure click on ***Surface support*** icon on ***Elements*** toolbar, then ***Selection palette*** appears. Click on the bottom two domains. Finish with ***OK,*** then the following window shows up:

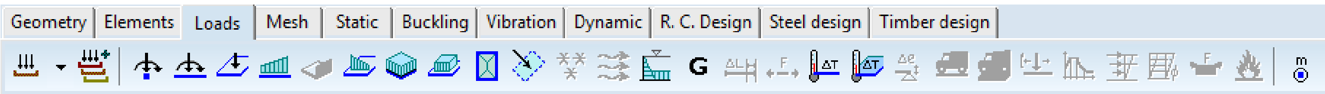


Chang***e*** $\boldsymbol{R_x}$ and $\boldsymbol{R_y}$ to ***1E+3***, then click ***OK***. The following will be displayed in the main window:

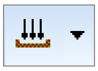


Loads

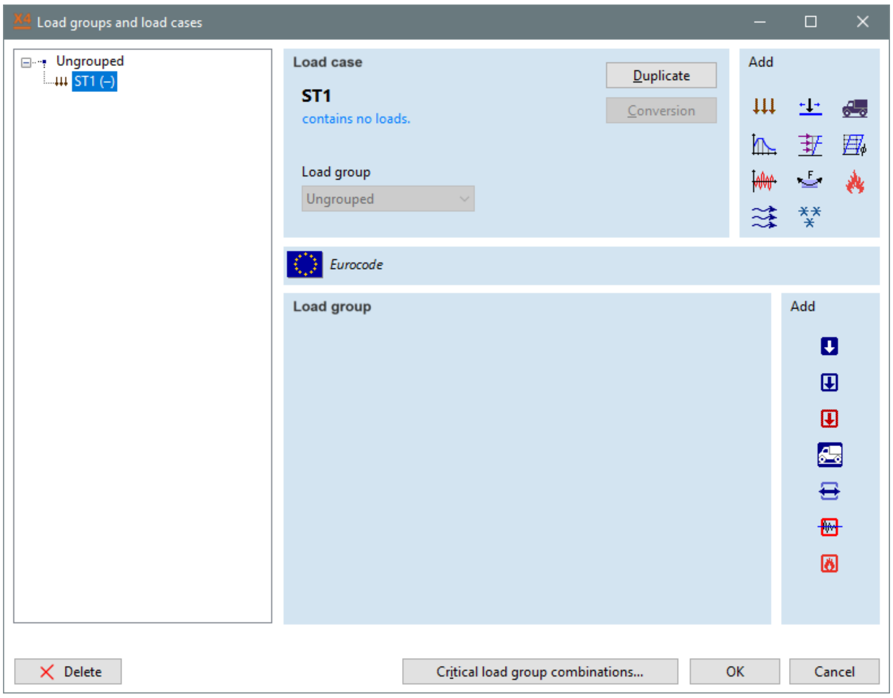
To define loads click ***Loads*** tab:



Load cases and load groups



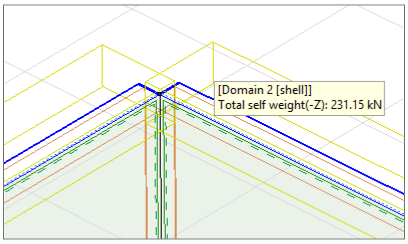
To define load cases click ***Load cases and load groups*** icon on the ***Loads*** toolbar. The following window will be displayed:



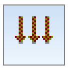
Click on the selected load case ***ST1*** and rename it to ***SELF WEIGHT***, then click ***OK*** to close dialog.

Self weight

To define self weight, click on ***Self weight*** icon. On the ***Selection palette*** click on choose ***All*** (*) icon or press grey * on the keyboard. Click ***OK*** to close ***Selection palette***. Dashed lines along the domain contours represent the self weight. Moving the cursor to a domain edge, a tooltip appears showing the weight of the specified domain:

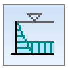


Static load case

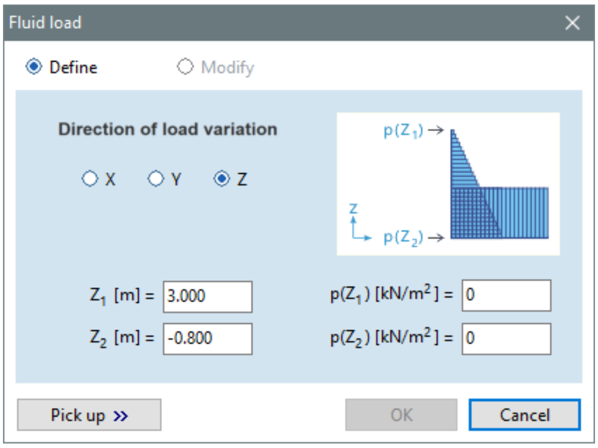


To create another load case, click again the ***Load cases and load groups*** icon and click on ***Static load case*** button in the ***Add*** new load case group box. Enter '***WATER***' as the name of the new load case in the tree on the left. Click ***OK*** to close the dialog.

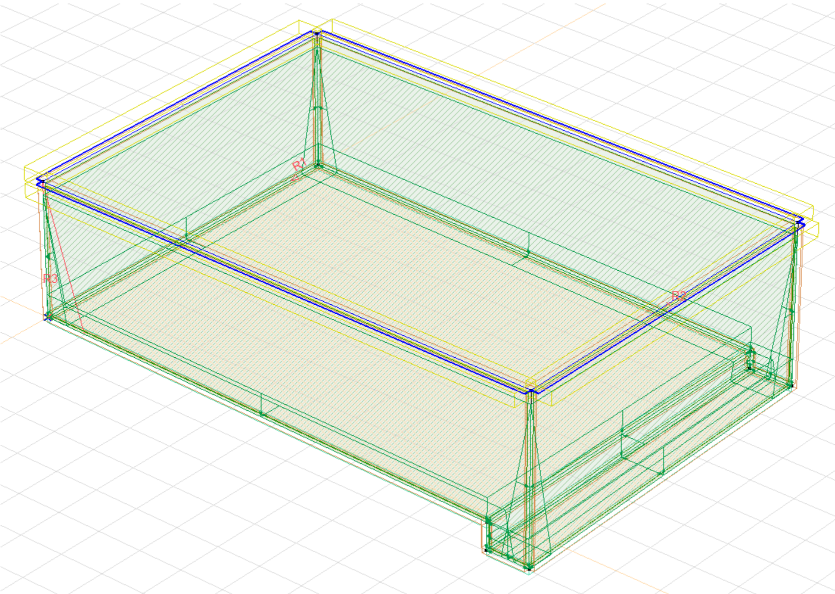
Fluid loads



To define the water load, click on ***Fluid loads*** icon. On the ***Selection palette*** click choose ***All*** (*) icon or press grey ***** on the keyboard. Click ***OK*** to close the ***Selection palette***. The following window will be displayed:

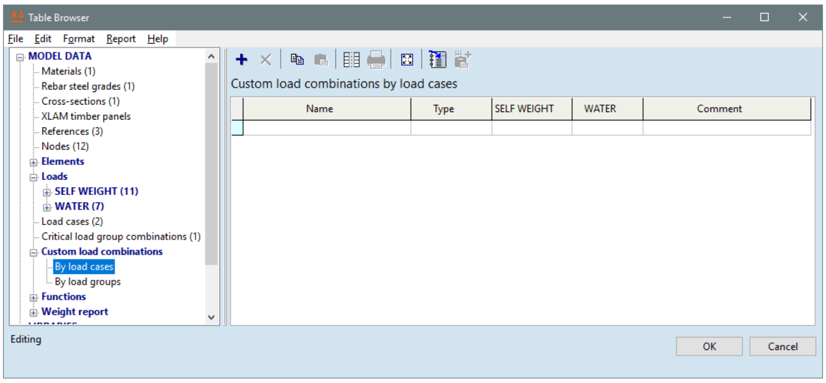


To define water level ***30 mm*** under the top edge of the reservoir change ***Z1 [m]=3.000*** to ***2.7***, and set the bottom pressure value ***p(Z₂) [kN/m²]*** to ***–35*** (pressure is in the negative local ***z*** direction) and click ***OK.*** The following will be displayed:



Load combinations

To create load combinations, click on ***Load combinations*** icon. You get to the load combinations table in the Table Browser:



New row

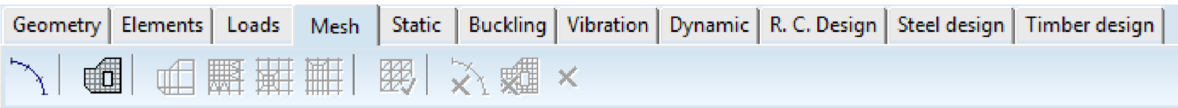
To create new load combinations, click on ***New row*** icon. Leave the name on default ***Co.#1*** then select: ***– (user defined combination)***. Then enter ***1.35*** into the column of case ***SELF WEIGHT*** and ***1.00*** into the column of ***WATER*** load case. Use ***Tab*** or ***Enter*** to jump to the next cell. Click ***OK*** to close the dialog.

Speed buttons

Use the speed buttons: turn off ***Supports, Reference*** and ***Object contours in 3D*** on ***Graphic symbols*** tab.

Mesh

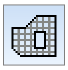
To create finite element mesh change tab to ***Mesh***.



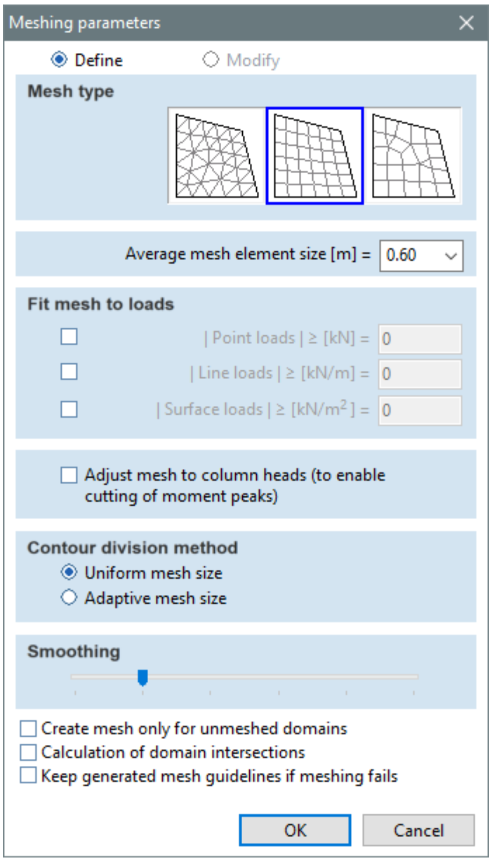
Speed buttons

Using speed button turn off ***Load display*** (the fifth icon on the right).

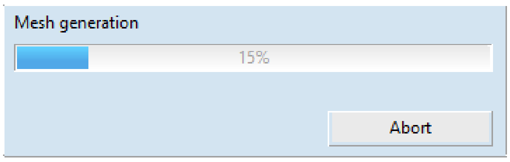
Domain meshing



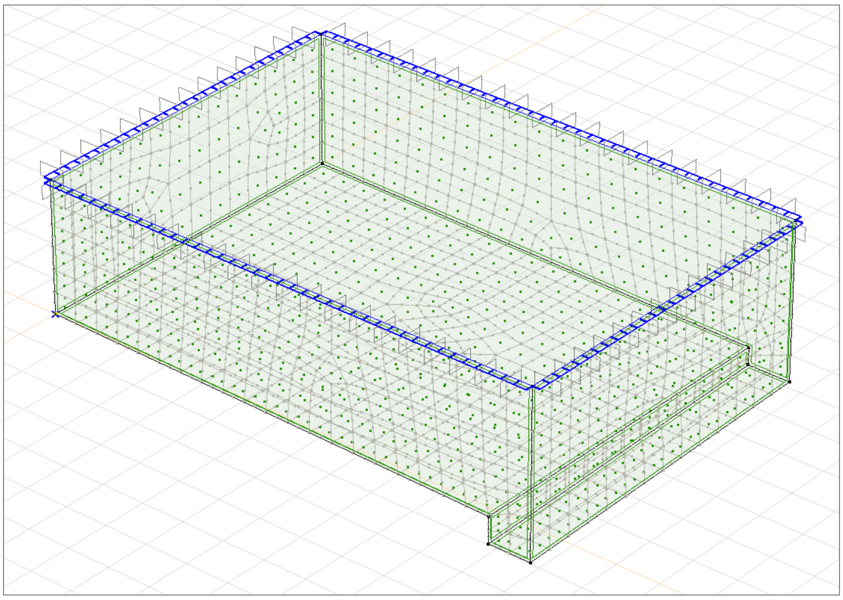
Click on ***Domain meshing*** icon. On ***Selection palette*** click ***Select all*** (*) icon or press grey ***** on the keyboard. Click on ***OK*** to close the ***Selection palette.*** Select rectangular mesh type (in centre) and set ***Average Mesh Element Size*** to ***0.60 m***.



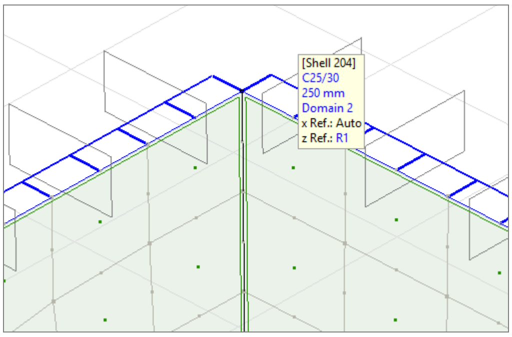
Click on ***OK*** to close dialog. Meshing process is can be followed on the status bar:



After completing, the following will be displayed:



Green points at the centre of surface elements represent ***Centre point***s of the shells. Moving the mouse over a centre point, a hint appears after a while showing properties of the finite element.



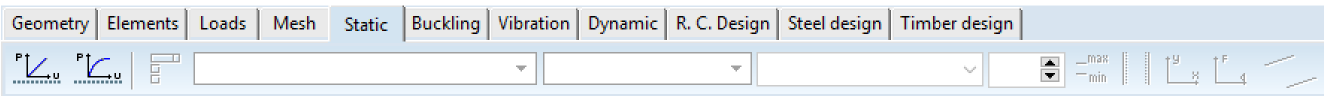
Speed buttons

Turn off ***Nodes***, ***Surface centre*** and ***Domains*** on ***Graphic Symbols*** tab, then turn off also the ***Mesh Display***.

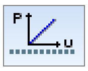
With this last step, we have finished finite element modelling.

Static

To run the static analysis, change tab to ***Static***.

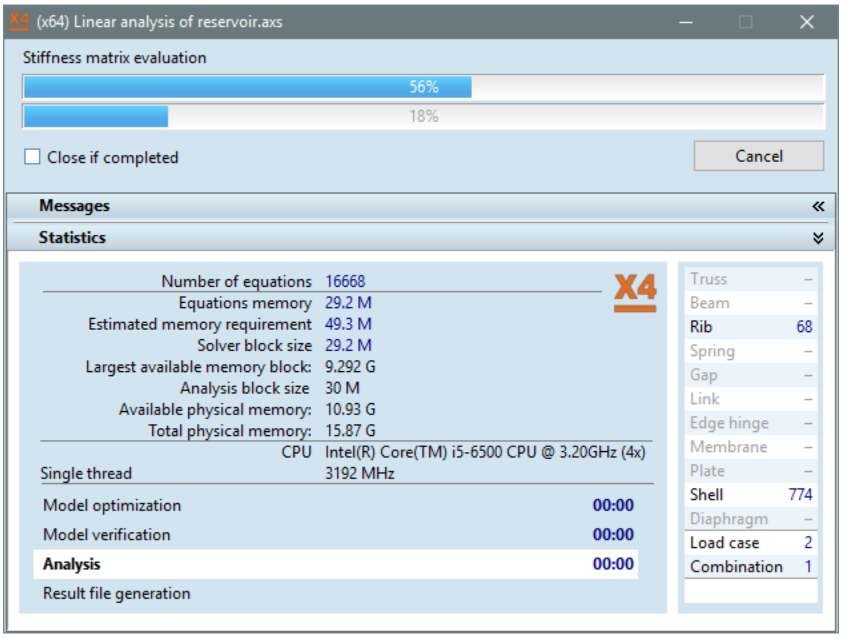


Linear static analysis



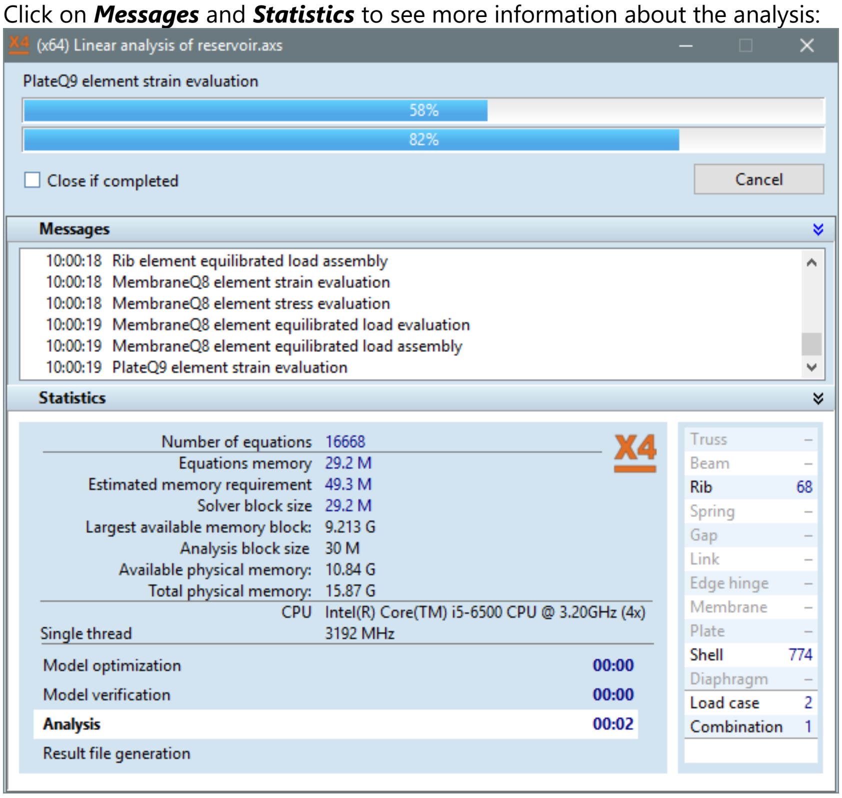
Click on ***Linear static analysis*** to start analysis.

The following dialog shows up giving feedback about the process of the analysis:



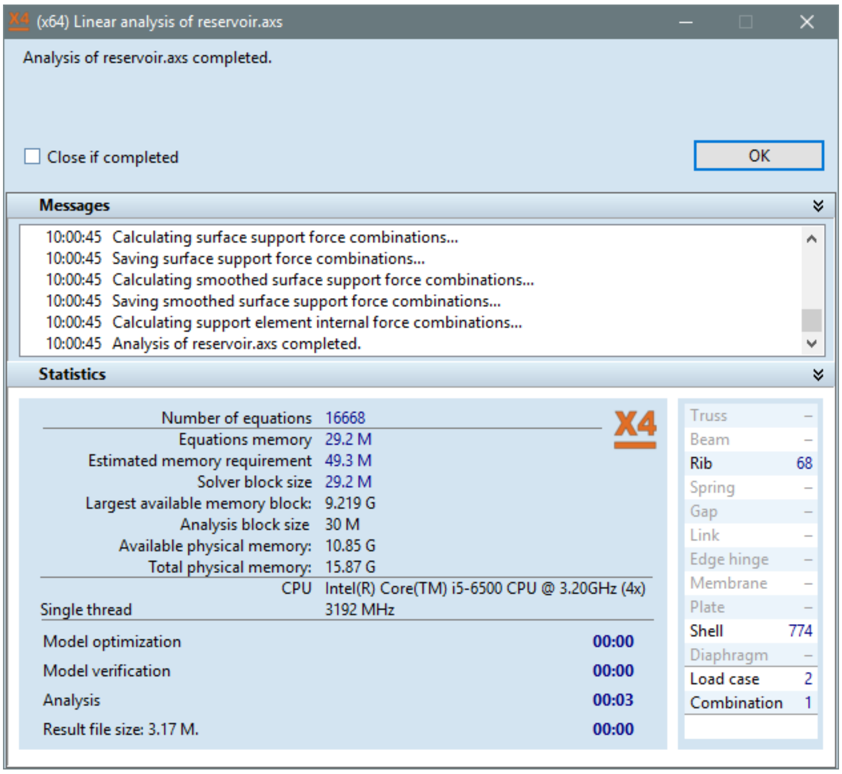
Messages, Statistics

Click on ***Messages*** and ***Statistics*** to see more information about the analysis:



The ***Estimated Memory Requirement*** shows the necessary amount of memory to run the analysis. If this value is higher than the available physical memory, then ***AxisVM*** uses the hard disk to swap memory blocks during the calculation. If the system of equations fits into the physical memory the calculation is considerably faster.

The following window shows up after analysis:

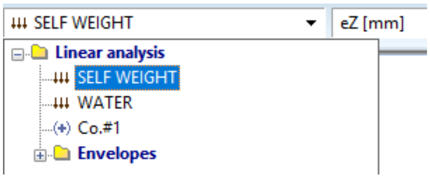


After click on ***OK***, then ***Static*** tab is activated showing displacement of the structure in ***ez*** direction in case ***SELF WEIGHT*** load and displayed in ***Isosurfaces 2D*** mode.

Numbering

Click ***Numbering*** speed button and turn on ***Write values to Surfaces*** and ***Min./Max. only***.

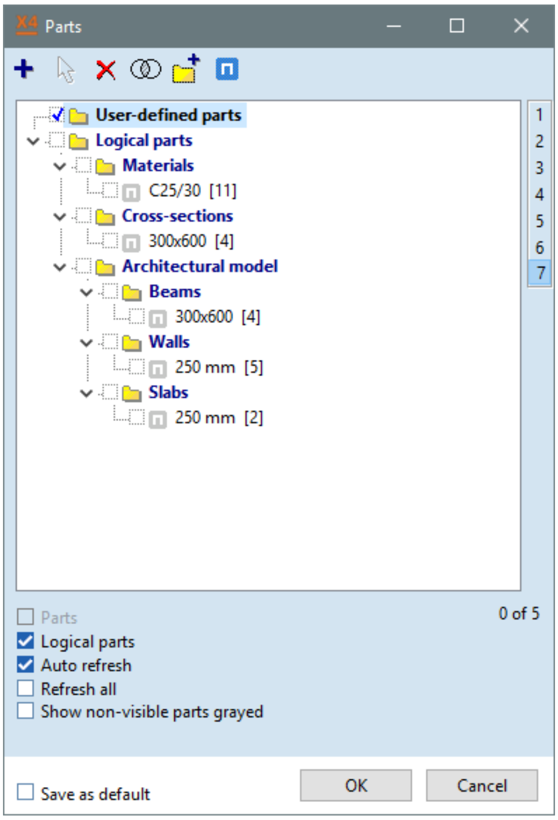
To see the result for the ***WATER*** load case, click on the dropdown button of the combo box displaying ***SELF WEIGHT*** and select ***WATER***:



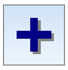
Select ***ey [mm]*** result component to display.

Parts

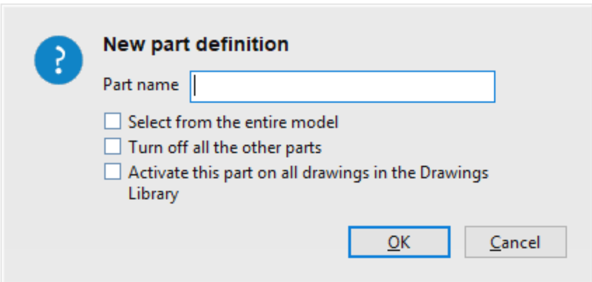
To hide the front wall (for better visibility) of the reservoir create a part. Click ***Parts*** icon on the icon bar on the left, then ***Parts*** dialog shows up:



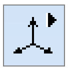
New



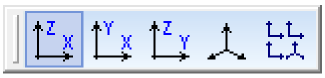
Define a part containing all the elements but the front wall, click on ***New*** icon and specify the name as ***1***:



View

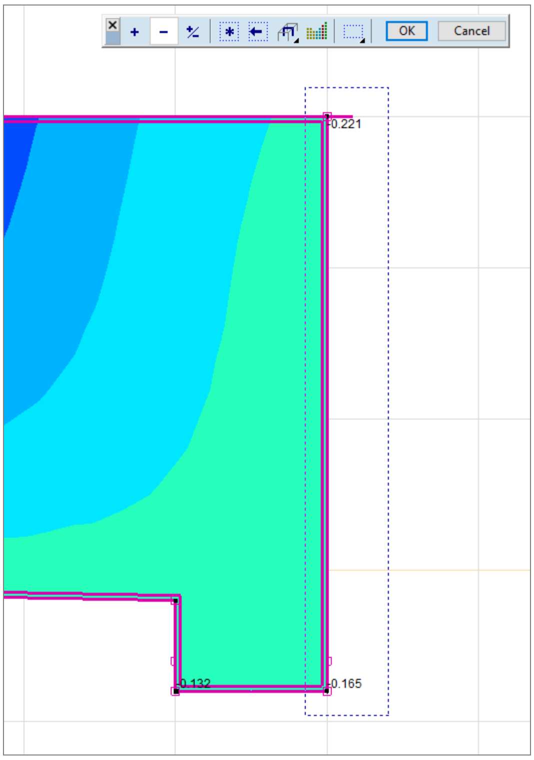


Change view to ***X-Z*** plane!



Press and hold wheel of the mouse, to pan the view, so the right part of model is fully visible. Use the bottom slider if your mouse does not have a wheel button.

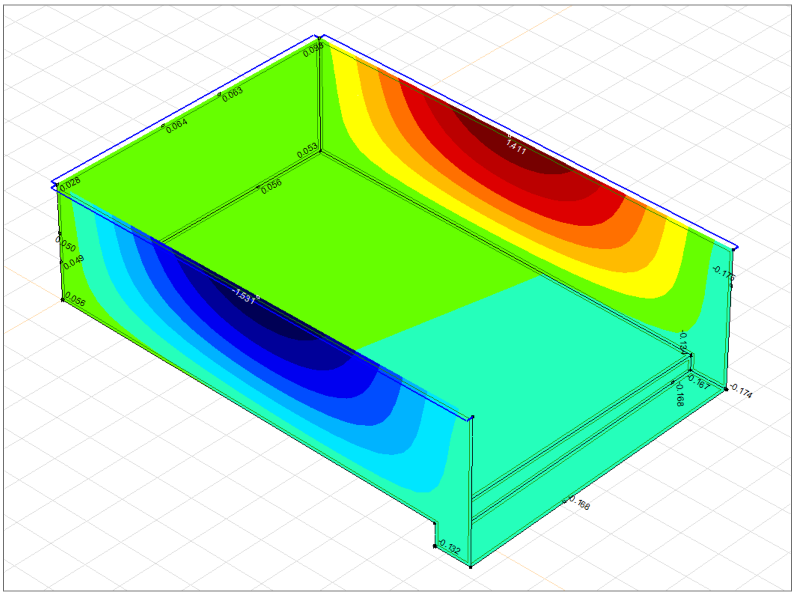
Use selection rectangle to unselect the wall on right hand side from the selection:



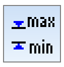
Click on ***OK*** on the ***Selection palette***, then close ***Parts*** dialog window with ***OK***.

Restore the previous view

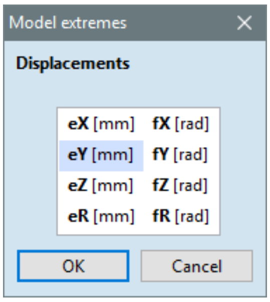
Restore the previous view with clicking the icon among ***Zoom*** toolbar – or activate ***Perspective*** view with ***Ctrl+4*** keys. Defining the part, the front wall is not visible in the next view:



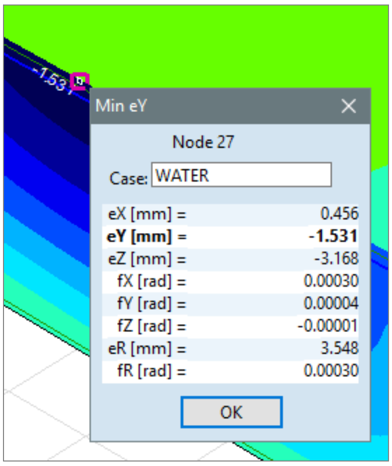
Min, max values



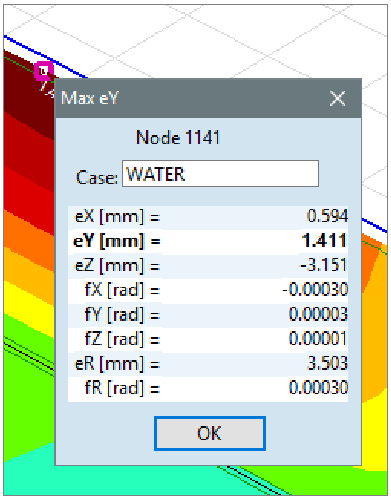
To find extreme values of horizontal displacements, click ***Min, max values*** icon. The following window shows up:



Select one of the deformation components: ***eY [mm]***. Click on ***OK*** to show maximum negative value and its location.



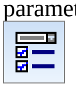
Confirm with ***OK***, then result dialog jumps to the maximum positive value.



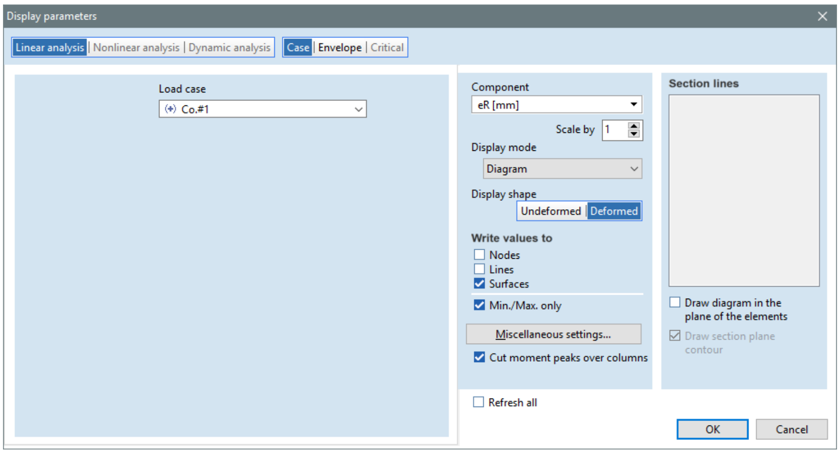
Close the window with ***OK***.

Select load combination ***Co.#1*** and ***eR*** resultant displacement.

Result display parameters



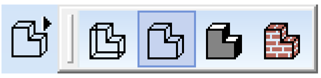
Click on ***Result display parameters*** icon, set ***Display shape*** to ***Deformed***, ***Display mode*** to ***Diagram*** and ***Scale by*** to ***2***:



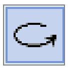
Close window with ***OK***.

Hidden line removals

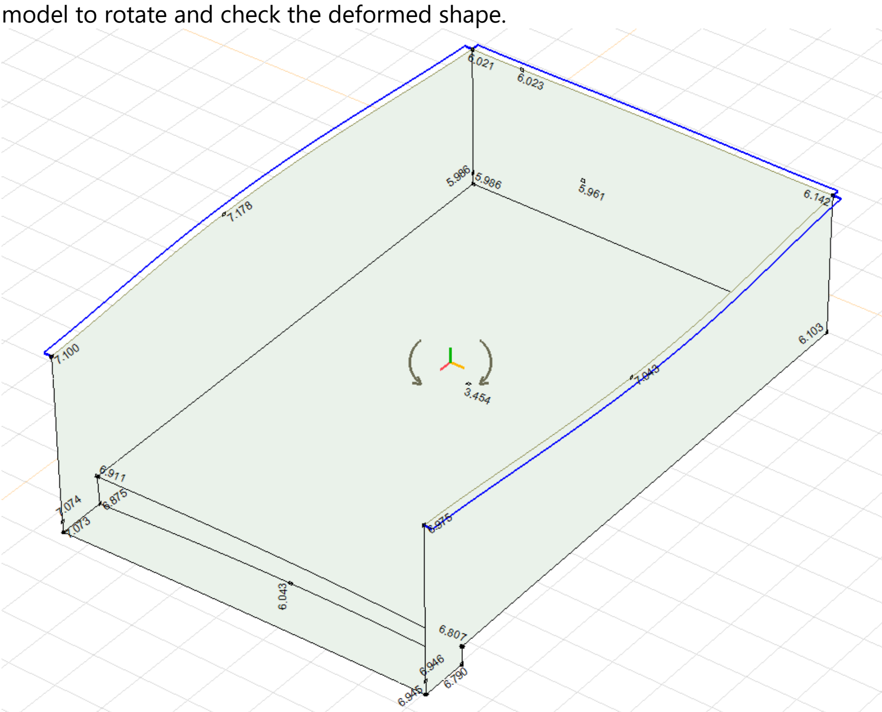
Go to the bottom right of the screen, turn on the ***Mesh display*** speed button and at the icon bar on the left change the ***Display mode*** to ***Hidden line removal***:



Rotate



Click on ***Rotate view*** icon in the ***Zoom*** toolbar at the bottom left corner of the main window. Drag the model to rotate and check the deformed shape.

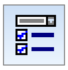


Press ***Esc*** to exit.

Restore the previous view

Now, restore the view with pressing its icon among ***Zoom*** toolbar.

Result display parameters

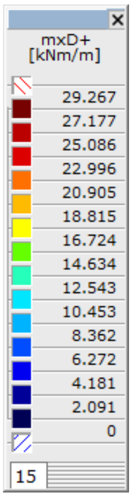


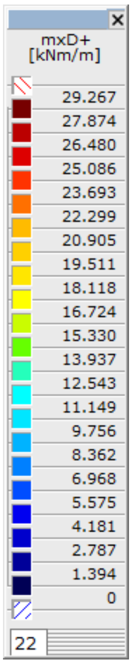
Click on ***Result display parameters*** icon: change ***Display shape*** to ***Undeformed***, ***Display mode*** to ***Isosurfaces 2D*** and restore ***Scale by*** to ***1***.

Use speed button to turn off ***Mesh display*** and change result component to ***Surface Internal Forces – mxD+***.

Color legend

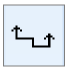
The ***Colour legend*** shows boundary values of each colour. ***Adjust number of boundary values*** by dragging the bottom edge of this palette, set it to ***22***:



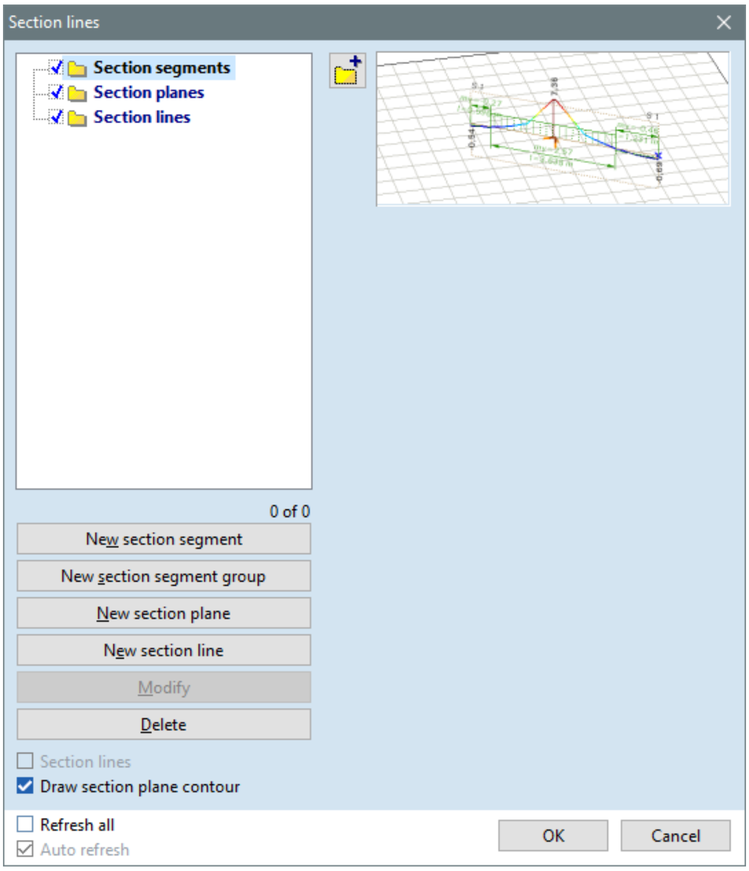


Change result component to ***Surface Internal Forces*** – ***myD-***.

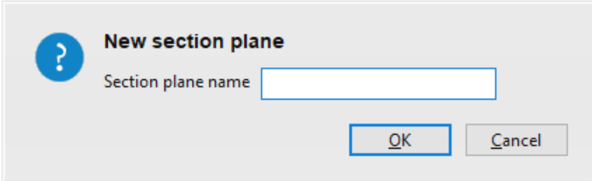
Section lines



To specify sections for displaying ***myD-*** diagram, click on ***Section Lines*** icon on the icon bar on the left side. The following window will be displayed:

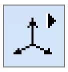


To define a new section plane, click on ***New section plane*** button in the dialog. The following message box will be displayed to enter name of the section:

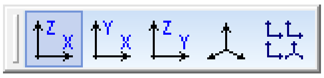


Type in ***1*** as the name of the section plane and confirm with ***OK***, then close the ***Section Lines*** window as well with ***OK***.

Views

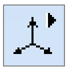


Change view to ***X-Z*** plane – or press ***Ctrl+1*** keys!

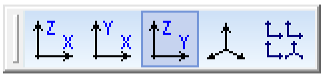


Specify a section plane in the middle of the reservoir.
The section plane can be defined by two points when side, front or top view is active. In front view, click on the middle point of the rib and enter the second point somewhere under the first point in the vertical plane.
Finishing the data input, the ***Section Lines*** dialog shows up. Close it with ***OK***.

Views



Change view to ***Y-Z*** plane – or press ***Ctrl+3*** keys!

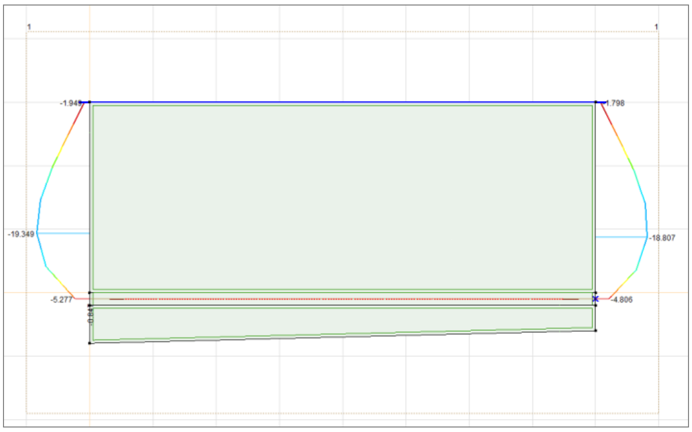


Change ***Display mode*** to ***Section line***:

Numbering

Click on ***Numbering*** speed button and turn on ***Write Values to Lines***.

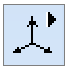
The result is the following:



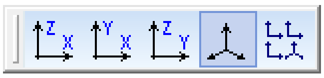
Speed buttons

Turn off ***Section Line Display mode.***

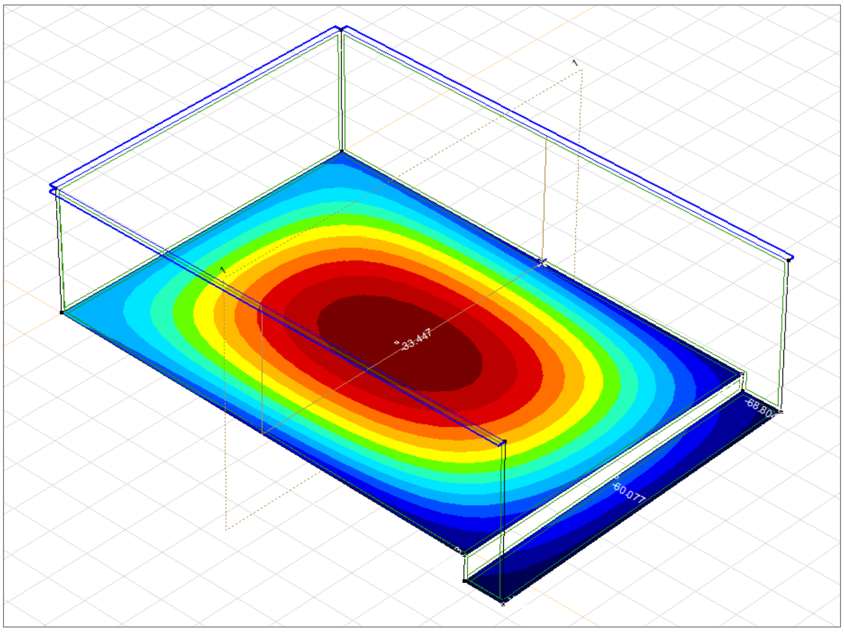
Views



Change to ***Perspective*** view.

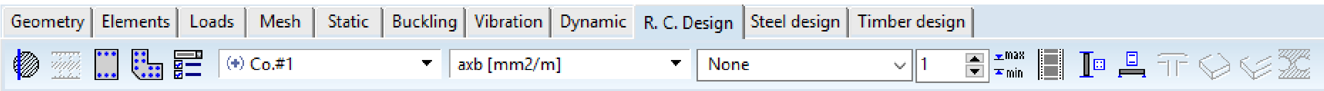


Change ***Display mode*** to ***Isosurfaces 2D***! Select **Surface Support Internal Forces** – ***Rz*** which shows vertical reactions of the surface support and the following will be displayed:

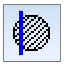


R. C. design

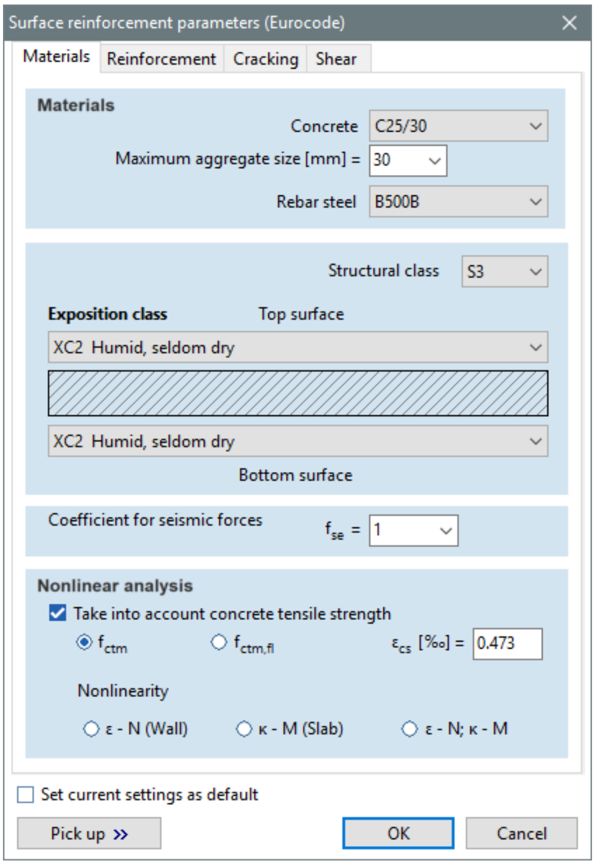
To analyse the required reinforcement of the reservoir, change tab to ***R. C. design***:



Reinforcement parameters

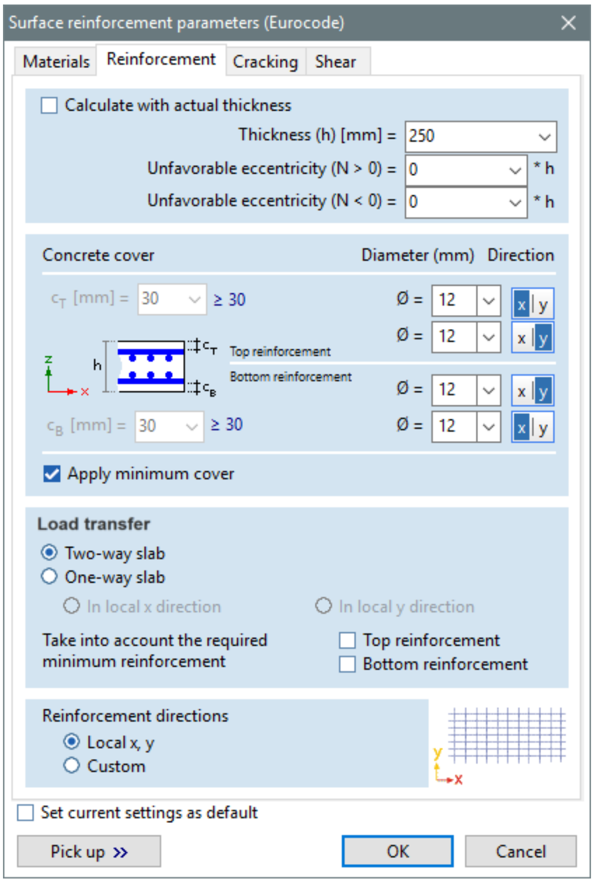


Click on ***Reinforcement parameters*** icon then select ***All*** (*) and confirm with **OK**, then the following window will be displayed:



Set ***Exposition classes*** to ***XC2*** for ***Top*** and ***Bottom surface*** on ***Materials*** tab.

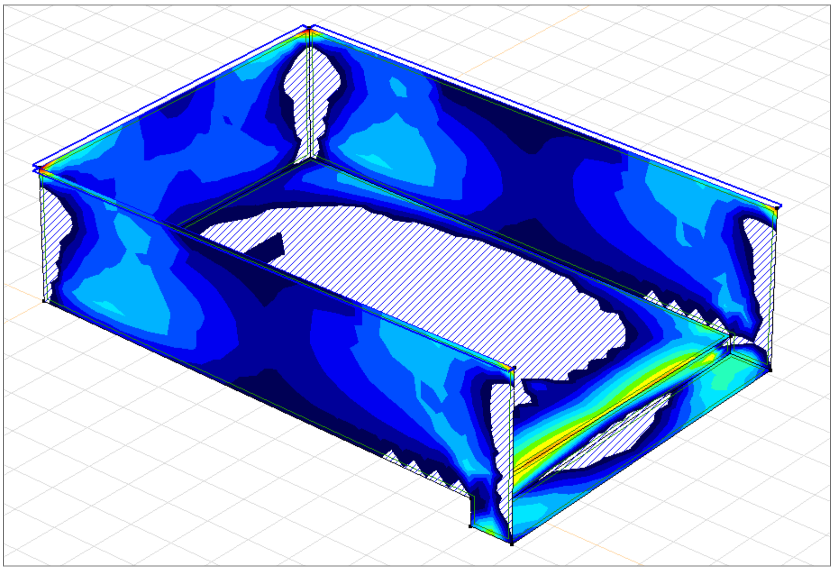
Set ***Primary direction of reinforcement*** as ***x*** for ***Top*** and ***Bottom reinforcement*** on ***Reinforcement*** tab. Check ***Apply minimum cover*** checkbox:



By closing the parameters setting dialog window with ***OK***, the results of calculated bottom reinforcement in local ***x*** direction - ***axb [mm²/m]*** can be seen on the screen. Change ***Display mode*** to ***Isosurfaces 2D***.

Speed buttons

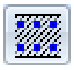
Click on ***Numbering*** speed button and turn off ***Write Values to Surfaces*** and ***Lines.*** The following will be displayed:



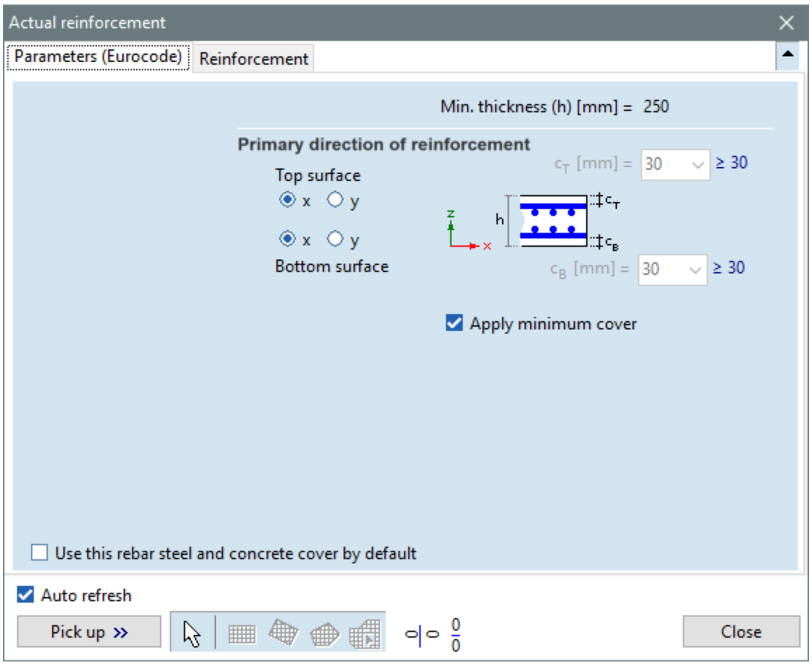
Check also ***ayb***, ***axt, ayt*** reinforcement values!

Check crack width in case of load combination ***Co.#1*** using actual reinforcement ***Ø12/150 mm*** in both ***x*** and ***y*** direction at top and bottom. Let us define actual reinforcement.

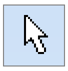
Actual reinforcement



Click on ***Actual reinforcement*** icon, the following window shows up:

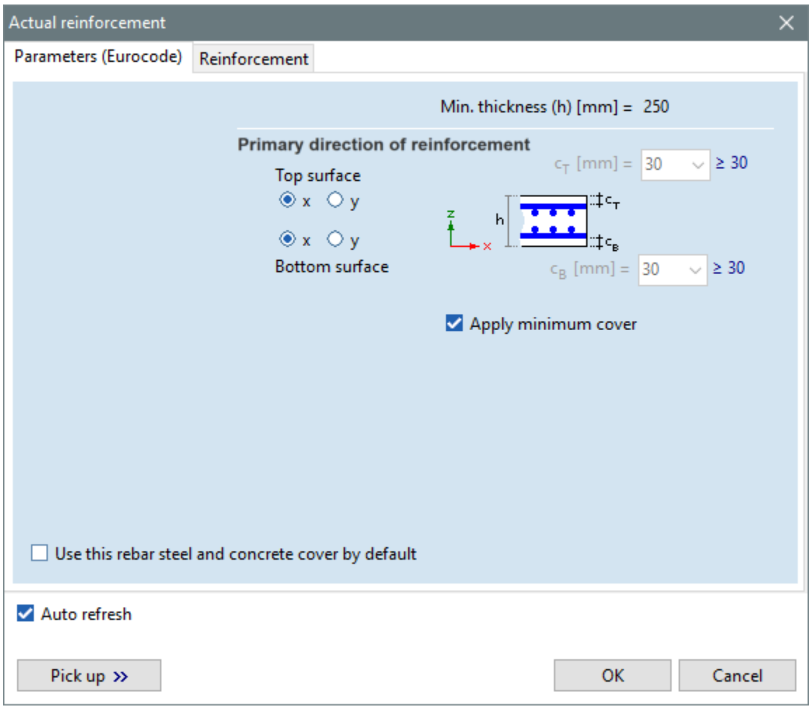


Selection



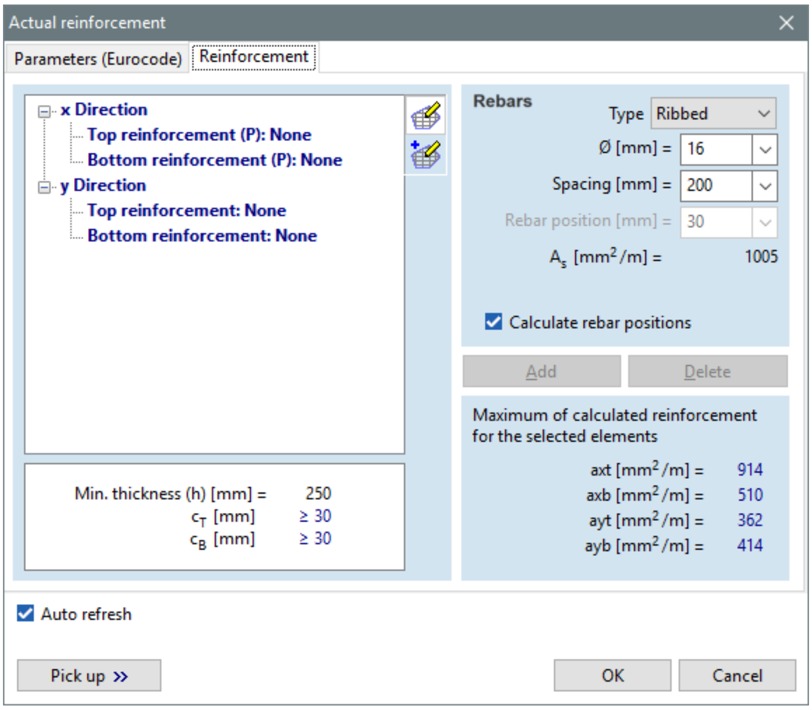
Specify the domains for which reinforcement should be assigned. Click on ***Selection*** function at the bottom of the window. Click on ***All*** (*) on ***Selection palette*** and close it with ***OK***.

The following can be seen after the selection:



Set ***x*** direction for ***Primary direction of reinforcement*** on top and bottom surface on ***Parameters*** tab and check the ***Apply minimum cover*** checkbox.

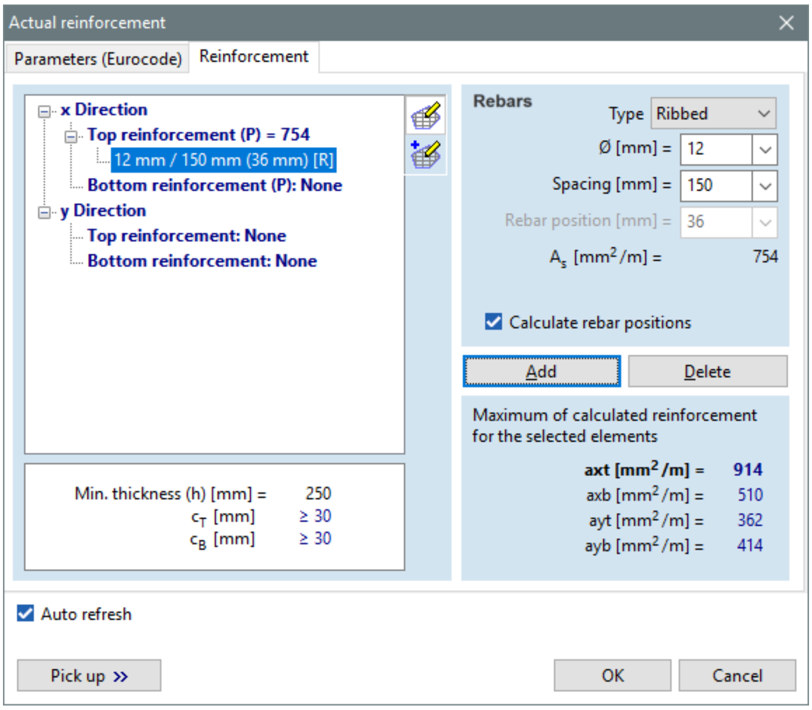
Now, click on ***Reinforcement*** tab:



Set reinforcement rebar on right hand side and assign it to each four reinforcement layers on left hand side.

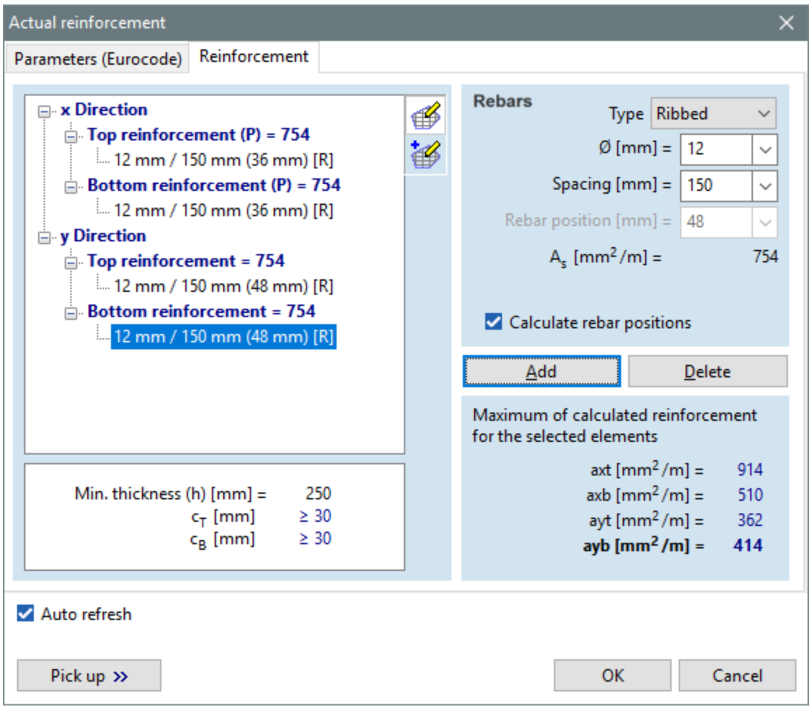
Set diameter to ***12*** for ***Ø[mm]=*** and ***Spacing [mm]*** to ***150***. Check **Calculate rebar positions** checkbox then select ***x Direction*** – ***Top reinforcement*** in top left box and click on ***Add*** button.

The following will be displayed:



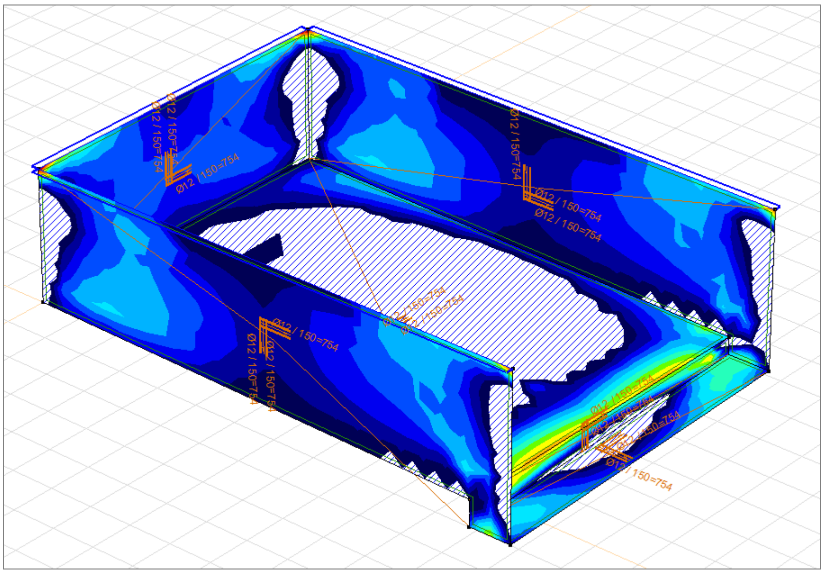
Specify the other reinforcement layers by using the above method.

Finally, the following will be the result:

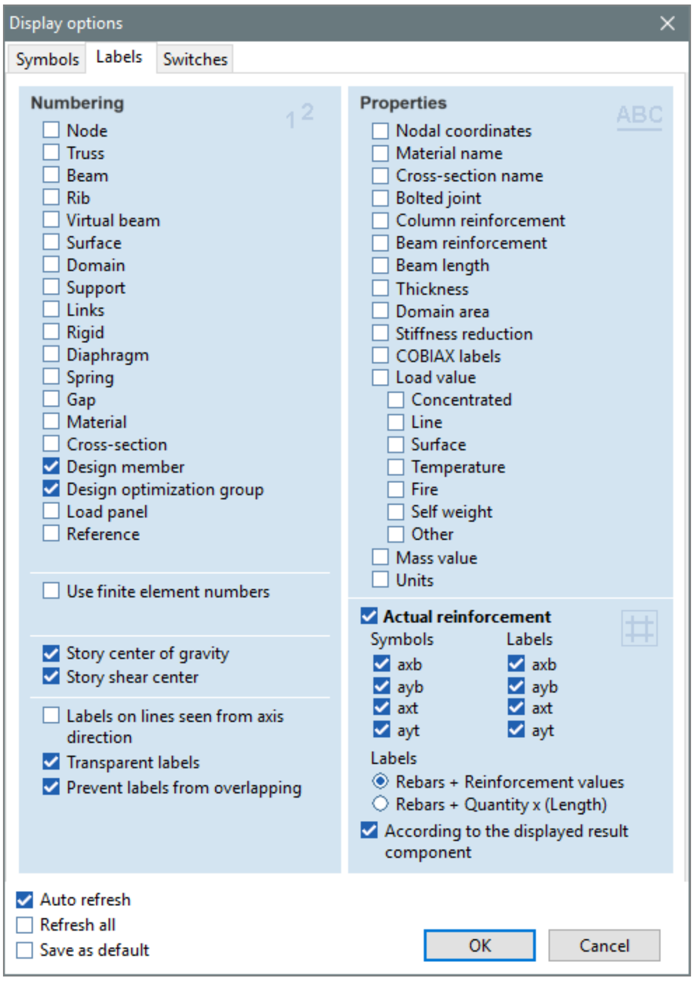


As we can see, the rebar position is calculated automatically taking into account concrete cover, the position of each reinforcement layers and the diameters of the applied rebars.

Close window with ***OK*** button, the specified reinforcement is displayed on the domains:

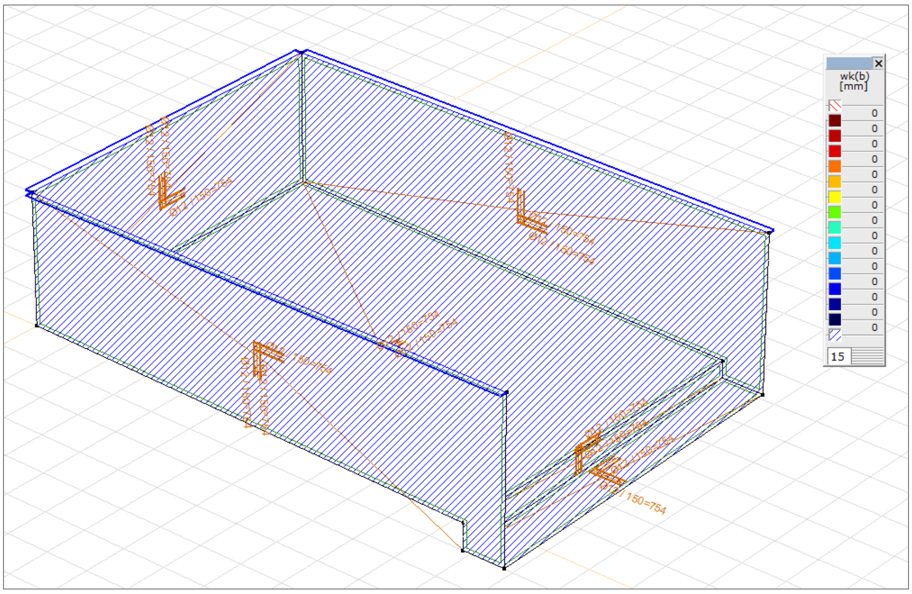


The thick brown lines and their titles indicates the given actual reinforcement. If only one direction of reinforcement is visible on the screen, then open ***Display options***. On ***Label*** tab switch off ***According to the displayed result component*** function.



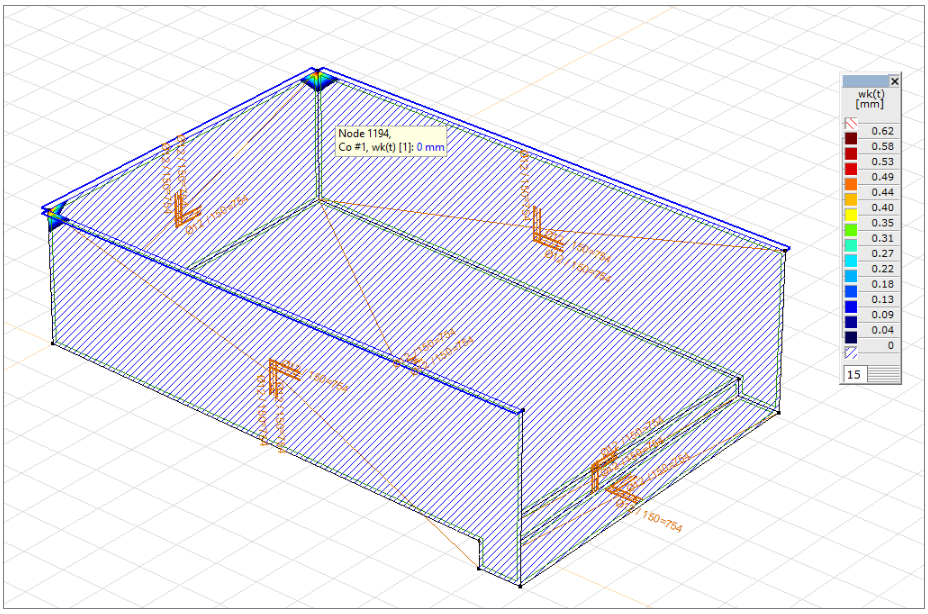
Crack width

Select ***Cracking – wk(b)*** result component which is showing crack width at bottom surface of the domains (outer face of reservoir, remark: the location of 'bottom' depends on the position of the local coordinate system).



All domains are hatched in blue what denotes that there are no cracks on outer face of domains. Check crack widths on the inner face of the reservoir!

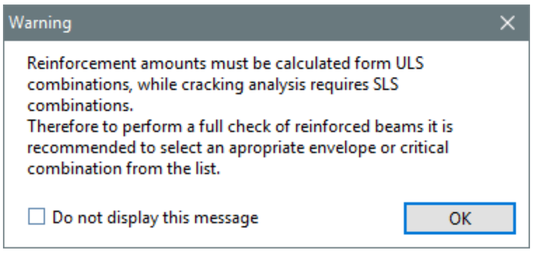
Select ***Cracking*** – ***wk(t)*** result component which is showing crack width at top surface of the domains. The following will be displayed:

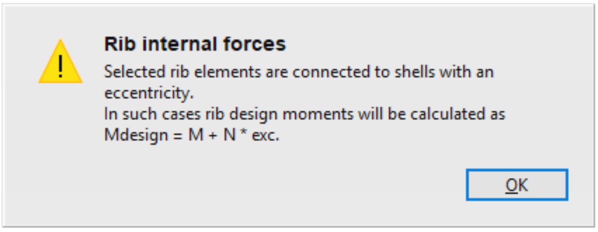


Here you can see cracks on some inner corners of the domains.

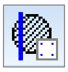
Beam reinforcement design

To design reinforcement of the ribs, click on ***Beam reinforcement design*** icon, then click on the longer, farthermost rib on top of the side wall and finally click on ***OK***. The following warning message shows up:



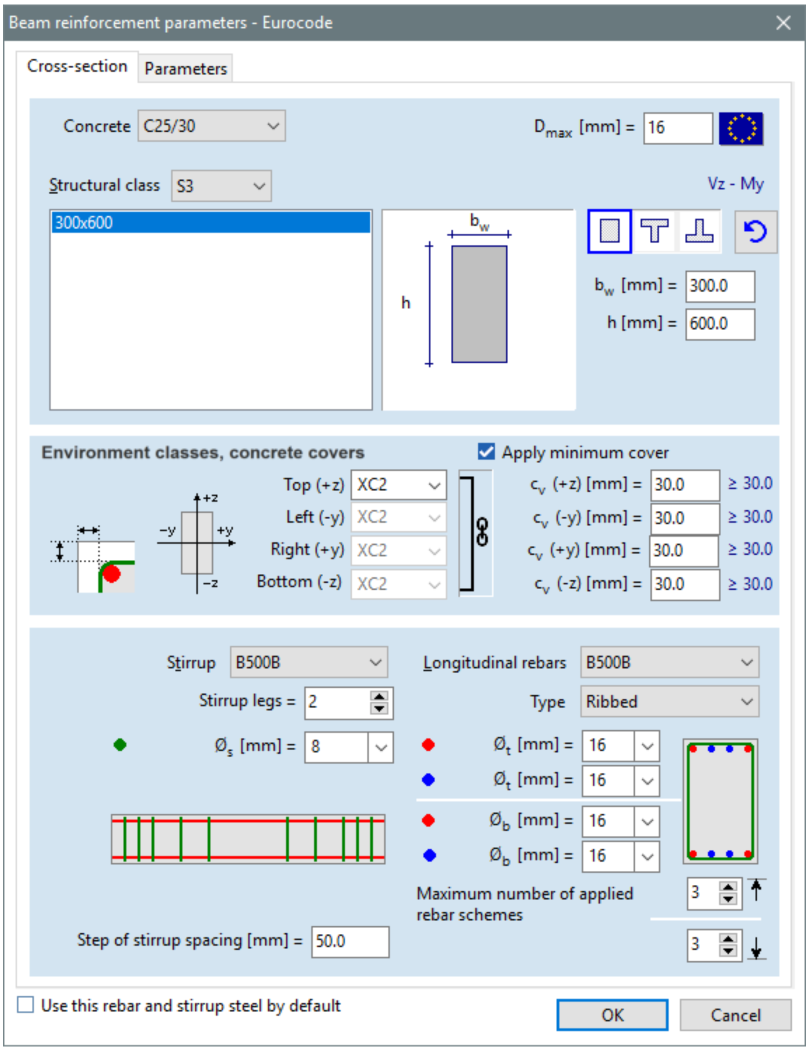


Beam reinforcement parameters



By closing warning message, the ***Beam reinforcement parameters*** window shows up.

Set the parameters as shown in the next figures:

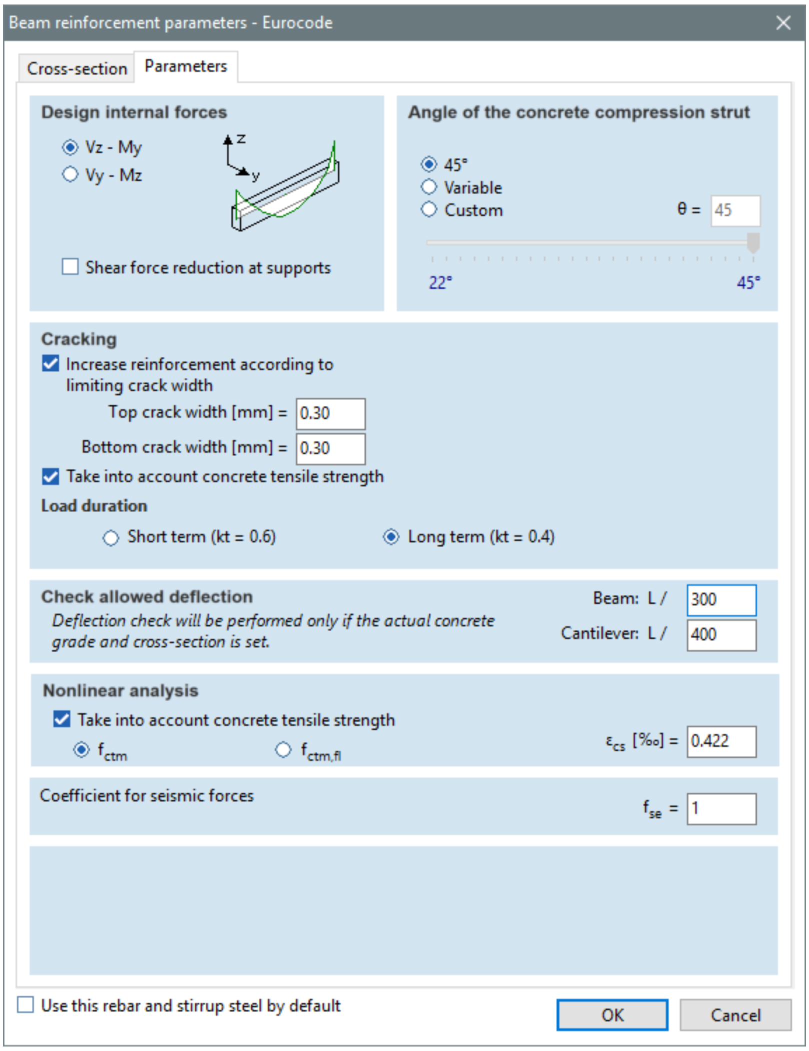


Specify rectangular shape for the rib on ***Cross-section*** tab, and select ***S3 structural class***.

Use ***XC2 Environment class*** on all four sides of the rib and check ***Apply minimum cover***.

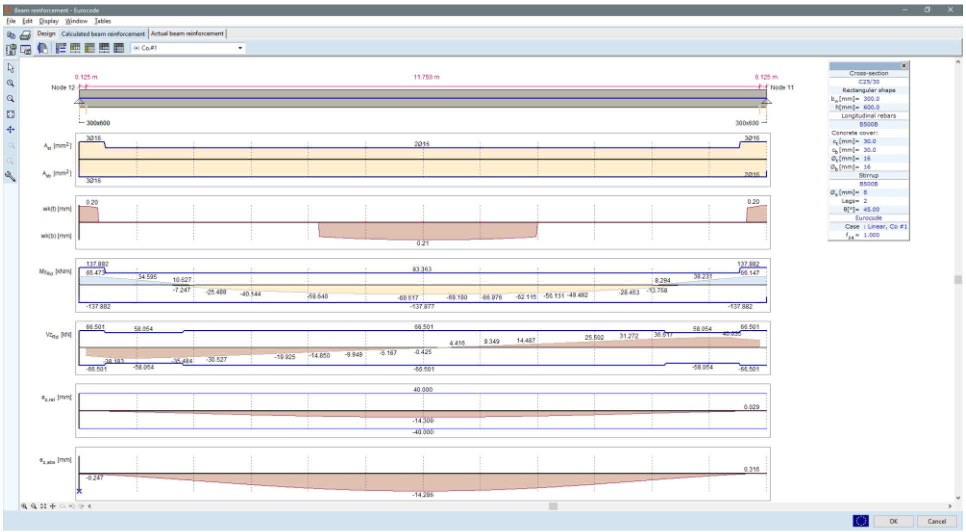
Set ***B500B*** material rebar for ***longitudinal rebars*** and ***stirrups***. Use rebar ***Ø8 mm*** for stirrup and ***Ø16 mm*** for main bars.

For crack width control check ***Increase reinforcement according to limiting crack width*** checkbox on ***Parameters*** tab:



After click on ***OK***, then the following window shows up displaying the envelope internal force diagram (in case of load combination ***Co.#1***) and the calculated amount of reinforcement (main bars and link).

By clicking on ***Calculated beam reinforcement*** tab, the required reinforcement, crack width considering the calculated reinforcement and bending resistance, shear capacity, etc... can be seen.



On Actual beam reinforcement tab, the calculated reinforcement can be assigned to the rib or can be specified according to the user's intention. This feature is not presented now, similar example can be found in chapter named '***Beam model***'.

Click on ***OK*** to close window.

Notes

Notes