

STEP BY STEP TUTORIAL

Edited by: Inter-CAD Kft. ©2018 Inter-CAD Kft. All rights reserved All brand and product names are trademarks or registered trademarks. Intentionally blank page

TABLE OF CONTENTS

| 1. | BEA | AM MODEL | 5 |
|----|-----|---|-----|
| 2. | FRA | AME MODEL | |
| 3. | SLA | AB MODEL | 63 |
| 4. | ME | EMBRANE MODEL | 103 |
| 4 | .1. | GEOMETRY DEFINITION USING PARAMETRIC MESH | |
| 4 | .2. | GEOMETRY DEFINITION USING DOMAINS | |

Intentionally blank page

1. BEAM MODEL

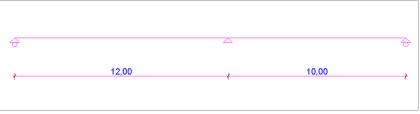
Objective

Start

New

Views

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

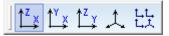
Create a new model by clicking on *New* icon. In the dialogue window replace *Model file name* with *'beam'*, select *Eurocode* from *Design code*s and set *Unit and formats* to *EU*.

| New model | | | × |
|--------------------------------|--|------------------|---|
| Select a view to start with | | <u>F</u> older | C:\Users\lstván T\Desktop\Step by Step 🗸 🖂 |
| , | <u>M</u> odel fil | e name: | beam |
| Top view | <u>U</u> nits and | | Eurocode EU Change settings Eu Change settings |
| Front view | Page header | | |
| Perspective | (Company logo) | Projec Analys | |
| | | | |
| | Project Analysis by Model: beam.axs | | |
| | | | OK Cancel |

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.

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 | - | | | | tab to de Is are dis | - | metry ar | nd structura | al propertie | es of the beam | . On the tab | the icons of |
|-----------------------------------|----------|-------|------|--------|-------------------------|-----------|----------|----------------|--------------|----------------|-------------------------------|--------------|
| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | |
| | * 4 | | • | í í | 12 🖬 | A | a 🌧 🌈 | ∉ ⊢ , | 8 1 1 | 표 🏭 İ 🛨 | <u>14</u> <u>14</u> <u>14</u> | 1 VB 🔐 |

Draw objects

directly

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



Horizontal beam

C

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:

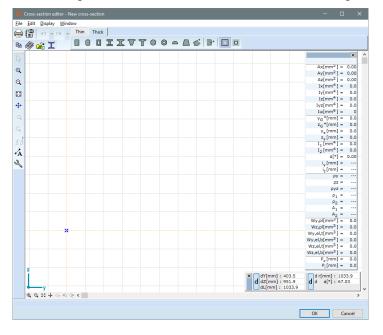
| Warning No element cross-section defined. Browse cross-section libraries Cross-section editor | |
|--|------------|
| | <u>O</u> K |

Select Browse cross-section libraries and click on OK.



Cross-section

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



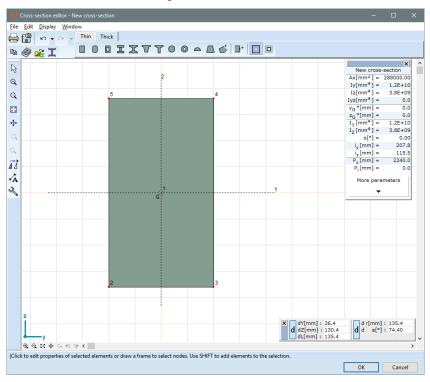
Rectangular shape



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

| Rectangular shape | × |
|-------------------|--|
| | Dimensions b [mm] = 400 h [mm] = 720 |
| h | Rotation $\alpha [°] = 0$ |
| + | Place Cancel |

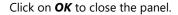
Enter **720** for height (*h[mm]*=) and click on *Place* button. (The *b* size of the rectangle shape is equal to the default value, so it does not have to be overwritten now.) Click anywhere in the workspace to place the cross-section. The following result can be seen:



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:

| 8 | | oss-section |
|---|-------|-------------------------------|
| | Name | 400x720 |
| | 🔽 Rem | nember cross-section position |
| | | <u>O</u> K <u>C</u> ancel |



The following message shows up:



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

| <u>D</u> esign code | <u>M</u> aterials | | C25/30 | | _ | |
|--|---|---|---|--|---|-------------|
| CSA S6-06 [Rev. 2010 DIN (German) Eurocode Eurocode [A] Eurocode [B] Eurocode [CZ] Eurocode [CI] Eurocode [FIN] Eurocode [FIN] Eurocode [FIN] Eurocode [RL] Eurocode [PL] Eurocode [RO] | S320GD+Z275 S350GD+Z275 C12/15 C16/20 C20/25 C25/30 C30/37 C35/45 C40/50 C45/55 C50/60 C14 C16 C18 | ^ | $\begin{tabular}{ c c c c } \hline Type & & \\ \hline E [N/mm^2] & v & \\ \hline v & & \\ \hline \alpha_T [1/^cC] & \\ \rho [kg/m^3] & \\ f_{ck} [N/mm^2] & \\ \gamma_c & & \\ \hline \alpha_{cc} & \\ \hline \phi_t & & \\ \hline \end{tabular}$ | Concrete Isotropic 31500 0.20 1E-5 2500 25.00 1.500 1.00 2.00 | | |
| Eurocode [NK] Eurocode [UK] MSZ (Hungarian) NBCC 1995 | C20 C22 C24 C27 C30 C35 | ~ | | | | OK ancel |

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 \boldsymbol{x} axis of the beam will be pointed in the direction of the beam and the local \boldsymbol{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.

| dL[m]: 13.097 | × | d | dX[m] : dY[m] : dZ[m] : dL[m] : | 9.700 0.000 | d | d r[m] : d a[°] : dh[m] : | 47.79 |
|---------------|---|---|--|----------------|---|---------------------------------|-------|
|---------------|---|---|--|----------------|---|---------------------------------|-------|

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.

| Geometry check | × |
|--|---|
| Geometry check Tolerance [m] = 0.001 Selection of problematic nodes Selection of intersecting elements Select unattached nodes or lines List deleted nodes | |
| Check domain contours Tolerance [m] = 0.001 | |
| Check all loads Rebuilding of loads may cause loss of results. | |
| OK Cancel | |

After the geometry check a summary of actions shows:

| 6 | Geometry check | | |
|---|---|---|------------|
| | Nodes deleted: Lines deleted: Surfaces deleted: | 0 | |
| | | | <u>O</u> K |

Setting

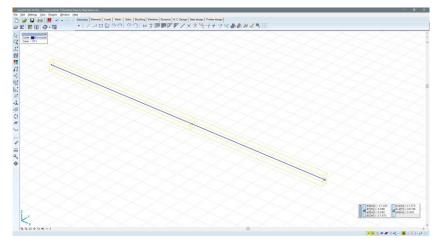
perspective view

T

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.



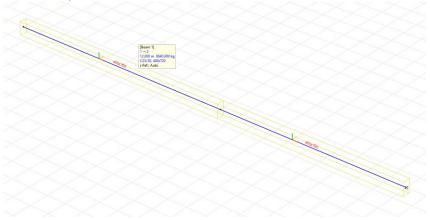
By closing the perspective panel, the view mode is preserved.

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.

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.

| Support for Node 2 | × |
|---|--|
| Define OModify | |
| Direction Global Global Referential Edge relative Reference | Z Z X Z Z X Z R Z R Z R Z R Z R Y R YY R Y |
| $R_{X} [kN/m] = 1E+10 \checkmark$ $R_{Y} [kN/m] = 0 \checkmark$ $R_{Z} [kN/m] = 1E+10 \checkmark$ $R_{XX} [kNm/rad] = 0 \checkmark$ $R_{YY} [kNm/rad] = 0 \checkmark$ $R_{ZZ} [kNm/rad] = 0 \checkmark$ | |
| Pick up » Calculation | OK Cancel |

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:

| Nodal supports | | | × |
|--|-------------|----------------------|-----------|
| Define | O Modify | | |
| Direction Global Referential Beam/Rib rel Edge relative | ative | z × | |
| Reference | ~ | Nonlinear parameters | |
| R _X [kN/m] = | 0 ~ | | |
| R _Y [kN/m] = | 0 ~ | | |
| R _Z [kN/m] = | 1E+10 ~ | | |
| R _{XX} [kNm/rad] = | 0 ~ | | |
| R _{YY} [kNm/rad] = | 0 ~ | | |
| R _{ZZ} [kNm/rad] = | 0 ~ | | |
| Pick up >> | Calculation | | OK Cancel |

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.

| Geometry Elements Lo | oads Mes | Static | Buckling Vib | pration Dynamic | R. C. Design | Steel design | Timber design | | |
|----------------------|----------|--------------|--------------|-----------------|--------------|----------------------|---------------|-------------|---|
| 프 • 블' 수 쇼 | ⊉ 📶 | d 🖉 (| • 🥭 🛛 | 📎 🗱 😹 🕯 | G ≞‡ | • ^F • 🚺 🛔 | ▲ <u>▲</u> | 🗠 🗽 亜 亜 🛓 🖁 | 5 |

Load cases and load groups

Ш

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

| X4 Load groups and load cases | | | | - | | × |
|--------------------------------------|---|---------------------------------|------------------|------------------|--------|------|
| ⊡ Ungrouped Luu ST1 () | Load case ST1 contains no load Load group Ungrouped | ds. V | <u>Duplicate</u> | lww - | ₹ | |
| | Eurocode | | | | | |
| | Load group | | | | E E | |
| × Delete | | Cr <u>i</u> tical load group co | mbinations | ОК | Ca | ncel |

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

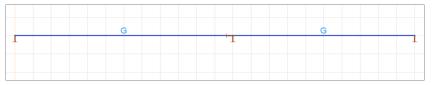
⇔

Self weight

G

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

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/s² 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:

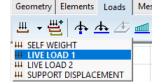
| istributed loads (Be | am 1) | | × |
|----------------------|---|-------------------------|--------------------------------|
| Oefine | O Modify | | |
| Direction | ong element rojective | | |
| Туре | Position | | |
| 4 | By length By ratio | a ₁ = 0 | a₂ = 1.000 € |
| 4 | | Startpoint | Endpoint |
| 4 | p _{X1} [kN | l/m] =0 🗸 | p _{X2} [kN/m] =0 |
| | p _{Y1} [kN | l/m] = <mark>0 ∨</mark> | p _{Y2} [kN/m] =0 ∨ |
| | p _{Z1} [kN | l/m] = <mark>0 ∨</mark> | p _{Z2} [kN/m] =0 ∨ |
| | m _{TOR1} [kNm | n/m] =0 ~ | m _{TOR2} [kNm/m] =0 ~ |
| Pick up » | | | OK Cancel |

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

Щ. –

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

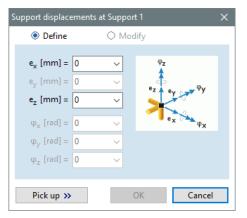


The list contains all the load cases defined and the actual one is highlighted with blue background. Select load case *LIVE LOAD 2* with mouse or arrow keys.

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

۵e ک[‡]

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 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).

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

| X dX[m]: 9.084 dY[m]: 0 dZ[m]: -7.777 dL[m]: 11.958 | d r[m] : 11.958 d a[°] : 319.43 dh[m] : 0 |
|--|---|
| | □ Beam Support |
| 📃 🗙 🖾 🗐 🕷 🖷 🖽 | 🔇 🗤 🔠 👭 🐉 🖓 😕 12 👘 |

Load combinations

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

| le [dit Ermit Beport Help MODELDATA Materials (1) -Consistentials (1) -Consistentials (1) -References Benerets Benerets Benerets Benerets Benerets Cash dit argue combinations by load cases Type SELF WBGHT [LNE LOAD 1 LIVE LOAD 2 SUPPORT DSPLACEMENT Comment Data Benerets Benerets Benerets Cash diff (22) Benerets Cash diff (22) Benerets Cash diff (22) Cash di |
|---|
| - Materials (1) Custom load combinations by load cases - Cross-sections (1) |
| Consistention (1) Custom load combinations by load cases XLAM timetre panels Type SELF WEIGHT LIVE LOAD 1 LIVE LOAD 2 SUPPORT DISPLACEMENT Comment Bements Bements SELF WEIGHT (22) SELF WEIGHT (22) SELF WEIGHT (11) BUT LOAD 1 (11) SUPPORT DISPLACEMENT (11) Lead case (6) SELF WEIGHT (22) SELF WEIGHT (23) |
| Britemans type SEF-WeigH1 LVE LOAD 1 LVE LOAD 2 SUPPORT DISPLACEMENT WeigH1 V/E LOAD 1 LVE LOAD 2 SUPPORT DISPLACEMENT Comment Ip Benerats Image: Second and an analysis Image: Second and analysis Image: Second analysis Image: Second analysis Ip Benerats Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis Image: Second analysis |
| Node (2) |
| Loads \$ \$SLY WE GAD 1(1) \$ UVE IGAD 2(1) \$ UVE IGAD 2(2) \$ |
| Image: Star Weight (22) Image: UNE LOAD 1 (1) Image: UNE LOAD 2 (2) |
| Control Nad combinations Optimized combinations Optimized combined Optimized combined Optimized combined |
| LTman VIII |
| OK Car |

New row

| () | | |
|----------------------|----------|-------------|
| SELF WEIGHT | 1.3 5 | <tab></tab> |
| LIVE LOAD 1 | 1.5 0 | <tab></tab> |
| LIVE LOAD 2 | 0 | <tab></tab> |
| SUPPORT DISPLACEMENT | 1.0 0 | <tab></tab> |
| | | |
| SELF WEIGHT | 1.3 5 | <tab></tab> |
| LIVE LOAD 1 | 0 | <tab></tab> |
| HASZNOS 2 | 1.5 0 | <tab></tab> |
| | 1.0 | <tab></tab> |

each load cases must be set. Set load factors as follows:

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

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

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

| SELF WEIGHT | 1.0 0 | <tab></tab> | | | | |
|---|--------------|---|---|--------------------------------|-----------|----------------|
| LIVE LOAD 1 | 0 | <tab></tab> | | | | |
| LIVE LOAD 2 | 0.6 0 | <tab></tab> | | | | |
| SUPPORT DISPLACEMENT | 0 | <tab></tab> | The follow | ving table sl | nows afte | er data input: |
| 20 Table Browser | | | | - | • • × | |
| Bite Edit Figmed Beyond Legical MODULDATA | y load cases | WEIGHT LIVE LOAD 1 1.35 1.50 1.35 0 1.00 0 | LIVE LOAD 2 SUPPORT DISPLA 0 1.30 0.40 | EMENT Commi 100 100 0 | ant | |
| Editing Co #3, Load combination name | | | | OK | Cancel | |

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.

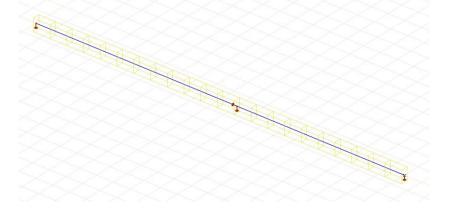
| Geometry Element | s Loads | Mesh | Static | Buckling | Vibration |
|------------------|---------|----------|--------|----------|-----------|
| $\gamma = 0$ | | # | | | × |

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

| Mesh parameter for line elements | × |
|--|-------------|
| Meshing criterion | |
| O Maximum deviation from arc | d [m] = 🔍 🗸 |
| Maximum element size | d [m] = 🔍 🗸 |
| Division into N segments | N = 12 🗸 |
| O By angle | d [°] = 🗸 🗸 |
| | |
| | OK Cancel |

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:



| Click on Static tab for running | analysis: |
|--|-----------|
|--|-----------|

| Geometry Elements Loads Mesh | Static Buckling Vibration | Dynamic R. C. Design Steel design | Timber design | |
|------------------------------|---------------------------|-----------------------------------|---------------|--|
| | ▼ | ▼ | ~ | |

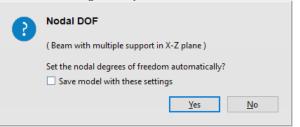
Linear static analysis

4

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:

| 🚨 (x64) Linear analysis of beam.axs | - | | × |
|-------------------------------------|---|----|---|
| Analysis of beam.axs completed. | | | |
| | | | |
| | | | |
| Close if completed | | OK | |
| | | | |
| Messages | | | * |
| Statistics | | | « |

Click on Statistics to see more information about the analysis:

Static

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.

| 💥 (x64) Linear analysis of beam.axs | | | × |
|-------------------------------------|--|--------------|----|
| Analysis of beam.axs completed. | | | |
| | | | |
| | | | |
| | | 01/ | _ |
| Close if completed | | ОК | |
| Messages | | | « |
| Statistics | | | × |
| Statistics | | | • |
| Number of equations | 69 🗙 🕺 | Truss | - |
| Equations memory | 3 k | Beam | 22 |
| Estimated memory requirement | | Rib | - |
| Solver block size | | Spring | - |
| Largest available memory block: | | Gap | - |
| Analysis block size | | Link | - |
| Available physical memory: | | Edge hinge | - |
| Total physical memory: | | Membrane | - |
| | Intel(R) Core(TM) i5-6500 CPU @ 3.20GHz (4x) 3192 MHz | Plate | |
| Single thread | | Shell | - |
| Model optimization | 00:00 | Diaphragm | - |
| Model verification | 00:00 | Load case | 4 |
| Analysis | 00:00 | Combination | |
| | | Complitation | 5 |
| Result file size: 138 k. | 00:00 | | |

Result display parameters

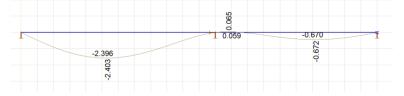


The following window shows up after clicking on **Result display parameters** icon and do the following settings:

| Display parameters | | × |
|--|--|---------------|
| Linear analysis Nonlinear analysis Dynamic analysis Case Envelope Critical | | |
| Load case III SELF WEIGHT v | Component eZ (mm) Scale by 10 Display mode Display mode Display shape Undeformed Deformed Write values to Virite values to Surfaces Min./Max. only Miscellaneous settings C ut moment peaks over columns | Section lines |
| | Refresh all | |
| | | OK Cancel |

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:



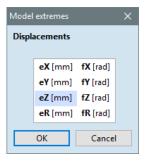
The scale of the diagram can be given or set at the field next to the **Display mode**.

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**.

| Geometry Elements | Loads Mesh Static Buckling Vibration Dynamic R. C. Design Steel design Timber des |
|-------------------|---|
| | HI SELF WEIGHT |
| | (*) Co.#3 (SLS Quasipermanent) a. b. Envelopes |

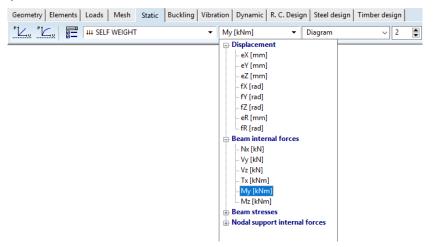
Min, max values

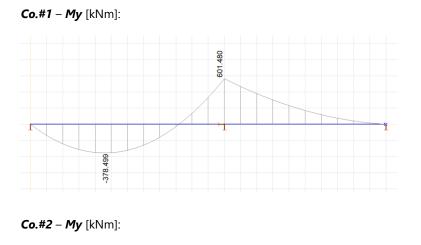
To find minimal and maximal values click on **Min, Max Values** icon. The following window shows up:

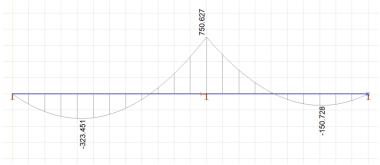


Select *ez* and click on *OK*. The result panel shows the highest negative deformation and its position, after clicking on *OK* the highest positive deformation is shown. Click again on *OK* to close.

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*).







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

| Geometry Elements Loads Mesh Static | Buckling Vibration Dynamic | R. C. Design Steel design Timber design |
|-------------------------------------|----------------------------|---|
| () 💮 🔛 🛄 🚰 😭 (+) Co.#1 (ULS) | ▼ Nx [kN] | ▼ None ~ 1 💽 ≭min 🔳 📭 📮 🗍 🚸 🖉 🌋 |

Beam reinforcement
designClick 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 Cross-section Parameters Concrete C25/30 D_{max} [mm] = 16 \sim \odot Vz - My Structural class S4 ~ Д Т 5 b_w [mm] = 400.0 h h [mm] = 720.0 Environment classes, concrete covers Apply minimum cover Top (+z) XC1 c_v (+z) [mm] = 25.0 ≥ 25.0 \sim Left (-y) XC1 c_v (-y) [mm] = 25.0 ≥ 25.0 8 Right (+y) XC1 ≥ 25.0 c_v (+y) [mm] = 25.0 Bottom (-z) XC1 ≥ 25.0 c_v (-z) [mm] = 25.0 Stirrup B500B \sim Longitudinal rebars B500B \sim Stirrup legs = 2 * Ribbed Туре \sim Ø_s [mm] = 10 Ø_t [mm] = 16 ~ Ø_t [mm] = 25 ~ Ø_b [mm] = 25 \sim Ø_b [mm] = 25 ~ € ↑ 3 Maximum number of applied rebar schemes Step of stirrup spacing [mm] = 50.0 € ↓ 3 Use this rebar and stirrup steel by default OK Cancel

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 ($\emptyset t$, $\emptyset b$) 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 following | js: |
|---|--|
| eam reinforcement parameters - Eurocode | × |
| Cross-section Parameters | |
| Design internal forces Vz - My Vy - Mz Shear force reduction at supports | Angle of the concrete compression strut (a) 45° (b) Variable (c) Custom $\theta = 45^{\circ}$ 22° 45° |
| Cracking ✓ Increase reinforcement according to limiting crack width Top crack width [mm] = 0.30 Bottom crack width [mm] = 0.30 ✓ Take into account concrete tensile strength Load duration ♦ Short term (kt = 0.6) |) Long term (kt = 0.4) |
| Check allowed deflection Deflection check will be performed only if the a grade and cross-section is set. | Beam: L / 250 Cantilever: L / 400 |
| Nonlinear analysis ☑ Take into account concrete tensile strength ④ f _{ctm} ○ f _{ctm,fl} Coefficient for seismic forces | $\epsilon_{cs} [\%_{o}] = 0.409$ $f_{se} = 1$ |
| | |
| Use this rebar and stirrup steel by default | OK Cancel |

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.

| Warning | × |
|--|----------|
| Reinforcement amounts must be calculated form L combinations, while cracking analysis requires SLS combinations. Therefore to perform a full check of reinforced bea recommended to select an apropriate envelope or combination from the list. | ms it is |
| Do not display this message | OK |

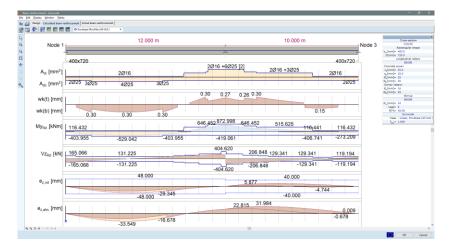
After taking notice of the warning, *Beam reinforcement* window in the background becomes active.

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.

| splay | | > |
|--|----------|-------------------------|
| Diagrams | Display | Labeling of extremes |
| Model | Z | |
| Envelope of reinforcement [A _{st}] | Z | ~ |
| Side reinforcement against torsion [A _{sl,T+}] | | |
| Cracking [wk(t)] | V | ~ |
| Stirrup spacing [s _w] | | |
| Moment resistance [My _{Rd}] | V | ~ |
| Maximum shear force [Vz _{Rd}] | Z | ~ |
| Torsion resistance [Tz _{Rd}] | | |
| Bending moment utilization [My _{Ed} /My _{Rd}] | | |
| Shear utilization [Vz _{Ed} /Vz _{Rd}] | | |
| Torsion utilization [Tz _{Ed} /Tz _{Rd}] | | |
| Relative deflection [e _{z,rel}] | ~ | ~ |
| Absolute deflection [e _{z,abs}] | ~ | ~ |
| Relative deflection, utilization [e _{z,rel} /e _{zmax}] | | |
| Absolute deflection, utilization [e _{z,abs} /e _{zmax}] | | |
| Display beam with its real proportions Show allowed deflection Vertical grid | dx [m |] = 1.000 |
| Set current settings as default | OK | Cancel |

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.

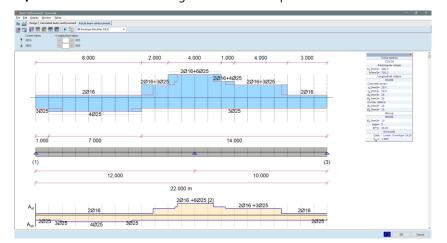


In the upper figure, pink coloring indicates the required reinforcement in case of the selected load combination (or load case). The blue coloring indicates the actual reinforcement (minimum reinforcement in the corners of the link). The vertical dashed lines represent the meshing of the beam (see before), 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.

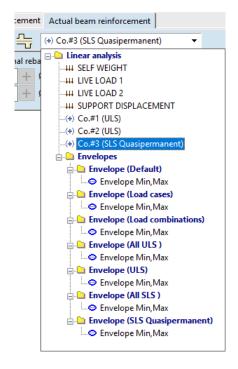
| | Corner rebars +Longitudinal rebar | | | | | | | | |
|---|-----------------------------------|---|--|---|-----|--|--|--|--|
| ↑ | 2Ø16 | — | | + | Ø25 | | | | |
| ¥ | 2Ø25 | — | | + | Ø25 | | | | |

Select *Envelope Min, Max (ULS)* combination in the dropped-down list next to *Apply calculated re-inforcement* 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.



Apply calculated reinforcement

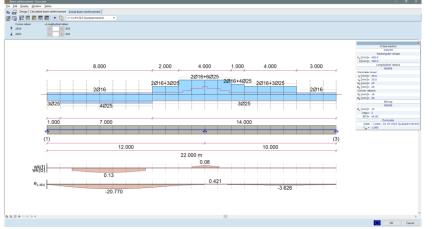
Sets display parameters



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.

| iagrams | Displa | · | eling of tremes |
|--|-----------------------|---------------|--------------------|
| Model | ~ | | |
| Envelope of reinforcement [A _{st}] | | | |
| ide reinforcement against torsion [A _{sl,T+}] | | | |
| Cracking [wk(t)] | ~ | | |
| Stirrup spacing [s _w] | | | |
| Moment resistance [My _{Rd}] | | | |
| Maximum shear force [Vz _{Rd}] | | | |
| 「orsion resistance [Tz _{Rd}] | | | |
| Bending moment utilization [My _{Ed} /My _{Rd}] | | | |
| Shear utilization [Vz _{Ed} /Vz _{Rd}] | | | |
| Forsion utilization [Tz _{Ed} /Tz _{Rd}] | | | |
| Relative deflection [e _{z,rel}] | | | |
| Absolute deflection [e _{z,abs}] | ~ | | |
| Relative deflection, utilization $[e_{z,rel}/e_{zmax}]$ | | | |
| Absolute deflection, utilization [e _{z,abs} /e _{zmax} |] | | |
| Diagram size | | | |
| Margins Lef | 5% ~ | Right 25% | 5 V |
| ○ Zoomable | rtical size of result | diagrams 100 | % ~ |
| Display beam with its real proportions Show allowed deflection Vertical grid | | dx [m] = 1.00 | 0 |
| Set current settings as default | | | |

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



Nonlinear static analysis



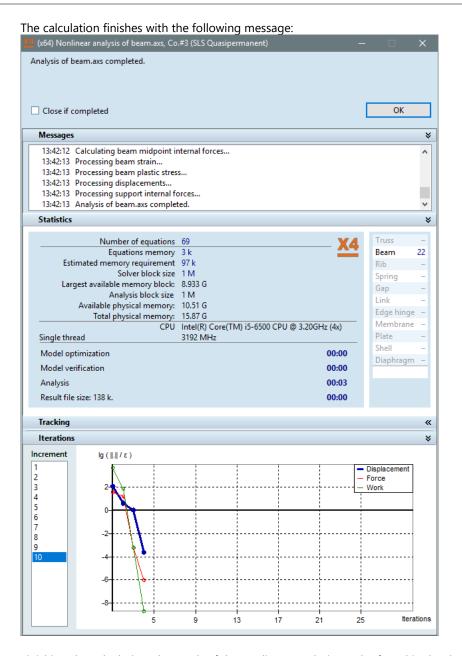
The above results for deflection are results of an approximate calculation (creep is approximated but without considering the effect of shrinkage). A more accurate calculation can be made by nonlinear static analysis. This can be done by closing the **Beam reinforcement** window and going back to **Stat***ic*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

Load cases Convergence criteria 🖃 🐺 All Maximum iterations 30 Load cases Displacement 0.001 E Gad combinations Force ULS ULS Co.#1 (ULS) Co.#2 (ULS) 0.001 Work Use secant stiffness (only in appropriate cases) Co.#3 (SLS Quasipermanent) Use reinforcement in calculation 1 of 7 Actual reinforcement Solution control ✓ Creep
✓ Shrinkage Tracked node **φ**≠0 Force
 Displacement Direction: X Nonlinearity ✓ Follow nonlinear behaviour of ✓ Follow geometric nonlinearity of materials and finite elements beams, trusses, ribs and shells Load factor Equal increments 1.0000 10 Number of increments O Increment function Store last increment only <Bilinear> 0 OK Cancel 10

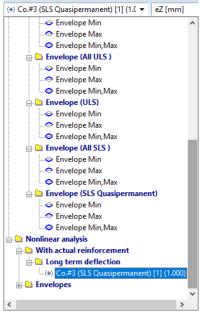
The only purpose of the nonlinear analysis is to study the deformation of the structure, so check **Co.#3** (**SLS Quasipermanent**) load combination in **Load cases** list. Check **Use reinforcement in calculation**, and considering **Creep** and **Shrinkage**. Set the other parameters according to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

| 🎽 (x64) Nonlinear analysis of beam.axs, Co.#3 (SLS Quasipermanent) — 🛛 🗌 | | | | | | | | | | |
|--|------|--|--|--|--|--|--|--|--|--|
| Beam element equilibrated load assembly | | | | | | | | | | |
| | | | | | | | | | | |
| 70% | | | | | | | | | | |
| Close if completed | | | | | | | | | | |
| | | | | | | | | | | |
| Messages | ≽ | | | | | | | | | |
| 13:42:11 Nodal displacement vector assembly | ^ | | | | | | | | | |
| 13:42:11 Updating displacements 13:42:11 Beam element strain evaluation | | | | | | | | | | |
| 13:42:11 Beam element stress evaluation | | | | | | | | | | |
| 13:42:12 Beam element equilibrated load evaluation | | | | | | | | | | |
| 13:42:12 Beam element equilibrated load assembly | × | | | | | | | | | |
| Statistics | ≷ | | | | | | | | | |
| Number of equations 69 YA Truss | - | | | | | | | | | |
| | 22 | | | | | | | | | |
| Estimated memory requirement 97 k Rib Solver block size 1 M Soring | - | | | | | | | | | |
| Solver block size T M Spring Largest available memory block: 8.933 G Gap | - | | | | | | | | | |
| Analysis block size 1 M | _ | | | | | | | | | |
| Available physical memory: 10.51 G Total physical memory: 15.87 G Edge hinge | - | | | | | | | | | |
| CPU Intel(R) Core(TM) i5-6500 CPU @ 3.20GHz (4x) Membrane | - | | | | | | | | | |
| Single thread 3192 MHz Plate | - | | | | | | | | | |
| Model optimization 00:00 Shell | - | | | | | | | | | |
| Model verification 00:00 | | | | | | | | | | |
| Analysis 00:01 | | | | | | | | | | |
| Result file generation | | | | | | | | | | |
| | | | | | | | | | | |
| Tracking | * | | | | | | | | | |
| Iterations | ¥ | | | | | | | | | |
| Increment Ig (. / ε) | | | | | | | | | | |
| 1 Displacement – Force | | | | | | | | | | |
| 3 3 | | | | | | | | | | |
| 4 2 | | | | | | | | | | |
| 5 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 | | | | | | | | | | |
| | | | | | | | | | | |
| | | | | | | | | | | |
| -1 | | | | | | | | | | |
| -2 | | | | | | | | | | |
| -3 | | | | | | | | | | |
| 4 | | | | | | | | | | |
| 5 9 13 17 21 25 Iterat | ions | | | | | | | | | |
| | | | | | | | | | | |

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.



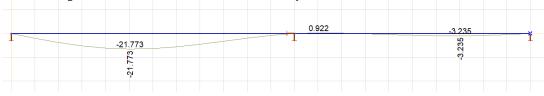
Finishing the calculation, the result of the nonlinear analysis can be found in the drop-down list.



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**)

| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | | |
|----------|----------|----------|------------|----------|--------------|-----------|---------|--------------|--------------|---------------|------------|-----------------------|---|
| | | (+) Co.# | #3 (SLS Qi | uasiperm | anent) [1] (| 1.(🔻 eZ | [mm] | ▼ D | liagram | ~ 30 | max min | Ť ^y ₹ ₹ | _ |

The following result is obtained from the nonlinear analysis:



2. FRAME MODEL

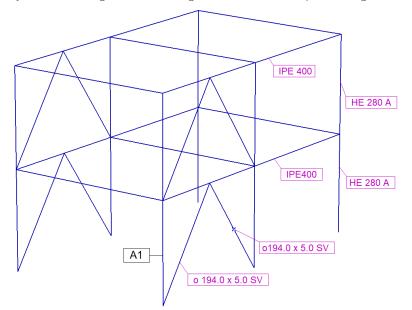
Objective

Start

New

Ρ

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 \times 5.0 SV. Use material S235 and apply Eurocode standard for the design.

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

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*.

| New model | | |
|-------------------------------------|--|--|
| Select a view to start with | <u>F</u> <u>M</u> odel file n | older C:\Users\István T\Desktop\Step by Step\FRAME\ V 2012 |
| Top view | Design Units and for <u>R</u> eport land | |
| Front view Z V Perspective | Page header (Company logo) | Project Analysis by Comment |
| | Project Analysis by Model: frame.axs | |
| | | OK Cancel |

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 -

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

Elements

| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | |
|----------|----------|------------|-----------------------|--------|----------|-----------|---------|--------------|--------------|---------------|--------------|
| | * L | d (| 0 (| í í | 1 🖌 🖿 | | a 🌧 🌈 | 「戽」, | \$ 1 1 | 표종 👎 | × 14 14 14 🌆 |

Draw objects



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

| [| × 🔁 😂 🖉 🔲 |
|---------------------|--------------|
| Туре | Beam |
| Material | ▼ >> |
| Allow variable cro | |
| Cross-section | |
| Local x orientation | Upwards |
| Local z reference | × Auto |
| End releases | |
| Height [m] | 3.000 |
| | |
| | |

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

The following window shows after clicking:

| Warning No element cross-section defined. Browse cross-section libraries Cross-section editor | |
|--|--|
| <u>Q</u> K | |

| vrary Parametric shapes in Thick | HE European wid | le flange beams | | | Paramete | ers | |
|---|-----------------------------------|-------------------------|-------------------------|----------------------------------|--|--------------------------|---|
| in Thick | | | | | | | 1 |
| ΤΠΤΤΟΟΠΠ | Name | Height [mm] | LTUIU | Ax [mm²] | h (mm) b (mm) tw (mm) | 270.0 A 280.0 8.0 | |
| | HE 240 B HE 240 M | 240.0 270.0 | 240.0 248.0 | 10600.31 19960.32 | Ax [mm ²] | | |
| +×< | HE 260 A HE 260 AA HE 260 B | 250.0 244.0 260.0 | 260.0 260.0 260.0 | 8684.24 6899.24 11846.74 | Ay [mm ²] Az [mm ²] Ix [mm ⁴] | 2120.40 625010.4 | |
| - I shapes | HE 260 B HE 260 M HE 280 A | 200.0 290.0 270.0 | 268.0 | 21966.74 9728.74 | ly [mm ⁴] Iz [mm ⁴] Ivz [mm ⁴] | 1.367626E8 4.762693E7 | |
| HD wide flange columns HE European wide flange beams | HE 280 AA HE 280 B | 264.0 280.0 | 280.0 | 7804.74 | ιω [mm ⁶] | | |
| HL beams with very wide flanges HP wide flange bearing piles | HE 280 M HE 300 A | 310.0 | 288.0 | 24018.74 | | | |
| IPE European I-beams IPN European standard beams | HE 300 AA | 283.0 | 300.0 | 8893.68 | | | |
| UB British universal beams UC British universal columns | HE 300 B HE 300 C | 300.0 320.0 | 300.0 305.0 | 14910.69 22510.68 20210.60 | ~ | | |
| er: Europe | No filtering | | | | | ! | |

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*.

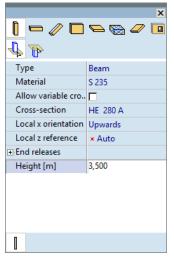
The following message shows up:



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.

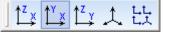
Set *Height* to 3.5 m in the window:



Note: in the window several parameters can be set which are not mentioned here. **End releases** can also be specified or the local coordinate 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.)

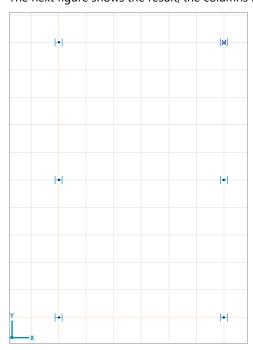


Change view to top view (**X**-**Y** plane), if it is not the actual view. Note: if necessary, view can be modified with this icon, even if an editing command is active.



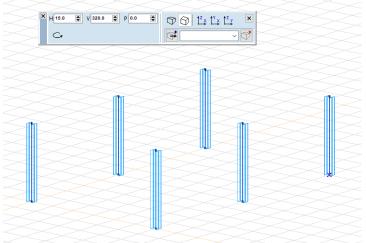
Column

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:





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





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**.

| ibrary Parametric shapes | | IPE European I-bea | ms | | | | Parameter | rs | |
|--|-----|--------------------------------|-------------------------|----------------|----------------------|---|---|------------------------|---|
| Thin Thick | | Name 🔺 | Height [mm] | Width [mm] | Ax [mm²] | | h [mm] b [mm] tw [mm] | 400.0 180.0 8.6 | D |
| - I II I T O O D | П | IPE 220 | 220.0 | 110.0 | 3337.62 | ^ | tf [mm] Ax [mm ²] | 13.5 8448.12 | |
| | С | IPE 240 IPE 270 | 240.0 270.0 | 120.0 135.0 | 3912.52 4595.40 | | Ay [mm ²] Az [mm ²] | 4537.16 | 5 |
| $+ + \times \land \leftarrow = = \cdots$ | | IPE 300 IPE 330 | 300.0 330.0 | 150.0 160.0 | 5382.10 6261.92 | | Ix [mm ⁴] | 512579.6 2.313401E8 | 5 |
| - I shapes | | IPE 360 IPE 400 | 360.0 400.0 | 170.0 180.0 | 7274.22 8448.12 | | Iz [mm ⁴] Iyz [mm ⁴] | 1.317857E7 0 | 7 |
| HE European wide flange beams HL beams with very wide flanges | | IPE 450 IPE 500 | 450.0 500.0 | 190.0 200.0 | 9883.83 11553.92 | | Ιω [mm ⁶] | 4.826562E11 | 1 |
| HP wide flange bearing piles | | IPE 550 IPE 600 | 550.0 600.0 | 210.0 220.0 | 13443.90 15600.75 | | | | |
| <mark>IPE European I-beams</mark> IPN European standard beams | | IPE 750 x 137 | 753.0 | 263.0 | 17459.74 | | | | |
| — UB British universal beams — UC British universal columns | | IPE 750 x 147 IPE 750 x 173 | 753.0 762.0 770.0 | 265.0 267.0 | 18750.04 22134.36 | ~ | | | |
| ilter: Europe | | No filtering | | | | | | : | |
| | l s | napes / IPE European | l-beams | | | | | | |

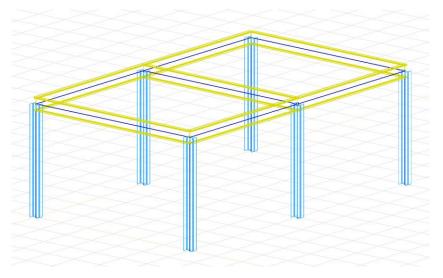
Close window with **OK**, then the selected cross-section is now displayed on the panel:

| | × |
|---------------------|-----------------|
| 🛾 🛏 🖉 🗖 | Se 🛞 🖉 🗖 |
| Туре | Beam |
| Material | S 235 |
| Allow variable cro | |
| Cross-section | IPE 400 🔻 🗉 🎞 😕 |
| Local x orientation | Forward |
| Local z reference | × Auto |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| | |
| \square | Õ |

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:



Change to Beam function on Draw objects directly panel.

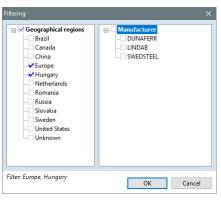


Cross-section



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 **0194.0** x **5.0** SV in **Cross-section** list:



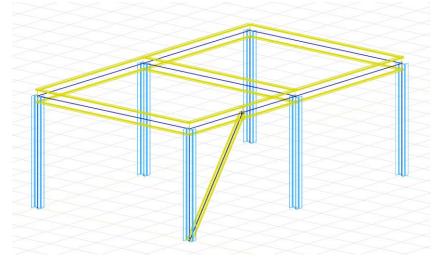
| 24 Cross-Section Library import | | | | | | | × |
|---|--|--|---|---|--|---|---|
| Library Parametric shapes | O Hungarian pipes | | | | Parameter | s | |
| Thin Thick | Name | Height [mm] | Width [mm] | Ax [mn | h (mm) b (mm) tw (mm) | 194.0 194.0 5.0 | ^ |
| − I II I I O O D □ II II L II T L J + + × ∧ ⊂ ¬I ··· Pipes O Dunaújvárosi Hungarian pipes O Hungarian pipes | III 0 133.0 X 18.0 0 133.0 X 20.0 0 133.0 X 20.0 0 159.0 x 4.0 SV 0 159.0 x 4.5 SV 0 159.0 x 4.5 SV 0 159.0 x 4.5 SV 0 194.0 x 4.0 SV 0 194.0 x 4.0 SV 0 194.0 x 4.0 SV 0 194.0 x 5.0 SV 0 194.0 x 4.0 SV 0 194.0 x 5.0 SV 0 219.0 x 4.5 SV 0 219.0 x 5.0 SV 0 219.0 x 5.0 SV 0 219.0 x 5.0 SV | 133.0 133.0 159.0 159.0 194.0 194.0 194.0 219.0 219.0 219.0 219.0 219.0 | 133.0 133.0 159.0 159.0 194.0 194.0 194.0 219.0 219.0 219.0 219.0 | 6 ^ 7 1 2 2 2 2 2 2 2 3 3 3 4 | tw [mm] tf [mm] Ax [mm ²] Ay [mm ²] Az [mm ²] kx [mm ⁴] iz [mm ⁴] iω [mm ⁶] | 5.0 2968.20 1485.70 1485.84 2.652518E7 1.325998E7 1.325998E7 0 | * |
| Filter: Hungary | 0 219.0 x 0.0 SV 0 245.0 x 4.0 SV 0 245.0 x 4.5 SV < No filtering Pipes / 0 Hungarian pipes | 245.0 245.0 | 219.0 245.0 245.0 | 4 3 3 | (| \mathcal{O} | |
| | O 194.0 x 5.0 SV | | | | ОК | Cancel | |

Close window with OK.

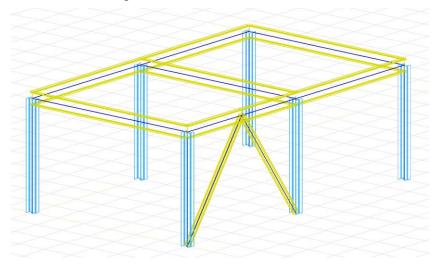


لے

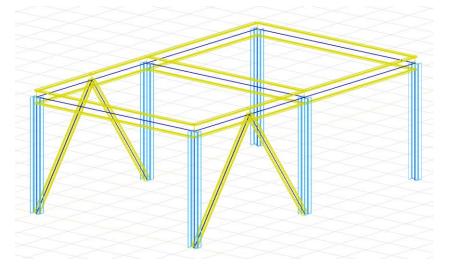
Draw polygon from the bottom node of **A1** to the centre of beam in **Y** direction as shown below:



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



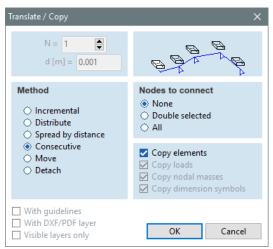
Press *Esc* to close polyline function, then repeat previous steps to specify the bracing elements on the other side of the structure.



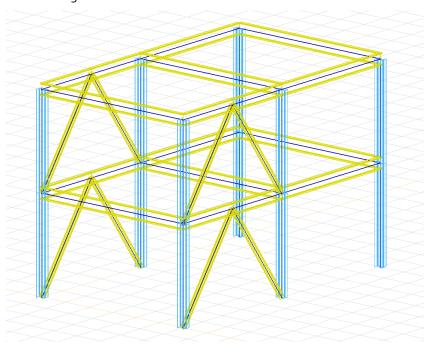
Translate / Copy

ΔĂ

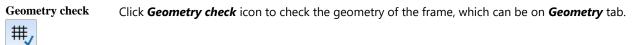
Now, all the structure will be copied in *Z* direction to define the upper storey. Activate *Translate* icon, select all the elements and in *Translate* window set the next:



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 Elements Loads Mesh | Static Buckling Vibration Dynam | nic R. C. Design Steel design Timber design | |
|------------------------------|---------------------------------|---|-------|
| | ☺◯ ⊬津鬪躑₿ | ▛ ▛ ✓ ☓ 🏷 –ᠠ/ ٭ –≯ ≍ & & & ◄ ๔ | ∠ ♥ ⊞ |

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.

| Geometry check X |
|---|
| Geometry check |
| Tolerance [m] = 0.001 |
| Selection of problematic nodes Selection of intersecting elements Select unattached nodes or lines List deleted nodes |
| Check domain contours Tolerance [m] = 0.001 |
| Check all loads Rebuilding of loads may cause loss of results. |
| OK Cancel |

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

| 6 | Geometry check | |
|---|---|------------|
| | Nodes deleted: 0 Lines deleted: 0 Surfaces deleted: 0 | |
| | | <u>0</u> K |

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

Zoom to fit

]€, €, ⊠ ↔ Ӌ €, €

Rendered view

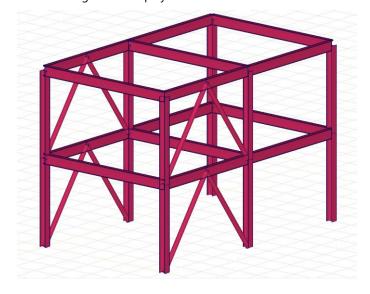


23

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



The following will be displayed:



Step by step tutorial Display options Activate Display options and uncheck Object contours in 3D on Symbols tab. ⇔ Symbols Labels Switches Graphics symbols Local systems Beam Node Trusses Rib Virtual beam 🔽 Beams Virtual beams Surface Ribs 🗌 Domain Center of circle Support 🔽 Domain Spring Surface center 🗌 Gap Mesh Link Edge hinge 🗹 Nodal support Load panel Line support Surface support ☑ Loads Display footings Concentrated Dimension lines Line Springs Surface 🛃 Gap elements Temperature Links Fire Self weight Other Rigids Diaphragm Reference 🗹 Load panel Cross-section shape Abutting wall or parapet (for snow End releases loads) / Edges with return corner Structural members (for wind loads) Reinforcement param. Load distribution scheme Reinforcement domain Derived beam load Mass Moving load phases Thickness reference points Transparent load diagrams Object contours in 3D Auto refresh Refresh all ОК Cancel Save as default

Wireframe view ß

C C C **C** 🔒

Change back to Wireframe view:

| Elements | | То | create | nodal s | supports | change | tab to El | ements: | | |
|----------|----------|-------|-----------------------|---------|----------|-----------|------------------|----------------|--------------|-----------------|
| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design |
| | * 1 | | d (| đđ | 12 🛙 | { 🗠 🚥 | • 🐡 🌈 | F F ⊢ , | x / x | 11 전 전 것 전 전 18 |

Nodal support

슶

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:

| Nodal supports | | | | | | × |
|---|---------------------|---|-------------------|--------|--|----|
| Oefine | O Modify | | | | | |
| Direction Global Referential Beam/Rib r Edge relative | relative | | | Z X | RZZ RZ RZ RY RX RX RXX | |
| Referenc | e | ~ | 🗌 Nonlinear paran | neters | | |
| R _X [kN/m] |] = 1E+10 ~ | | | | | |
| R _Y [kN/m] |] = 1E+10 ~ | | | | | |
| R _Z [kN/m] |] = 1E+10 ~ | | | | | |
| R _{XX} [kNm/rad] |] = 1E+10 ~ | | | | | |
| R _{YY} [kNm/rad] |] = 1E+10 ~ | | | | | |
| R _{ZZ} [kNm/rad] |] = 1E+10 ~ | | | | | |
| Pick up » | <u>C</u> alculation | | | | OK Cano | el |

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

| $R_{\chi} [kN/m] =$ | 1E+10 ~ |
|-----------------------------|---------|
| $R_{\gamma} [kN/m] =$ | 1E+10 ~ |
| $R_{Z} [kN/m] =$ | 1E+10 ~ |
| R _{XX} [kNm/rad] = | 0 ~ |
| R _{YY} [kNm/rad] = | 0 ~ |
| R _{ZZ} [kNm/rad] = | 이~ |

then close window with **OK**.

Loads

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

| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | |
|----------|----------|-------------|------|--------|----------|-----------|---------|--------------|--------------|---------------|--------|---|
| 표 • 끝 | 🗄 🛧 ረ | <u>∔</u> ∠± | ₫ 4 | 1 25 1 | • 2 | 🛛 📎 👌 | ¥ 3 | 🖕 G 📇 | • Е . 🌡 от 🕻 | 📕 🕰 🏭 🛀 🖷 | 土に主用する | m |

Load cases and load groups

Щ. 🗸

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

| 🞽 Load groups and load cases | | - | | × |
|------------------------------|---|---------------|--------|-----------------------------|
| ⊡ Ungrouped ⊥ ST1 (-) | ST1 Conversion Load group Interview | idd ₩ ₩ | ₩ - | |
| | Eurocode | | | |
| | Load group | | Add |)) 2 } |
| × Delete | Critical load group combinations OK | | Ca | ncel |

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:

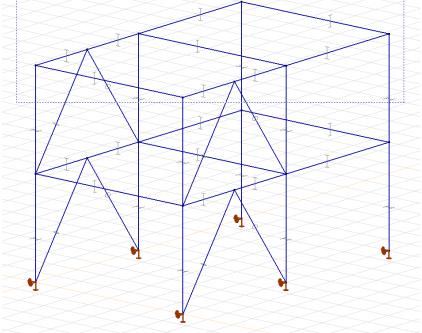


Load along line elements

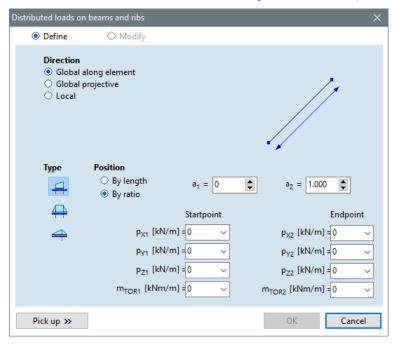
أآلته

beams with selection rectangle.

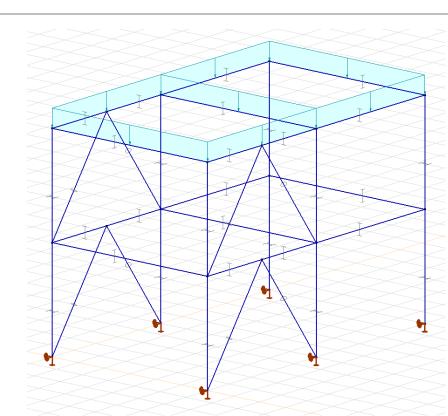




Confirm selection with **OK**, then in the following window set load parameters.



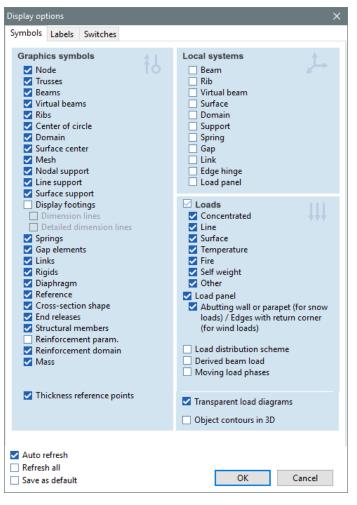
Set **p**_{Z1} and **p**_{Z2} to **-25**, then finish with **OK**, then the load is indicated on the beam elements in cyan:



Display options



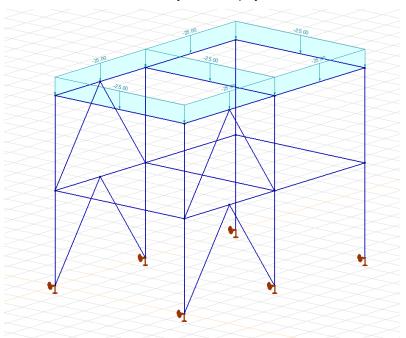
If the load intensity is not labelled automatically (this function is not active), then open **Display op-***tions*:



| Display options | × |
|--|--|
| Symbols Labels Switches | |
| Numbering Node Truss Beam Rib Virtual beam Surface Domain Support Links Rigid Diaphragm Spring Gap Material Cross-section Design member Design optimization group Load panel Reference Use finite element numbers Story center of gravity Story shear center Labels on lines seen from axis direction Transparent labels Prevent labels from overlapping | Properties ABC Nodal coordinates Material name Cross-section name Bolted joint Column reinforcement Beam reinforcement Beam reinforcement Beam reinforcement Beam reinforcement Column reinforcement Beam reinforcement Beam reinforcement Domain area Stiffness reduction COBIAX labels Load value Concentrated Line Surface Temperature Fire Self weight Other Mass value Units Image: Surface Actual reinforcement Symbols Labels axb axb axb yt ayt ayt ayt Rebars + Reinforcement values Rebars + Quantity x (Length) According to the displayed result component |
| Auto refresh Refresh all Save as default | OK Cancel |

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

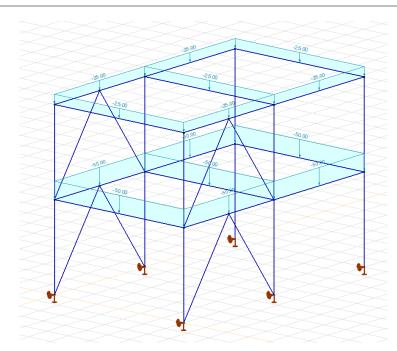
Close with **OK**, then load intensity will be displayed:



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

Static load case

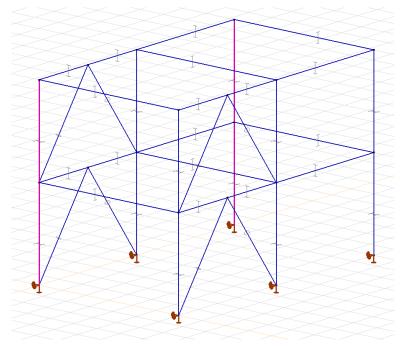
Load along line elements

<u>aani</u>

Click on *Load cases and load groups* icon.

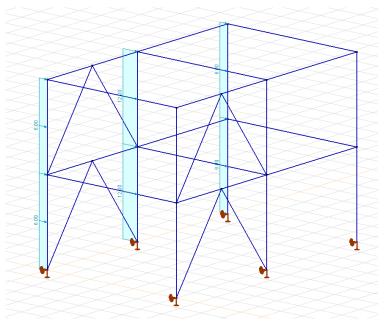
Add a new load case to the list by clicking on *Static* icon in *New case* top icon row, and rename it to *WIND*. After confirming with *OK*, all loads predefined will 'disappear' because the active load case will be the newly defined one, as it is shown on *Info palette*.

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:

| <u>File Edit Format Report Help</u> | | | | | | |
|--|-----------------|-------------|------|---------|------|------|
| Haterials (1) A Refar State grades (1) Coss=sections (2) LAM timber panels References Nodes (24) Bements Loads WIND (6) Load case (2) Critical load group combinations (1) Custom load combinations [b] Contact case | + X Ra C II - C | LIVE LOAD 1 | WIND | Comment | | |
| By load groups Functions | | | | | | |
| Weight report LIBRARIES | | | | | | |
| Editing | | | | ОК | Cano | el : |



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></tab> |
|-------------|------|-------------|
| WIND | 1.50 | <tab></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.

43

| Static | | Clie | ck on S | Static t | ab for ru | nning aı | nalysis: | | | | |
|----------|----------|-------|----------------|-----------------|-----------|-----------|----------|--------------|--------------|---------------|--|
| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | |
| °L,, °L | | | | | | • | | - | | ~ | |

Linear static



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

| 🚻 (x64) Linear analysis of frame.axs | - | - 🗆 | Х |
|--|----------|-------------|----|
| Analysis of frame.axs completed. | | | |
| | | | |
| | | | |
| Close if completed | | ОК | _ |
| | | UK | |
| Messages | | | ~ |
| Statistics | | | × |
| | | | _ |
| Number of equations | | Truss | - |
| Equations memory | | | 38 |
| Estimated memory requirement | | Rib | - |
| Solver block size | | Spring | - |
| Largest available memory block: | | Gap | - |
| · · · · · · · · · · · · · · · · · · · | 1 M | Link | - |
| Available physical memory: | | Edge hinge | - |
| Total physical memory: CPU | | Membrane | - |
| Single thread | 3192 MHz | Plate | - |
| | 00:00 | Shell | - |
| Model optimization | 00:00 | Diaphragm | - |
| Model verification | 00:00 | Load case | 2 |
| Analysis | 00:00 | Combination | 1 |
| Result file size: 80 k. | 00:00 | | |

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.

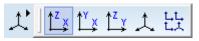


Change results view from *Isosurfaces 2D* to *Diagram*:

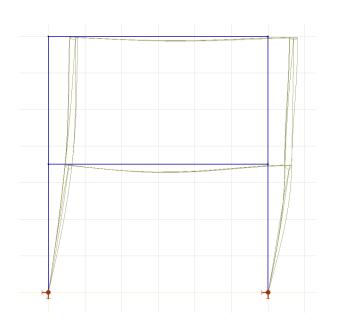
Diagram Diagram Section line Filled section line diagram Isolines Isosurfaces 2D None



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



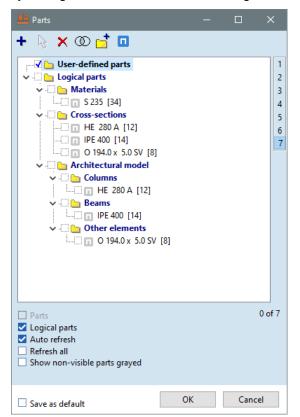
The following will be displayed:





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

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

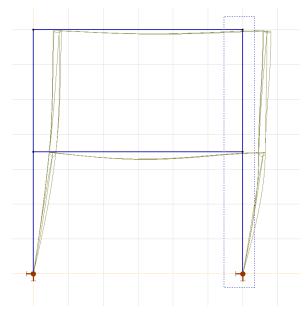




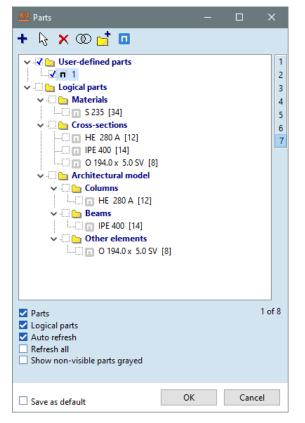
Click on *New* icon and the following window enter 1 to the name data field, then close with *OK*.

| • | New part definition | | | | | | | | | | | |
|---|--|--|--|--|--|--|--|--|--|--|--|--|
| | Part name | | | | | | | | | | | |
| | Select from the entire model Turn off all the other parts Activate this part on all drawings in the Drawings Library | | | | | | | | | | | |
| | <u>Q</u> K <u>C</u> ancel | | | | | | | | | | | |

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.



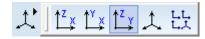
Finish selection with **OK**, then the defined part is added to the list in the window:



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



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



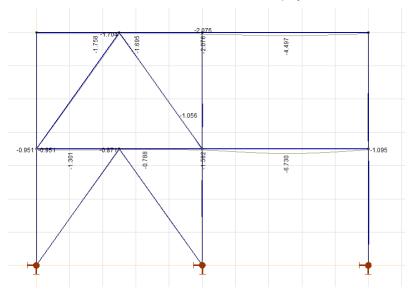
Result display parameters



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

| Display parameters | | × |
|--|---|---------------|
| Linear analysis Nonlinear analysis Dynamic analysis Case Envelope Critical | | |
| Load case (*) Co.#1 (ULS) v | Component eZ (mm) Scale by 1 Display mode Display mode Display shape Undeformed Write values to Vite values to Vite values to Vite values to Vite values to Min./Max. only Miscellaneous settings C Lut moment peaks over columns | Section lines |
| | Refresh all | OK Cancel |

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

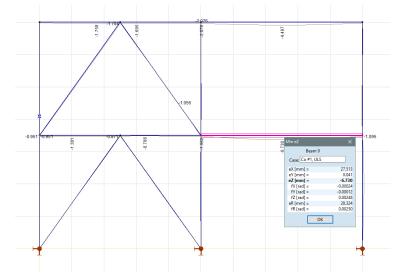






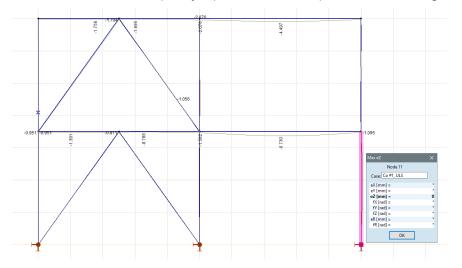
To find location of maximum deformation use *Min, Max values* function. Click on icon and select one of the deformation components in the following window:

| Model | extremes | | × |
|--------|----------|----------|---|
| Displa | cements | | |
| | eX [mm] | fX [rad] | |
| | eY [mm] | fY [rad] | |
| | eZ [mm] | fZ [rad] | |
| | eR [mm] | fR [rad] | |
| | ОК | Cance | I |

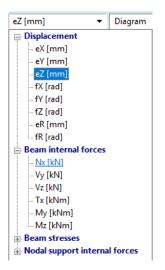


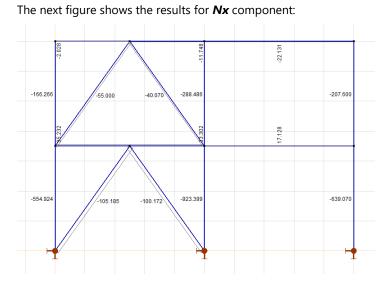
Confirm with **OK**, the software shows the maximum negative value and its location as well:

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

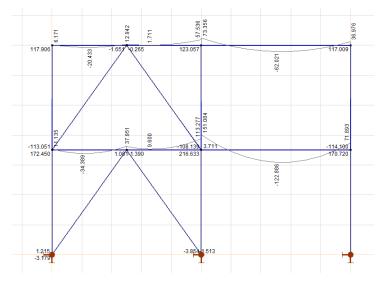


Select **Beam internal forces** – **Nx** [kN] component from the drop-down list among result components.

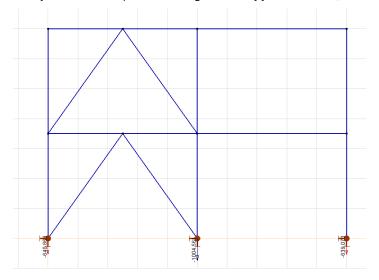




Change result component to **My** to see the bending moment diagram:



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



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

 Geometry
 Elements
 Loads
 Mesh
 Static
 Buckling
 Vibration
 Dynamic
 R. C. Design
 Steel design
 Timber design

 Image: The Co.#1 (ULS)

 Nx [kN]
 Image: Steel design
 Image: Steel design</lit

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**:

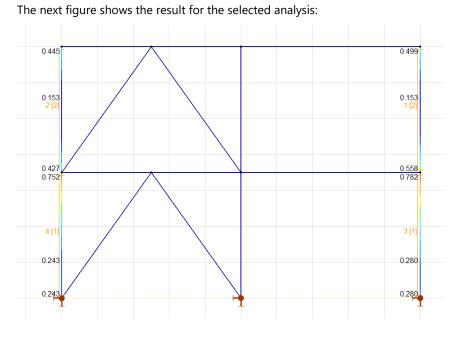
| Design parameters - Eurocode | × |
|--|--|
| Material S 235 Cross-section Original cross-section | 280 A |
| ULS (Ultimate Limit State) SLS (Serviceability Limit State) Design approach By section class (elastic / plastic) Section class (Automatic classification 1 0 2 0 3 0 4 | Buckling coefficients Image: Plexural buckling y Buckling factor z Buckling factor K ₂ = 1.25 Loss K ₂ = 1.000 |
| Design member ♥ Braced in local x-y plane Non-sway ♥ Braced in local x-z plane Non-sway Assemble design members ● ↓ ↓ ↓ ↓ | ✓ Lateral-torsional buckling Load position Top Center of gravity Bottom Custom ✓ Custom ✓ Estimated from kz, kw Fork supports to the nds User defined ✓ Web shear buckling |
| Coefficient for seismic forces $f_{se} = 1 $ | No stiffeners Transversal stiffeners OK Cancel |

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

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

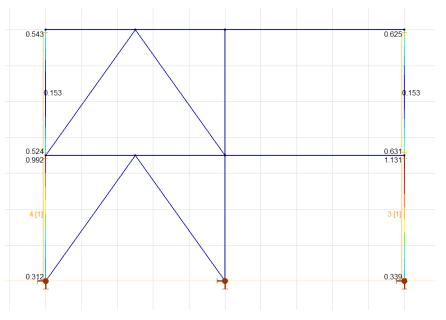
| kling | Vibration | Dynamic | R. C. Design | Steel design | Timber design |
|-------|-----------------|---------------|---------------|-----------------|-----------------|
| Nx [k | (N] | | | • | Isosurfaces 2D |
| E B | eam intern | al forces | | | |
| | Nx [kN] | | | | |
| | Vy [kN] | | | | |
| | Vz [kN] | | | | |
| | - Tx [kNm] | | | | |
| | - My [kNm] | | | | |
| | - Mz [kNm] | | | | |
| ⊞ B | eam stress | es | | | |
| ÷ C | ritical utiliz | ation | | | |
| 🛓 Li | mit states | | | | |
| | nalysis | | | | |
| | <u>N-M-V (A</u> | xial force-B | ending-Shear) | Ш | |
| | - N-M-Buck | kl (Axial For | ce-Bending-Fl | exural Buckling | a) [] |
| | - N-M-LTB | uckl (Axial f | orce-Bending- | Lateral torsion | al buckling) [] |
| | Vy (Shear(| y)) [] | | | |
| | Vz (Shear(| z)) [] | | | |
| | Vw-M-N (| Web shear- | Bending-Axial | force) [] | |
| ÷ R | esistances | | | | |
| | | | | | |

N-M-V Axial force-Bending-Shear



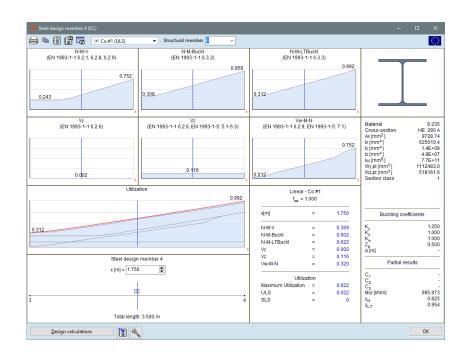
Utilization

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 | | | |
| $N_{Ed_{11}} = -554.92 \text{ kN } V_{y,Ed_{11}} = 1.69 \text{ kN } V_{z,Ed_{11}} = 50.18 \text{ kN } M_{y,Ed_{11}} = 17244.99 \text{ kNcm} = 172.450 \text{ kNm } M_{z,Ed_{11}} = 17244.99 \text{ kNcm} = 172.450 \text{ kNm} = 172.450 kN$ | = -471 | 66 kNcm | |
| | | oo ni tem | |
| $= -4.717$ kNm $M_{x,Ed_{11}} = -5.01$ kNcm $= -0.050$ kNm | | | |
| $\eta_{NMV_{ab}} = \eta_{MV} = 75.2$ % passed | | | |
| - courpl - cours | | | |
| 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 | | | |
| $C_{mv} = 1 \ge 0.4$ Table B.3 | | | |
| $C_{mz} = \max(0.2 + 0.8 \cdot \alpha_{mz}, 0.4) = \max(0.2 + 0.8 \cdot 0.371, 0.4) = 0.497 \ge 0.4$ Table B.3 | | | |
| $f_{yy} = \min(\lambda_y^* - 0.2; 0.8) = \min(0.39 - 0.2; 0.8) = 0.193$ | | | |
| $f_{zz} = \min(2 \cdot \lambda_z * -0.6; 1.4) = \min(2 \cdot 0.53 - 0.6; 1.4) = 0.465$ | | | |
| $k_{yy} = C_{my} \cdot \left(1 + f_{yy} \cdot \frac{ N_{Ed_{11}} }{\chi_y \cdot N_{pl,Rd}} \right) = 1 \cdot \left(1 + 0.193 \cdot \frac{ (-554.92) }{0.9289 \cdot 2286.25} \right) = 1.05$ | | | ~ |
| 🗷 Substitution 🛛 100% 🧹 🖨 🎦 | O | < | |

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**.

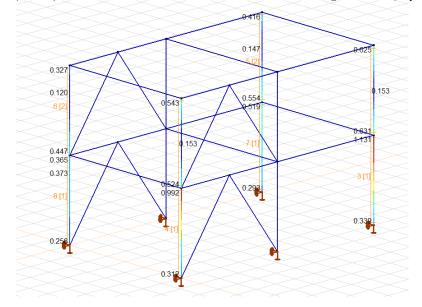
Change view to Perspective!



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

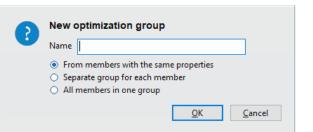
I

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:

| 2 Steel cross-section optimization | - | | × |
|--|---|-------|---|
| Design optimization groups Optimization | | | |
| + 🗟 × | | | |
| | | | |
| | | | |
| [Click to get result values or draw a frame to select. Use SHIFT to add elements to the selection. | | | |
| c | K | Cance | |



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.

| 💥 Steel cross-section optimization | -, | | – 🗆 X |
|---|--|--|---|
| Design optimization groups Optimization | on | | |
| + 1/3 × | Optimization from predefined shapes Parar | netric optimization | |
| [1] lower columns (HE 280 A) <4> | Objective of optimization Minimum weight Minimum height Minimum width Maximum utilization | Optimization checks ✓ Strength ✓ Flexural buckling ✓ Lateral torsional buckling ✓ Web buckling | Constraints 135.0 ≤ h [mm] ≤ 540.0 140.0 ≤ b [mm] ≤ 560.0 |
| | Candidates | 0 of 0 Model cross-sections | Library |
| | Model cross-sections Library | ↓ I shapes | |
| | | | OK Cancel |

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:

| | Objective of optimization | Optimization checks | Constraints |
|--|---|----------------------|--|
| | Minimum weight | Strength | 135.0 ≤ h [mm] ≤ 540.0 |
| | Minimum height | Flexural buckling | |
| | Minimum width | Lateral torsional bu | ıckling 140.0 ≤ b [mm] ≤ 560.0 |
| | Maximum utilization 0.850 | 🗹 Web buckling | - |
| | Candidates | 69 of 137 | |
| | Candidates | 09 01 137 | Model cross-sections Library |
| | Model cross-sections | <u>^</u> | BR_W_HP_AL |
| | - Library | | BR_W_HP_CX |
| | | × | BR_W_HP_PF |
| | - HE European wide flange bea | ims | . Chinese H shapes |
| | HE 100 A | | Chinese I-beams |
| | HE 100 AA | | HD wide flange columns |
| | HE 100 B | | 🖶 HE European wide flange beams <mark>s</mark> |
| | HE 100 M | | HL beams with very wide flanges |
| | HE 120 A | | HP Canadian I-beams (imperial) |
| | HE 120 AA | | HP Canadian I-beams (metric) |
| | HE 120 B | | HP wide flange bearing piles |
| | HE 120 M | | 🖶 I Hungarian I-beams |
| | HE 140 A | | 🗓 I Romanian I-beams |
| | HE 140 AA | | 🗄 I Russian I beams |
| | HE 140 B | | IPE European I-beams |
| | HE 140 M | | Here IPN European standard beams |
| | HE 160 A | | 🖶 M Canadian I-beams (imperial) |
| | HE 160 AA | | M Canadian I-beams (metric) |
| | HE 160 B | | 🗄 S Canadian I-beams (imperial) |
| | HE 160 M | | 🗊 S Canadian I-beams (metric) |
| | HE 180 A | ¥ | CID C |
| the second s | and a line CHET to a difference in the | - the | |
| ick to get result values or draw a frame | to select. Use SHIFT to add elements to the sel | ection. | |
| | | | OK Cancel |

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:

| 34 Steel cross-section optimization | | | - 🗆 X |
|--|---|--|--|
| Design optimization groups Optimizatio | n | | |
| + 🗟 🗙 | Optimization from predefined shapes P | arametric optimization | |
| [1] lower columns (HE 280 A) <4> [2] upper columns (HE 280 A) <4> | Objective of optimization Minimum weight Minimum height Minimum width Maximum utilization 0.850 | Optimization check ✓ Strength ✓ Flexural buckling ✓ Lateral torsional b ✓ Web buckling | 135.0 ≤ h [mm] ≤ 540.0 |
| | Candidates | 69 of 137 | Model cross-sections Library |
| | Library - Library - I shapes - HE European wide flange b | | BR_SOLDADOS_ESPECIAIS_400 |
| | HE 100 A HE 100 A HE 100 AA | Jeans | B-BR_W_HP_AL B-BR_W_HP_CX |
| | HE 100 M HE 120 A | | BR_W_HP_PF Chinese H shapes Chinese I-beams |
| | HE 120 AA HE 120 B HE 120 M | | HD wide flange columns HE European wide flange beams HE beams with very wide flanges |
| | HE 140 A HE 140 AA HE 140 B | | ↔ HP Canadian I-beams (imperial) ↔ HP Canadian I-beams (metric) ↔ HP wide flange bearing piles |
| | HE 140 B HE 140 M HE 160 A | | Her Wide range bearing piles Hungarian I-beams Homanian I-beams |
| | HE 160 AA HE 160 B HE 160 M | | I Russian I beams IPE European I-beams IPE Furopean standard beams |
| < >> | HE 100 M | v | HIN European standard beams |
| | | | OK Cancel |

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 crosssections 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.

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

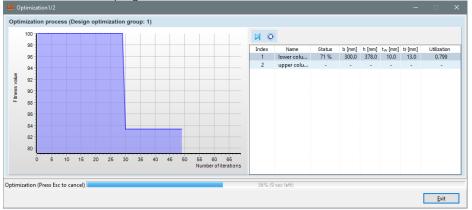
| <u>×4</u> | | | | | | | | | | | | | | | | | | | | | |
|-----------|------------------------|---------------------------|-----------------------------|--------------------|--------------|-------------|------------|-----------|-----------|--------------|-------------------------|------------|-----------|------|--------|----------|---------------|------------|--------|--------------|---------|
| Desi | n optimization grou | ps Optimization | | | | | | | | | | | | | | | | | | | |
| 8 | | (+) Co.#1 (ULS) | | <u>O</u> ptimiza | tion 🚺 | | | | | | | | | | | | | | | | |
| | Group | Original / optim shape | Optimization utilization | Allowed utilizat | Utilization | M [kg/m] | ΣM [kg] | ΔM [%] | b (mm) | h (mm) (r | t _w nm] [| ty mm] | Objective | Str. | Buckl. | LTBuckl. | Web buckl. | Error | Metho | d Op | . Repla |
| | lower columns | HE 280 A | 1.131 | 0.850 | 1.131 | 76.371 | 1069.189 | - | | 270.0 | 8.0 | 13.0 Weigl | ht | • | • | • | • | - | | 1 | |
| | upper columns | HE 280 A | 0.631 | 0.850 | 0.631 | 76.371 | 1069.189 | | | | | 13.0 Weigl | ht | • | • | • | • | - | | 1 | |
| | | - | - | - | - | - | - | - | - | - | - | - | | | | | | - | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | <i>C</i> , | D Repl | ace cross-se | ctions |
| (Clic) | to get result values o | r draw a frame to se | elect. Use SHIFT to | add elements to th | e selection. | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | 0 | К | Cancel |

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:

| gn o | optimization grou | ps Optimization | | | | | | | | | | | | | | | | | | | |
|------|-------------------|---------------------------|-----------------------------|------------------------------|-------------|-------------|-----------------|-----------|-----------|-----------|------------------------|-----------------------|-----------|------|--------|----------|---------------|-------|---------|----|---------|
| 6 | 🖨 🖩 🗊 (| (+) Co.#1 (ULS) | | Optimiza | tion 🜔 | | | | | | | | | | | | | | | | |
| | Group | Original / optim shape | Optimization utilization | Allowed utilizat | Utilization | M [kg/m] | ΣM [kg] | ΔM [%] | b (mm) | h (mm) | t _w [mm] | t ₄ mm] | Objective | Str. | Buckl. | LTBuckl. | Web buckl. | Error | Method | Of | ot. Rep |
| lo | ower columns | HE 280 A | 1.131 | 0.850 | 1.131 | 76.371 | 1069.189 | | 280.0 | 270.0 | 8.0 | 13.0 | Weight | • | • | • | • | - | | | , |
| | | HE 400 AA | 0.799 | - | 0.799 | 92.416 | 1293.818 | 21 | 1 300, | 378 | 9.5 | 13.0 | | | | | | - | Library | | |
| up | upper columns | HE 280 A | 0.631 | 0.850 | 0.631 | 76.371 | 1069.189 | | | 270.0 | 8.0 | | Weight | • | • | • | • | - | | | 1 |
| | | HE 260 A | 0.759 | - | 0.759 | 68.171 | 954.398 | | | | | | | | | | | - | Library | | |
| | | | | | | 00.171 | 934.398 | -11 | 1 260 | , 250 | , 7.5 | 12.5 | | | | | 1 | | Library | | |
| | | | | | | 05.171 | 3 34.398 | -11 | 1 260 | , 250 | . 7.5 | 12.5 | | | | | | | Livialy | | |

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.

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:

Replace cross-sections

| ign o | optimization grou | ups Optimization | | | | | | | | | | | | | | | | | | | |
|-------|-------------------|---------------------------|-----------------------------|------------------------------|-------------|-------------|------------|-----------|-----------|-----------|-------------|-----------|-----------|------|--------|----------|---------------|-------|---------|------|------|
| (| 🖨 🖩 🗊 (| (+) Co.#1 (ULS) | | Optimizz | tion 🜔 | | | | | | | | | | | | | | | | |
| | Group | Original / optim shape | Optimization utilization | Allowed utilizat | Utilization | M [kg/m] | ΣM [kg] | ΔM [%] | b [mm] | h [mm] | t,, [mm] | t, nm] | Objective | Str. | Buckl. | LTBuckl. | Web buckl. | Error | Method | Opt. | Repl |
| 1 Iov | ower columns | HE 280 A | 1.131 | 0.900 | 1.131 | 76.371 | 1069.189 | | 280.0 | | | | /eight | • | • | • | • | - | | 1 | |
| | | HE 400 AA | 0.799 | - | 0.799 | 92.416 | | 21 | 300, | | 9.5 | 13.0 | | | | | | - | Library | | |
| up | pper columns | HE 280 A | 0.631 | 0.900 | | 76.371 | 1069.189 | | 280.0 | | | 13.0 W | /eight | • | • | • | • | - | | 1 | |
| | | HE 260 A | 0.759 | - | 0.759 | | | | | | | | | | | | | | Library | | |
| | | | | | 0.735 | 68.171 | 954.398 | -11 | 260 | 250, | 7.5 | 12.5 | | | | | | - | Library | | |
| | | | | | 0.135 | 08.1/1 | 954.398 | -11 | 260 | 250 | 7.5 | 12.5 | | | | | | - | Lovery | | |

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

| ? | Replacing cross-sections Save the model under a new name before replacing cross-sections? | | | | | | | |
|---|---|-------------|------------|----------------|--|--|--|--|
| | | <u>Y</u> es | <u>N</u> o | <u>C</u> ancel | | | | |

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

In the following window click on Yes:

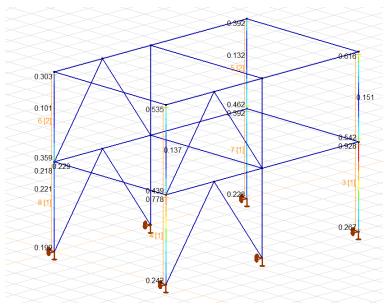
| | | | × | | | | | | |
|--|--------------|---------|----------------|--|--|--|--|--|--|
| You have modified the model. Results will be cleared. Save changes? | | | | | | | | | |
| Ye | s <u>N</u> o | Save as | <u>C</u> ancel | | | | | | |

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:

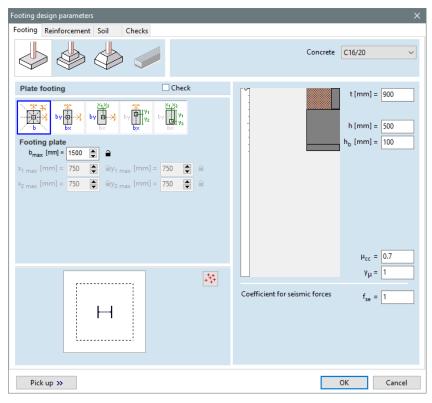
| Geometry Elements Loads Mesh Static | Buckling Vibration Dynamic | R. C. Design Steel design Timber design |
|-------------------------------------|----------------------------|---|
| () 💮 🔛 🛄 🧤 📰 (+) Co.#1 (ULS) | ▼ Nx [kN] | ▼ None 		 1 💽 ±min 📕 I⊡ 📮 TF 🕸 🖉 💯 |



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



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*:

| Footing design parameters | | | × |
|---|---------------------|--------------------------------|---------------------------|
| Footing Reinforcement Soil Check | 5 | | |
| | | Concrete | C30/37 ~ |
| Plate footing | Check | | t [mm] = 1700 |
| | ×1.×2 1 | | |
| | yz by the yz | 1 | h [mm] = 500 |
| Footing plate b _{max} [mm] = 1200 | | | h _b [mm] = 100 |
| x _{1 max} [mm] = 750 🐳 🎰y _{1 max} [mr | n] = 750 💽 📓 | 1 | |
| x _{2 max} [mm] = 750 | n] = 750 | | |
| | | | |
| | | | |
| | | | |
| | | | μ _{cc} = 0.7 |
| | 举 | | γμ = 1 |
| | | Coefficient for seismic forces | f _{se} = 1 |
| H | | | |
| | | | |
| | | | |
| | | | |
| Pick up » | | | OK Cancel |

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

| Footing design parameters | × |
|--|---|
| Footing Reinforcement Soil Checks | |
| Pooting Reinforcement Plate thickness: 500 mm Rebar steel $B500B$ \checkmark Concrete cover $c_T [mm] = 40 \checkmark (20 - 234) \qquad 0 [mm] = 16 \checkmark x y$ $\phi [mm] = 16 \checkmark x y$ $\phi [mm] = 16 \checkmark x y$ $c_B [mm] = 40 \checkmark (20 - 234) \qquad 0 [mm] = 16 \checkmark x y$ | Punching reinforcement d = 444 mm Rebar steel $B500B$ \checkmark Shear reinforcement $\emptyset_{sw} [mm] = 10 $ |
| Pick up >> | OK Cancel |



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

| Coarse | | dry or | | underwater | Fine | Void | consistence | | |
|---------------------------|-------|--------|------|------------|-------------|-------|-------------|------|------|
| | | damp | | | | ratio | stiff | firm | soft |
| Cobbles, gravel | Loose | ASL | ANL | AVL | | 0,4 | IK4 | IS4 | |
| | Solid | AST | ANT | AVT | Silt | 0,5 | IK5 | IS5 | IP5 |
| Mixed non-silty, | Loose | BSL | BNL | BVL | Silt | 0,7 | IK7 | IS7 | IP7 |
| sandy gravel | Solid | BST | BNT | BVT | | 1,0 | IK10 | IS10 | IP10 |
| Homogenous, | Loose | CSL | CNL | CVL | | 0,4 | JK4 | | |
| coarse and medium sand | Solid | CST | CNT | CVT | Lean clay | 0,5 | JK5 | JS5 | |
| | Loose | DSL | DNL | DVL | | 0,7 | JK7 | JS7 | JP7 |
| Mixed silty sand | Solid | DST | DNT | DVT | | 1,0 | JK10 | JS10 | JP10 |
| Homogenous fine | Loose | ESL | ENL | EVL | | 0,4 | KK4 | | |
| non-silty sand | Solid | EST | ENT | EVT | Medium clay | 0,5 | KK5 | KS5 | |
| | Loose | FSL | FNL | FVL | , | 0,7 | KK7 | KS7 | KP7 |
| Very fine sand | Solid | FST | FNT | FVT | | 1,0 | KK10 | KS10 | KP10 |
| | Loose | GSL | GNL | GVL | | 0,4 | LK4 | | |
| Very fine silty sand | Solid | GST | GNT | GVT | Fat clay | 0,5 | LK5 | LS5 | |
| | Jona | | 0.11 | 011 | , ac city | 0,7 | LK7 | LS7 | LP7 |
| | | | | | | 1,0 | LK10 | LS10 | LP10 |
| | | | | | | | | | |

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**.

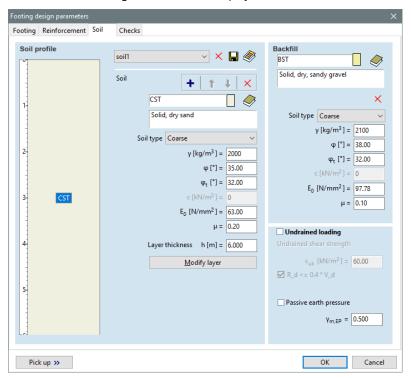
Enter 6,0 m in Layer thickness field.

Layer thickness

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



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:



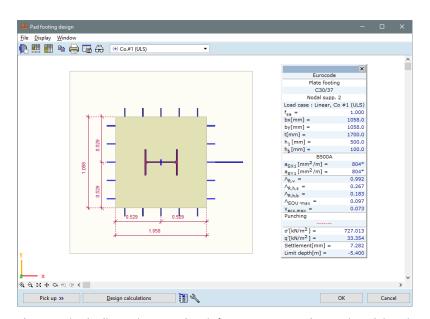
After click on **OK** and enter the **Name** of the specified soil profile:

| Save soil profile | × |
|---|--------|
| Name | |
| soil1 | \sim |
| Save a copy to the soil profile library | |
| | |
| OK Cancel | |

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**.

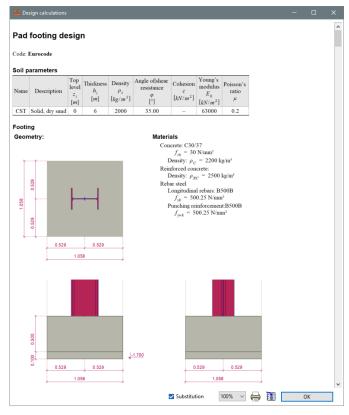
| Warning Size of the footplate is smaller than the critical punching line so punching cannot be checked. | |
|---|--|
| <u>K</u> | |



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.



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

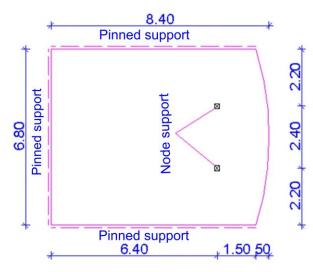
Objective

Start

New

ß

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

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

| New model | | | |
|--------------------------------|---|------------------|---|
| Select a view to start with | | <u>F</u> older | C:\Users\lstván T\Desktop\Step by Step\ |
| · [| <u>M</u> odel fi | le name: | Slab |
| Top view | | gn code | Eurocode |
| z H | <u>U</u> nits and <u>R</u> eport | | EU Change settings Eiglish |
| Front view | Page header | 1 | |
| Perspective | (Company logo) | Projec Analys | |
| | | Co | omment |
| | Project Analysis by Model: Slab.axs | | |
| | | | OK Cancel |

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.

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

Define of geometry -Elements

| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | | |
|----------|----------|-------|----------------|--------|----------|-----------|---------|--------------|--------------|---------------|---------|------|-------------|
| ØI | 🦸 🖌 | 0 | 0 (| í í | 12 🖿 | | • 🌧 🌈 | ■ (月) | Nº 1 1 | 표 🕹 🗍 | × 14 14 | j⊴ y | <u>'B</u> 🔒 |

directly

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





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:

| Warning No element material defined. Browse material library | |
|--|------------|
| | <u>O</u> K |

Material library import

| Click on OK t | o Browse | cross-section | libraries |
|----------------------|----------|---------------|-----------|
| | | CIUSS SCULUT | |

| 🔀 Material Library Import - Eu | rocode | | | | - | | \times |
|--|---|---|---|--|---|--------|----------|
| <u>D</u> esign code | <u>M</u> aterials | | C25/30 | | | | |
| CSA S6-06 [Rev. 2010 ^ DIN (German) Eurocode Eurocode [A] Eurocode [B] Eurocode [CZ] Eurocode [CI] Eurocode [FIN] Eurocode [H] Eurocode [NL] Eurocode [NL] Eurocode [SK] Eurocode [SK] Eurocode [VK] | S280GD+Z275 S320GD+Z275 S350GD+Z275 C12/15 C16/20 C20/25 C25/30 C30/37 C35/45 C40/50 C45/55 C50/60 C14 C16 C18 C20 | | $\frac{Type}{E [N/mm^2]}$ v $\alpha_T [1/°C]$ $\rho [kg/m^3]$ $f_{ck} [N/mm^2]$ γ_c α_{cc} ϕ_t | Concrete Isotropic 31500 0.20 1E-5 2500 25.00 1.500 1.00 2.00 | | | |
| | C22 C24 | | | | | ОК | |
| Image: NBCC 1995 ✓ < > | C27 C30 | ¥ | | | | Cancel | |

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**.

Step by step tutorial

| Comp | lex | slab |
|------|-----|------|
| 5 | | |

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:

| × | | dX[m] : | | | d r[m] | : | 9.062 |
|---|---|--------------------|--------|---|--------|---|-------------|
| | ы | dY[m] : dZ[m] : | -3.400 | ы | d a[°] | : | 337.96 0 |
| | a | dZ[m]: | 0 | a | dh[m] | : | 0 |
| | | dL[m] : | 9.062 | | | | |

Nodes in Relative coordinate system

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

| • | |
|--|---|
| X dX[m]: 7.9 dY[m]: 0 dZ[m]: 0 dL[m]: 0.400 | d r[m] : 0.400 d a[°] : 0 dh[m] : 0 |

X 7.9 **V** 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:

| Х | 0.5 | У | 3.4 | Ζ | 0 | <enter></enter> |
|---|-----|---|-----|---|---|-----------------|
| Х | 0 | y | 6.8 | Z | 0 | <enter></enter> |

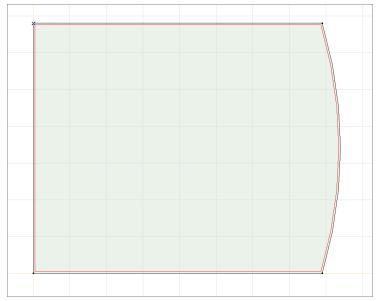


Click on *Line* icon on the auxiliary palette:

Press following keys to define next points:

X -7.9 V 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.

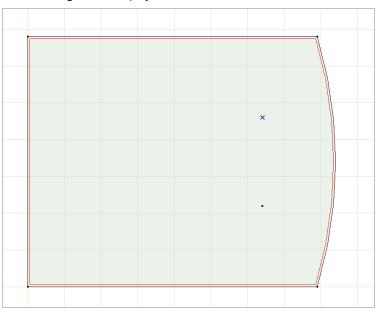
| Geometr | ry Click on <i>Geometry</i> tab: |
|----------|---|
| Geometry | Elements Loads Mesh Static Buckling Vibration Dynamic R. C. Design Steel design Timber design |
| • / | ୵୵ᆷᅌི॒Ӧ│ѷѺ│⊬ᅔ▦ᄤᅏᅏᄼᄿᆥᄿᆉᄽᄿᢀ◈》ፈᆂᄈᆱ |
| Node | Click on Node icon to create additional inner points and enter the following 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:

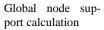




23

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

| Step by step tutorial | | 67 |
|-----------------------|---|----|
| | Change to Elements tab. Loads Mesh Static Buckling Vibration Dynamic R. C. Design Steel design Timber design | |
| Nodal support | Define nodal supports. Click on Nodal support icon, then select inner nodes and following window shows up: | |
| | Nodal supports X | |
| | Define Modify | |
| | Direction © Global © Referential © Beam/Rib relative © Edge relative | |
| | Reference Nonlinear parameters | |
| | $R_{X} [kN/m] = 1E+10 \ \lor$ $R_{Y} [kN/m] = 1E+10 \ \lor$ $R_{Z} [kN/m] = 1E+10 \ \lor$ $R_{XX} [kNm/rad] = 1E+10 \ \lor$ $R_{YY} [kNm/rad] = 1E+10 \ \lor$ $R_{ZZ} [kNm/rad] = 1E+10 \ \checkmark$ | |



Pick up »

¹P- Click on **Calculation...** button, then the following window shows up:

Calculation...

| Global node support calculation | | | × |
|---|--------|--|--|
| Column above Material C25/30 Cross-section L [m] = | | • Ø • 1 I | End releases X V main and the second secon |
| Column below Material C25/30 Cross-section L [m] = | | ∅ 1 1 1 | End releases X Y |
| R _X [kN/m] = R _Y [kN/m] = R ₇ [kN/m] = | 0 0 | R_{XX} [kNm/rad] = R_{YY} [kNm/rad] = R_{ZZ} [kNm/rad] = | 0 |
| | v | OK | Cancel |

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

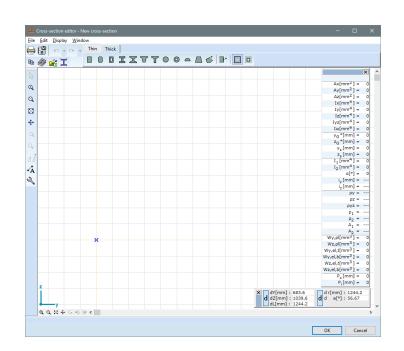
OK

Cancel

New cross-section

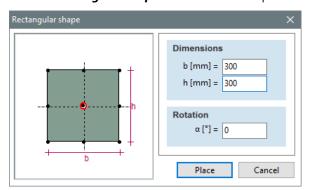
ction Click on *New cross-section* icon, the following window shows up:





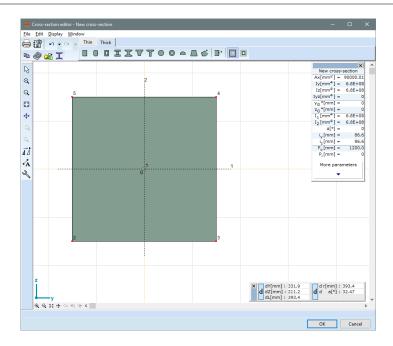
Rectangular shape

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:

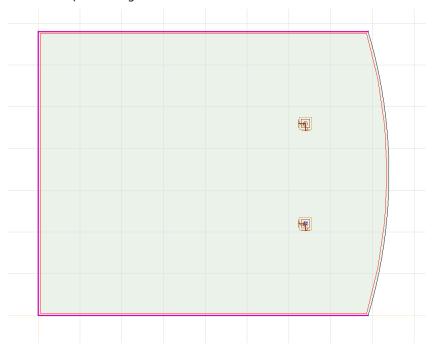


Click on **OK** and enter the **Name** of the new cross-section to save.

| 8 | New cross-section Enter cross-section name. |
|---|--|
| | Name 300x300 |
| | Remember cross-section position |
| | <u>O</u> K <u>C</u> ancel |

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.

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:



| e supports | | | | |
|---|------------|----------------------|-------|--------|
| Define | ⊃ Modify | | | |
| Direction Global Beam/Rib relative | ive | | Y T X | 1-1- |
| | | Nonlinear parameters | | |
| R _x [kN/m/m] | = 1E+7 ~ | | | |
| R _y [kN/m/m] | = 1E+7 ~ | | | |
| R _z [kN/m/m] | = 1E+7 ~ | | | |
| R _{xx} [kNm/rad/m] | = 1E+7 ~ | | | |
| R _{yy} [kNm/rad/m] | = 1E+7 ~ | | | |
| R _{zz} [kNm/rad/m] | = 1E+7 ~ | | | |
| Pick up » | alculation | | ОК | Cancel |

Calculation

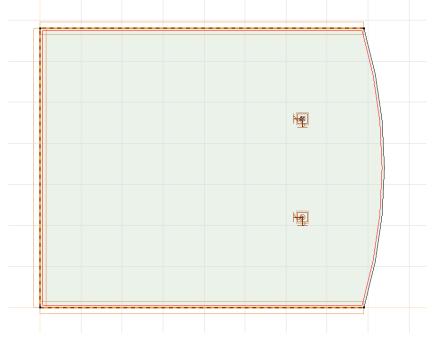
Local line support Click again on **Calculation..** button to calculate the stiffnesses automatically considering support conditions. In the next window set *L* [*m*] to 3 and *d* [*m*] to 300:

| Local line support calculatio | n | | × |
|-------------------------------|--|---------------------------|--------------|
| c L [r | /laterial C25/30 m] = 3.00 m] = 200 | | End releases |
| | 1aterial C25/30 m] = 3.00 m] = 300 | | End releases |
| R _x [kN/m/m] = | 2.33E+5 | R _{xx} [kNm/rad/ | m] = 2.8E+4 |
| R _y [kN/m/m] = | 9.33E+3 | R _{yy} [kNm/rad/ | [m] = 1E+0 |
| R _z [kN/m/m] = | 2.1E+6 | R _{zz} [kNm/rad/ | 'm] = 1E+0 |
| | | ОК | Cancel |

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

| Local line support calcu | ulation | | × |
|---------------------------|---|-------------------------------|-------------|
| U Wall above | Material C25/30 L [m] = 3.0 d [mm] = 200 | 00 | nd releases |
| Wall below | Material C25/30 L [m] = 3.0 d [mm] = 300 | v 🔊 | nd releases |
| R _x [kN/m/m] = | 3.5E+5 | R _{xx} [kNm/rad/m] = | 0 |
| R _y [kN/m/m] = | 0 | R _{yy} [kNm/rad/m] = | 1E+0 |
| R _z [kN/m/m] = | 3.15E+6 | R _{zz} [kNm/rad/m] = | : 1E+0 |
| | | ОК | Cancel |

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:

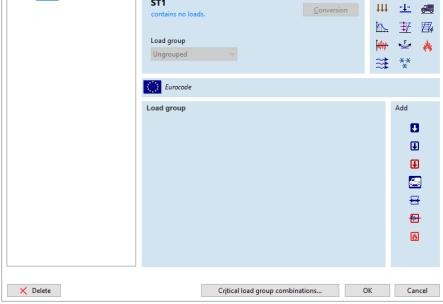




Next step is to define loads on the slab, click on *Loads* tab:

| Geometry Elements Load | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | |
|------------------------|-------|--------------|----------|------------|---------|--------------|--------------|---------------|-----------------|---|
| 프 - 블 수 쇼 스 | / 🛋 < |) 📠 🗸 | i 🖉 🕡 | 8 7 | ** 😤 🛙 | <u> </u> | + F + 🗖 🖟 | 👿 🚣 🛃 🛃 | 🗠 🛌 亜 🖽 🛥 💩 🐉 | 5 |



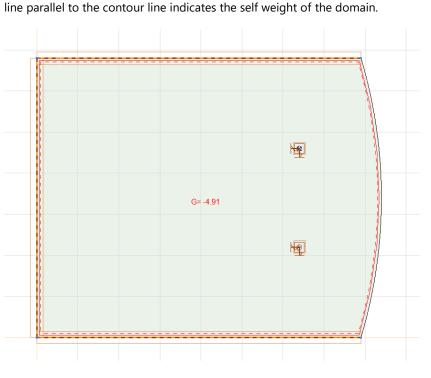


In the window that appears, click on the name ST1 in the top left corner and rename it to SELF-WEIGHT. (ST1 is an automatically generated load case, which should be renamed in our example.)

Click on Self weight icon then select all elements with All (*) button. By clicking on OK, a red dashed

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

| | × |
|------|---------------|
| Code | 🜔 Eurocode |
| Case | : SELF-WEIGHT |



Display options

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

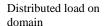
Self weight G

⇔

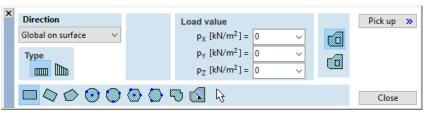




Open again *Load cases and load groups* window and create new *Static* load case with *Name FINISH-ES*, this load case will contain loads from slab finishes (in our example *2,5* kN/m²). Close with *OK*.



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.



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

| | Direction Global along element | | $\begin{array}{c} p_{X2} \ [kN/m] = \\ p_{Y2} \ [kN/m] = \\ p_{Z2} \ [kN/m] = \\ m_{tor2} \ [kNm/m] = \\ \end{array} \begin{array}{c} 0 \\ \checkmark \\ \end{array}$ | Pick up » |
|---|-----------------------------------|----------------------------------|---|-----------|
| / | ╱╱╘╔┓╲╲ | $\bigcirc \supset \checkmark \&$ | | Close |

Arc by three points

Load value

Set $p_z 1$ and $p_z 2$ load values at the endpoints to -1.

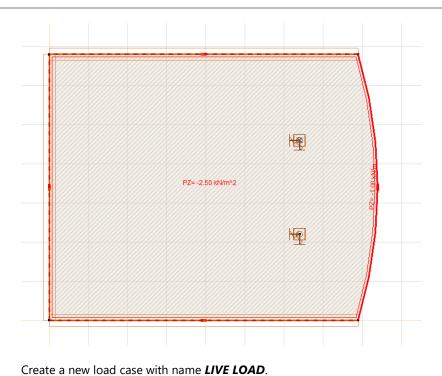
Select **Arc by three points** icon from bottom row, then click bottom, central and top points of the curved edge. Then again click on top point of edge, finally press **Esc** to exit.

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

Node
 Domain
 Support
 Material
 Cross-section
 Cross-section name
 Thickness
 Load value
 ✓ Line
 ✓ Surface
 ✓ Units

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

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



New Load case

111

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:

Specify live load on the slab in this load case. Define **Distributed load on domain** as shown previously.

| 24 Table Browser | | | | | | - | | × |
|--|---------------------------------|----------|-------------|----------|-----------|--------|-----|-----|
| <u>File Edit Format Report Help</u> | | | | | | | | |
| MODEL DATA Materials (1) | 🕂 🔸 🖻 💼 🔡 🖶 🖾 | 1 | | | | | | |
| Cross-sections (1) | Custom load combinations by loa | ad cases | | | | | | |
| - XLAM timber panels | Name | Туре | SELF-WEIGHT | FINISHES | LIVE LOAD | Commer | t | |
| - Nodes (6) | | | | | | | | _ |
| Elements | | | | | | | | |
| Loads SELF-WEIGHT (1) | | | | | | | | |
| FINISHES (2) | | | | | | | | |
| LIVE LOAD (1) | | | | | | | | |
| - Load cases (3) - Critical load group combinations (1) | | | | | | | | |
| Custom load combinations | | | | | | | | |
| - By load cases - By load groups | | | | | | | | |
| Functions | | | | | | | | |
| Editing | | | | | | | | |
| | | | | | | ОК | Car | cel |
| | | | | | | | | |

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></tab> |
|-------------|------|-------------|
| FINISIHES | 1.00 | <tab></tab> |
| LIVE LOAD | 0.30 | <tab></tab> |

Set load intensity to -3,0 kN/m².

New row



Enter these values into the appropriate cells.

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></tab> |
|-------------|------|-------------|
| FINISHES | 1.35 | <tab></tab> |
| LIVE LOAD | 1.50 | <tab></tab> |

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

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

| Step by step tu | torial | | | | | | | | | | 75 |
|-----------------|----------|-----------------|------------------|----------|----------------|-----------|---------|--------------|--------------|---------------|----|
| Mesh | S | elect Me | sh tab to | define (| domain me | sh: | | | | | |
| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | |
| \sum | | | ## | | $\sim 10^{-1}$ | × | | | | | |

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**).

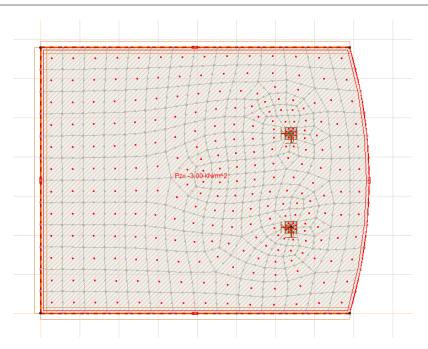
| Meshing parameters | × |
|---|---|
| 💿 Define 🔷 N | Nodify |
| Mesh type | |
| Average mesh | element size [m] = 0.65 🗸 |
| ∠ Line | int loads $ \ge [kN] = 0$ loads $ \ge [kN/m] = 0$ bads $ \ge [kN/m^2] = 0$ |
| Adjust mesh to colu cutting of moment processing of the second | |
| Contour division method Uniform mesh size Adaptive mesh size | I |
| Smoothing | |
| Create mesh only for unm Calculation of domain inte Keep generated mesh guid | ersections |
| | OK Cancel |

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

Progress bar shows the progress of meshing:

| Mesh generation | | |
|-----------------|-----|-------|
| | 19% | |
| | | |
| | | Abort |
| | | |

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



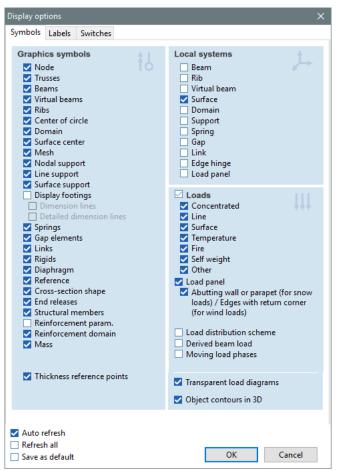
Moving the cursor over the centre of a finite elements (red dot), a hint shows up displaying the properties of the element:

| XIXIIX | 1•/X/•/X/ |
|--------------------------|-----------|
| [Plate 94] | |
| C25/30 | |
| 200 mm | |
| Domain 1 x Ref.: Auto | |
| z Ref.: Auto | |
| - WINTIN | |
| | 1/X//XX// |

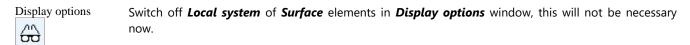


⇔

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.



Static

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

Click on Linear static analysis icon to run analysis.

| Geometry Elements Loads Mesh | Static Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | |
|------------------------------|-----------------|-----------|---------|--------------|---------------|---------------|---------------|---|
| | ſ | ▼ eZ | [mm] | - I | sosurfaces 2D | ~ 1 | ×max ▼ min | Ð |

Linear static

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:

| Ð | Nodal DOF |
|---|---|
| • | (Plate in X-Y plane) |
| | Set the nodal degrees of freedom automatically? Save model with these settings |
| | <u>Y</u> es <u>N</u> o |

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:

| PlateQ9 element equilibrated load evaluation | | Cancel |
|--|------------|-----------|
| 82% Close if completed 9:52:56 Winkler rib support element stress evaluation 9:52:56 Winkler rib support element equilibrated load evaluation 9:52:56 PlateQ9 element stress evaluation 9:52:56 PlateQ9 element stress evaluation 9:52:56 PlateQ9 element stress evaluation 9:52:56 PlateQ9 element equilibrated load evaluation 9:52:56 PlateQ9 element equilibrated load evaluation Statistics Number of equations Solver block size Solver block size Largest available memory block Analysis block size 3 M | | Cancel |
| Close if completed Close if completed 9:52:56 Winkler rib support element stress evaluation 9:52:56 Winkler rib support element equilibrated load evaluation 9:52:56 PlateQ9 element strain evaluation 9:52:56 PlateQ9 element stress evaluation 9:52:56 PlateQ9 element equilibrated load evaluation 9:52:56 PlateQ9 element equilibrated load evaluation Statistics Number of equations 3375 Equations memory 2.35 M Estimated memory requirement 8.51 M Solver block size 3.35 M Largest available memory block 9.107 G Analysis block size 3 M | | Cancel |
| Messages 9:52:56 Winkler rib support element stress evaluation 9:52:56 Winkler rib support element equilibrated load evaluation 9:52:56 PlateQ9 element strain evaluation 9:52:56 PlateQ9 element strain evaluation 9:52:56 PlateQ9 element stress evaluation 9:52:56 PlateQ9 element equilibrated load evaluation 9:52:56 PlateQ9 element equilibrated load evaluation 9:52:56 PlateQ9 element stress evaluation 9:52:57 Equations memory 2.35 M Estimated memory toblock size 2.35 M Largest available m | | Cancel |
| 9:52:56 Winkler rib support element stress evaluation 9:52:56 Winkler rib support element equilibrated load evaluation 9:52:56 Vinkler rib support element equilibrated load assembly 9:52:56 PlateQ9 element strain evaluation 9:52:56 PlateQ9 element stress evaluation 9:52:56 PlateQ9 element equilibrated load evaluation 9:52:56 PlateQ9 element equilibrated load evaluation Statistics | M A Truss | |
| 9:52:56 Winkler rib support element equilibrated load evaluation 9:52:56 Winkler rib support element equilibrated load assembly 9:52:56 PlateQ9 element strain evaluation 9:52:56 PlateQ9 element equilibrated load evaluation 9:52:56 PlateQ9 element equilibrated load evaluation Statistics Number of equations 3375 Equations memory 2:35 M Estimated memory requirement 8:51 M Solver block size 2:35 M Largest available memory block 9:107 G Analysis block size 3 M | M A Truss | |
| 9:52:56 PlateQ9 element equilibrated load evaluation Statistics Number of equations 3375 Equations memory 2.35 M Estimated memory requirement 8.51 M Solver block size 2.35 M Largest available memory block 9.107 G Analysis block size 3 M | Truss | |
| Number of equations 3375 Equations memory 2.35 M Estimated memory requirement 8.51 M Solver block size 2.35 M Largest available memory block 9.107 G Analysis block size 3 M | Truss | |
| Equations memory 2.35 M Estimated memory requirement 8.51 M Solver block size 2.35 M Largest available memory block: 9.107 G Analysis block size 3 M | | |
| Equations memory 2.35 M Estimated memory requirement 8.51 M Solver block size 2.35 M Largest available memory block: 9.107 G Analysis block size 3 M | V A Truss | |
| Estimated memory requirement 8.51 M Solver block size 2.35 M Largest available memory block: 9.107 G Analysis block size 3 M | Ream | |
| Solver block size 2.35 M Largest available memory block: 9.107 G Analysis block size 3 M | Rib | |
| Analysis block size 3 M | Spring | |
| | Gap | |
| | Link | |
| Available physical memory: 10.71 G | Edge h | , ingo |
| Total physical memory: 15.94 G | | |
| CPU Intel(R) Core(TM) i5-6500 CPU @ 3.20GHz Single thread 3192 MHz | (4x) Plate | 26 |
| Single thread 3192 MHz | Shell | 20 |
| Model optimization 00 | 00 Diaphr | |
| Model verification 00: | 00 Load c | |
| Analysis 00: | | ination |
| Result file generation | Combi | ination i |

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.

| 🚧 (x64) Linear analysis of Slab.axs | - 🗆 | × |
|--|------------|------|
| Analysis of Slab.axs completed. | | |
| | | |
| | | |
| Close if completed | Ok | (|
| | | |
| Messages | | × |
| 9:52:56 Calculating smoothed surface force combinations | | ^ |
| 9:52:56 Saving smoothed surface force combinations | | |
| 9:52:56 Calculating surface support force combinations | | |
| 9:52:56 Calculating smoothed surface support force combinations | | |
| 9:52:56 Calculating support element internal force combinations 9:52:56 Analysis of Slab.axs completed. | | |
| | | • |
| Statistics | | * |
| Number of equations 3375 | Truss | _ |
| Equations memory 2.35 M | Beam | - |
| Estimated memory requirement 8.51 M | Rib | - |
| Solver block size 2.35 M | Spring | - |
| Largest available memory block: 9.107 G | Gap | - |
| Analysis block size 3 M | Link | - |
| Available physical memory: 10.71 G | Edge hinge | - |
| Total physical memory: 15.94 G CPU Intel(R) Core(TM) i5-6500 CPU @ 3.20GHz (4x) | Membrane | |
| Single thread 3192 MHz | Plate | 268 |
| | Shell | - |
| Model optimization 00:00 | Diaphragm | - |
| Model verification 00:00 | Load case | 3 |
| Analysis 00:00 | Combinati | on 2 |
| Result file size: 1.27 M. 00:00 | | |
| | | |

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):

| +++ SELF-WEIGHT | • | eZ [ı |
|--------------------------------|---|-------|
| 🖃 🕒 Linear analysis | | |
| | | |
| | | |
| | | |
| (+) Co.#1 (SLS Quasipermanent) | | |
| (+) Co.#2 (ULS) | | |
| Envelopes | | |
| | | |

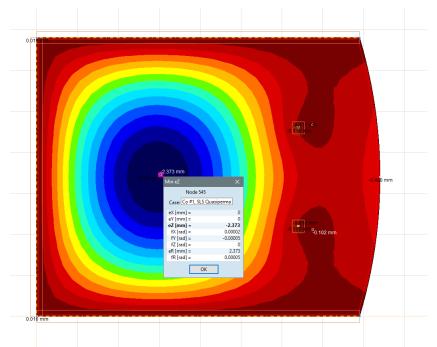
Deformation values are negative because the positive direction of global Z axis is opposite to the direction of the specified loads.

Min, max values

| 👱 max | |
|-------|--|
| 🖛 min | |

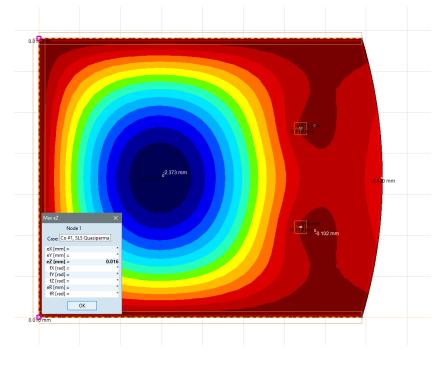
To find location of maximum deformation, use *Min, max values* function. By clicking on icon, shows up following window:

| Model extremes X | | | | | | |
|------------------|----------------|----------|--|--|--|--|
| Displacements | | | | | | |
| | | | | | | |
| | eX [mm] | fX [rad] | | | | |
| | eY [mm] | fY [rad] | | | | |
| | eZ [mm] | fZ [rad] | | | | |
| | eR [mm] | fR [rad] | | | | |
| | | | | | | |
| OK Cancel | | | | | | |



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

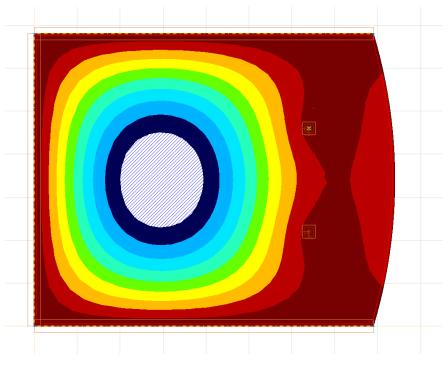
The **Color legend** shows boundary values of each color. Adjust number of boundary values by dragging the bottom edge of the palette:

| eZ [mm] | |
|------------|------------|
| 0.017 | |
| -0.154 | |
| -0.325 | > |
| -0.496 | eZ [mm] |
| -0.666 | |
| -0.837 | 0.017 |
| -1.008 | -0.249 |
| -1.178 | -0.515 |
| -1.349 | -0.780 |
| -1.520 | -1.046 |
| -1.690 | -1.311 |
| -1.861 | -1.577 |
| -2.032 | -1.842 |
| -2.203 | -2.108 |
| -2.373 | -2.373 |
| V/ | 1/2 |
| 15 | 10 |

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**.

| eZ Color legend setup | | × |
|--|--|---|
| 0.017 | Values Colors | |
| -0.249 -0.515 -0.780 -1.046 -1.311 -1.577 -1.842 -2.108 | Levels 10 Limits Round calculated values Min, Max of model -2.373 0.017 Min, Max of parts -2.373 0.017 Absolute max. of model -2.373 2.373 Absolute max. of parts -2.373 2.373 | |
| -2.1 | Custom Auto interpolate By step value △ = 1 ∑ave as Hatching for out of range values Opaque Transparent Color gradient direction Normal Reverse Isosurface contours Isoline labels | - |
| | Display Auto refresh Refresh all | |
| | OK Cancel | |

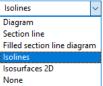
Press *Enter*, then *Auto interpolate* function sets the other boundary values. Close window with *OK*, in the main window the following will be displayed:



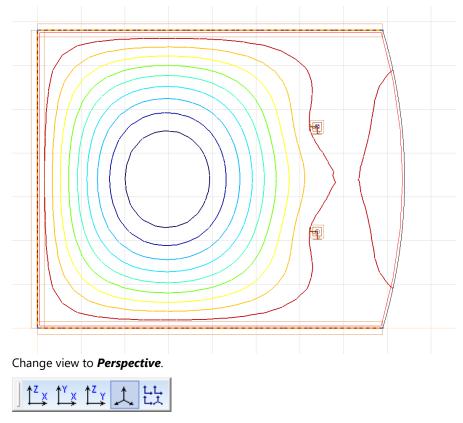
Area with larger than 2.1 mm is indicated by the hatched area in blue.

Isolines

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



The resulting figure is the next:





Change *Perspective settings* as shown below:

| H 30.0 V 320.0 P 0.0 V | |
|------------------------|-----|
| Observation distance | \$~ |

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

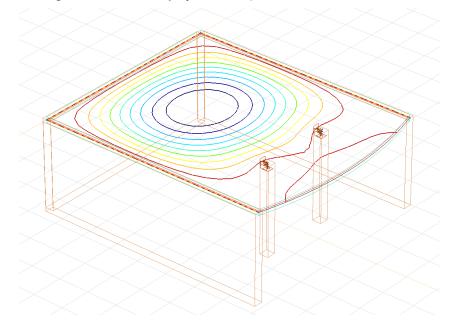
Result display

×

parameters

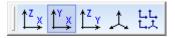


To see the deformed shape, click on **Result display parameters** icon. Select **Deformed Display shape** in the window and confirm with **OK**. Now, the deformed shape is displayed, if necessary change scale of the diagram (next to the **Display mode** drop down list).



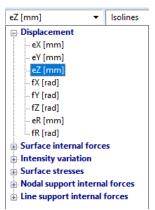


Change view to **X-Y Top view**!



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.

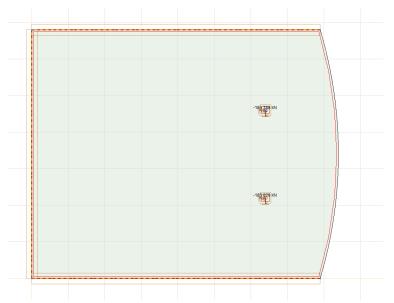




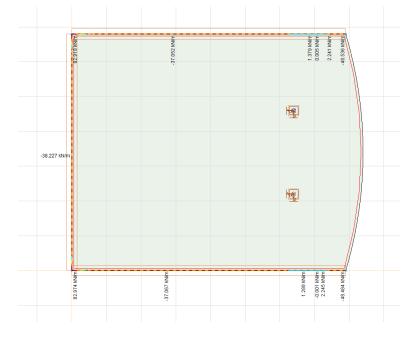
Activate **Results display parameters** icon:

| Display parameters | | > |
|--|---|---------------|
| Linear analysis Nonlinear analysis Dynamic analysis Case Envelope Critical | | |
| Load case (↔ Co.#2 (ULS) v | Component R2 [kN] Scale by 1 Display mode Display mode Display shape Undeformed Write values to Nodes Lines Surfaces Min./Max. only Miscellaneous settings C Cut moment peaks over columns | Section lines |
| | Refresh all | OK Cancel |
| | | |

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.

| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. De | sign | Steel design | Timber desig | In | | | |
|----------|----------|-------|----------|--------|----------|-----------|---------|----------|------|--------------|--------------|---------------|----|------------|-----------|
| | | (+) | Co.#2 (U | LS) | ▼ a | xb [mm2/n | ן ו | • | None | 2 | ~ 1 | ×max ▼ min | I: | T 🖓 | \$ |

Reinforcement parameters

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

| Surface reinf | forcement param | eters (Euro | code) | | | × |
|--------------------|--|-------------|-------------------|------------------------------|--|---|
| Materials | Reinforcement | Cracking | Shear | | | |
| Materia | l s Maximum agg | regate size | | | ~ | |
| Exposi | tion class | Top su | | tural class | S4 ~ | |
| XC1 E |)ry or underwater | | | | ~ | |
| XC1 E | Dry or underwater | Bottom s | | | ////////////////////////////////////// | |
| | | | unace | | | |
| Coeffic | ient for seismic f | orces | f _{se} = | 1 ~ | | |
| 🗹 Tak | earanalysis e into account co f _{ctm} | | ile stren | gth ε _{cs} [‰] = | = 0.473 | |
| Set curr Pick u | ent settings as de p » | fault | | OK | Cancel | |

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:

| Surface reinforcement paramete | ers (Euroc | ode) | | | × |
|--|------------------------------|------------|--------------|---|---|
| Materials Reinforcement C | racking | Shear | | | |
| Calculate with actual thi | ckness | | | | |
| Т | Thickness | ; (h) [mm | n] = 200 | ~ | |
| Unfavorable e | eccentric | ity (N > 0 | 0) = 0 | ~ * h | |
| Unfavorable e | eccentric | ity (N < 0 | 0) = 0 | ~ * h | |
| Concrete cover | | Di | iameter (n | nm) Direction | |
| c _T [mm] = 25 ∨ ≥ 25 | | | Ø = 1 | 0 🗸 🔀 | |
| TO TO THE | | | Ø = 1 | | |
| 7 6 7 1 | p reinforcen ttom reinfor | | | | |
| | | | Ø = 1 | | |
| c _B [mm] = 25 ∨ ≥ 25 | | | Ø = 1 | 0 ~ × y | |
| Apply minimum cover | | | | | |
| Load transfer | | | | | |
| Two-way slab | | | | | |
| One-way slab In local x direction | | O In Io | cal y direct | tion | |
| Take into account the requi | red | | op reinfor | | |
| minimum reinforcement | | | | nforcement | |
| Reinforcement directions | | | | +++++++++++++++++++++++++++++++++++++++ | |
| Local x, y | | | v | | |
| O Custom | | | ť, | ++++++++++++ ►X | |
| Set current settings as defau | ılt | | | | |
| Pick up >> | | | ОК | Cancel | |

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.

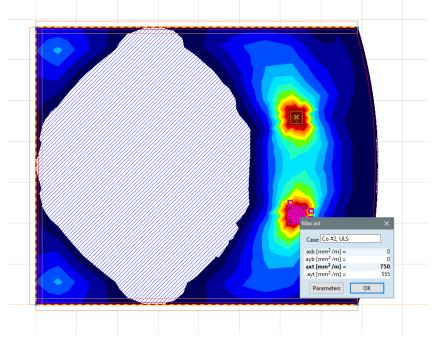
By setting the boundary values of **Color legend**, the areas that can be reinforced with a certain rebar spacing (e.g. for Ø10/160 reinforcement set 491 mm²/m) can be isolated.

Let us check the required top reinforcement in local **x** direction, click on **axt [mm²/m]** result component.



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

| /lodel extremes | × | | |
|------------------------------|------------------------------|--|--|
| Reinforcement valu | es | | |
| | | | |
| axt [mm ² /m] | ayt [mm ² /m] | | |
| axb [mm ² /m] | ayb [mm ² /m] | | |
| axb,axt [mm ² /m] | ayb,ayt [mm ² /m] | | |
| axb,ayb [mm ² /m] | axt,ayt [mm ² /m] | | |
| ОК | Cancel | | |



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.

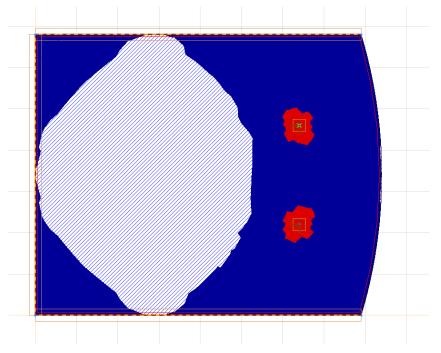
As a base reinforcement, use *Ø10/160 mm (491 mm²/m)* reinforcement on top and bottom layers in both direction. In the area of maximum moments use *Ø10/160 mm* extra reinforcement additionally (in these areas the total area of reinforcement will be *982 mm²/m*).

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.

| axt Color legend setup | × |
|------------------------|---|
| 982 | Values Colors |
| 491 0 | Levels 3 |
| | Limits Round calculated values |
| | O Min, Max of model 0 750 |
| | O Min, Max of parts 0 750 |
| | Absolute max. of model 0 750 |
| | Absolute max. of parts 0 750 |
| | © Custom |
| | Auto interpolate |
| | \bigcirc By step value $\triangle = 1$ |
| | ~ |
| | Save as |
| | Hatching for out of range values Opaque Transparent |
| | Color gradient direction Normal Reverse |
| | Isosurface contours |
| | Isoline labels |
| | Display |
| | ✓ Auto refresh |
| | ☐ Refresh all |
| | Calculate OK Cancel |



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*.

| • | New section pla | ne | | |
|---|--------------------|---------|------------|----------------|
| | Section plane name | column1 | | |
| | | | <u>O</u> K | <u>C</u> ancel |

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

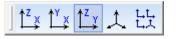
The following window will be displayed:

| Section lines | × |
|--|---|
| Section segments Section planes Section planes Section lines | × |
| | column1 Section plane |
| | Display mode |
| | Diagram only |
| | Diagram + average values Diagram + resultant integrated values |
| | Resultant integrated over the segment |
| | O Diagram with segment width |
| | Diagram on both side of the strip |
| | Δ _L [mm] = 0 |
| 1 of 1 | $\Delta_{L} [mm] = 0$ $\Delta_{R} [mm] = 0$ |
| Ne <u>w</u> section segment | · · · · · · · · · · · · · · · · · · · |
| New section segment group | Draw diagram in the plane of the elements |
| New section plane | Display |
| New section line | In all load cases |
| Modify | Current (Co.#2 (ULS)) In this load case only |
| Delete | For all result components |
| Section lines Draw section plane contour | Current (axt) For this result component only |
| Refresh all | OK Cancel |
| Auto refresh | |

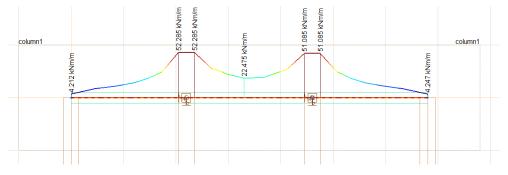
Click on **OK** to see moments in **x** direction in plane section of **column1**.



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



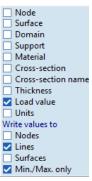
and change results from **Isosurfaces 2D** to **Section line**, then the internal forces are displayed in section defined previously. It can be seen on the diagram, that software cut moment peaks over columns because this function was activated previously when meshing was created (see **User's manual**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



View

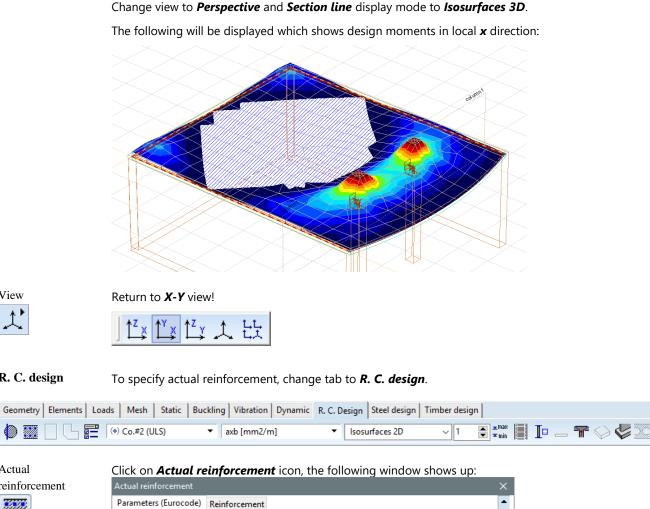
1'

R. C. design

0 🌌

Actual

reinforcement



Parameters (Eurocode) Reinforcement Min. thickness (h) [mm] = 200 Primary direction of reinforcement ∠ ≥ 25 c_{T} [mm] = 25 Top surface 💿 х 🗠 у 🔘 х 🔿 у t⊂. Bottom surface c_B [mm] = 25 ∠ ≥ 25 Apply minimum cover Use this rebar steel and concrete cover by default Auto refresh Pick up » Close

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:

| Actual reinforcement | | | | × |
|--|-----------------------|---|---------------------------------------|---|
| Parameters (Eurocode) Reinforcement | | | | |
| - x Direction - Top reinforcement (P): None - Bottom reinforcement (P) = 491 - 10 mm / 160 mm (30 mm) [R] - y Direction - Top reinforcement: None - Bottom reinforcement: None | | Rebars Type Ø[mm Spacing [mm Rebar position [mm A _s [mm ² /m] | h] = 160 \vee h] = 30 \vee = 49 | |
| ✓ Auto refresh | | <u>A</u> dd | <u>D</u> elete | |
| Pick up » 💦 🛄 🎝 💮 💭 | $ \circ \frac{0}{49}$ | 0 | Close | |

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

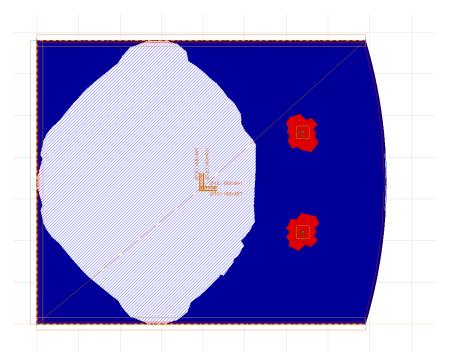
| Actual reinforcement | × |
|--|---|
| Parameters (Eurocode) Reinforcement | • |
| x Direction Top reinforcement (P) = 491 10 mm / 160 mm (30 mm) [R] Bottom reinforcement (P) = 491 10 mm / 160 mm (30 mm) [R] y Direction Top reinforcement = 491 10 mm / 160 mm (40 mm) [R] Bottom reinforcement = 491 10 mm / 160 mm (40 mm) [R] | RebarsTypeRibbed \emptyset [mm] =10 \checkmark β [mm] =160 \checkmark Spacing [mm] =40 \checkmark As [mm²/m] =491 \checkmark Calculate rebar positionsAddDelete |
| Auto refresh | হাত <u>491</u> Close |
| | 4 4 491 Close |

Reinforcement over an existing domain

Now, the set reinforcement should be assigned to the domain. Click on **Reinforcement over an exist**ing 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:



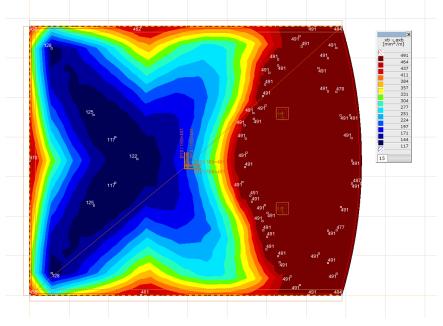
The thick brown lines and the titles indicates the given actual reinforcement. If only one direction of reinforcement is visible on the screen, then open **Display options**. On **Label** tab switch off **According to the displayed result component** function.

| Display options | × |
|---|---|
| Symbols Labels Switches | |
| Numbering 12 Node Truss Beam Rib Virtual beam Surface Domain Support Links Rigid Diaphragm Spring Gap Material Cross-section Design member Design optimization group Load panel Reference Use finite element numbers Visc Story center of gravity Story shear center Labels on lines seen from axis direction Transparent labels Prevent labels from overlapping Aute reference | Properties ABC Nodal coordinates Material name Cross-section name Bolted joint Column reinforcement Beam reinforcement Beam length Thickness Domain area Stiffness reduction COBIAX labels Concentrated Line Surface Temperature Fire Self weight Other Mass value Units Value Actual reinforcement Symbols Labels axb axb ayb ayb Suff ace Effect Concentrated Line Surface Effect Actual reinforcement Effect Symbols Labels axb axb ayb ayt Babels Effect Pebars + Reinforcement values Rebars + Quantity x (Length) According to the displayed result component Component |
| Auto refresh Refresh all Save as default | OK Cancel |

92

Select xb-axb result component in Reinforcement difference list.

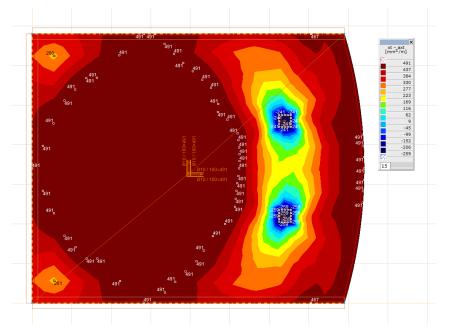
Click on **Numbering** icon (among **Speed buttons**) and uncheck **Write values to Surfaces** to see the also the values not only the colors:



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.

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.

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.

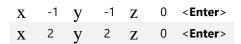
Using relative coordinates, enter the following coordinates:

Actual



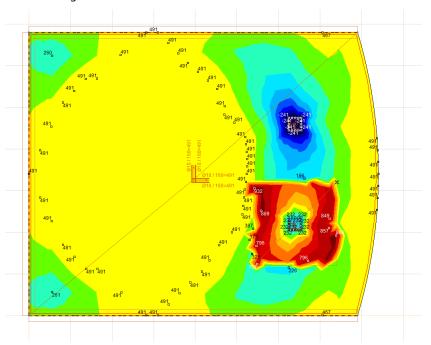
domain





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.

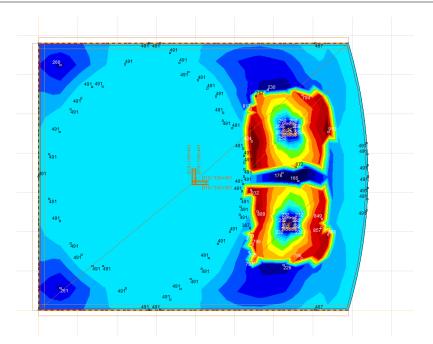


Copy this actual reinforcement domain to the upper nodal support. Click on **Translate** icon:

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**:

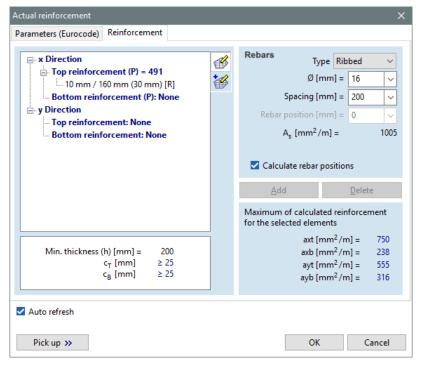
| Translate / Copy | × | | |
|--|--|--|--|
| N = 1 | e a a a | | |
| Method O Incremental O Distribute O Spread by distance | Nodes to connect | | |
| Consecutive Move Detach | ✓ Copy elements ☐ Copy loads ✓ Copy nodal masses ✓ Copy dimension symbols | | |
| ☐ With guidelines ☐ With DXF/PDF layer ☐ Visible layers only | OK Cancel | | |

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.

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



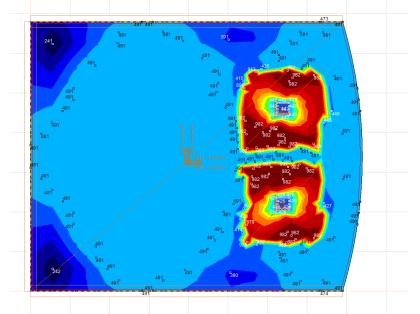
Actual reinforcement



| rameters (Eurocode) Reinforcement | | | | |
|--|----|--|--------------------------|---------|
| ■ x Direction | | Rebars | Type Ribbe | d V |
| — Top reinforcement (P) = 491 … 10 mm / 160 mm (30 mm) [R] | 24 | ç | ð [mm] = 1 | 0 ~ |
| Bottom reinforcement: None | | Spacing | g [mm] = 1 | 60 🗸 |
| | | | | |
| Top reinforcement = 491 | | Rebar position | n [mm] = 4 | 0 ~ |
| 10 mm / 160 mm (40 mm) [R] | | A _s (mm | n ² /m] = | 491 |
| Bottom reinforcement (P): None | | | | |
| | | Z Calculate reb | ar positions | |
| | | Add | D | elete |
| | | Maximum of calco for the selected ele | | rcement |
| | _ | ax | t [mm²/m] : | = 750 |
| Min. thickness (h) [mm] = 200 | | axi | b [mm ² /m] : | = 238 |
| c _T [mm] ≥ 25 | | | t [mm²/m] : | |
| c _B [mm] ≥ 25 | | ayl | o [mm²/m] : | = 316 |
| | | | | |
| Auto refresh | | | | |

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.

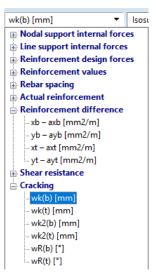


Now, the results of yt - ayt component are positive, so the actual (defined) reinforcement is sufficient for all reinforcement layers.

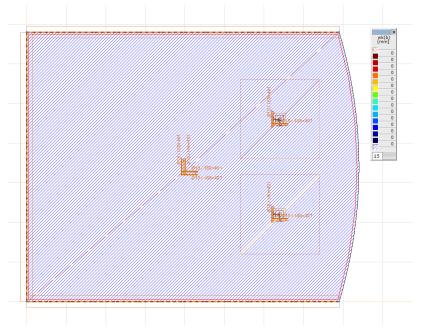
Check crack width in **Co.#1 (SLS)** load combination.

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).

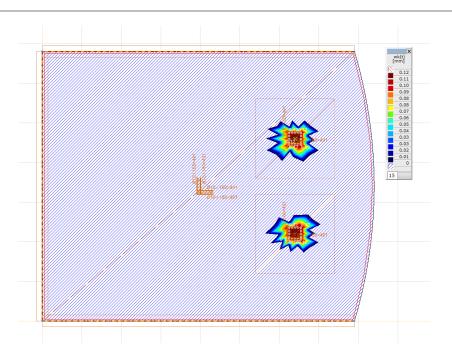


The following will be displayed:



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

analysis

Nonlinear static

→ u

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

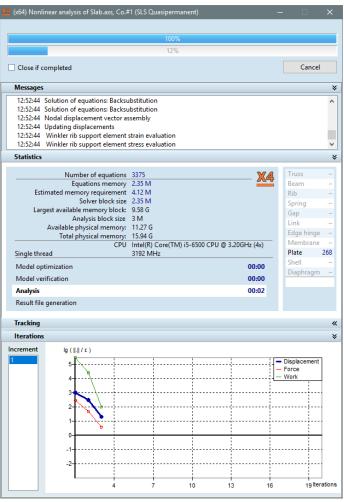
Click on **Nonlinear static analysis** icon and the following window shows up:

| reconnect state analysis | |
|---|--|
| Load cases All Coad cases Coad combinations Co.#2 (ULS) Co.#2 (ULS) Co.#2 (ULS) Co.#1 (SLS Quasipermanent) | Convergence criteria Maximum iterations 50 Displacement 0.001 Force 0.001 Work 1E-6 Use secant stiffness (only in appropriate cases) |
| 1 of 5 Solution control Force Displacement Direction: X | ✓ Use reinforcement in calculation |
| Pushover Maximal displacement: [mm] = 1 Equal increments Number of increments 10 | Nonlinearity ☐ Follow nonlinear behaviour of ☐ Follow geometric nonlinearity of materials and finite elements beams, trusses, ribs and shells |
| Increment function Bilinear> Bilinear> Bilinear> D 10 | Store last increment only OK Cancel |

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:

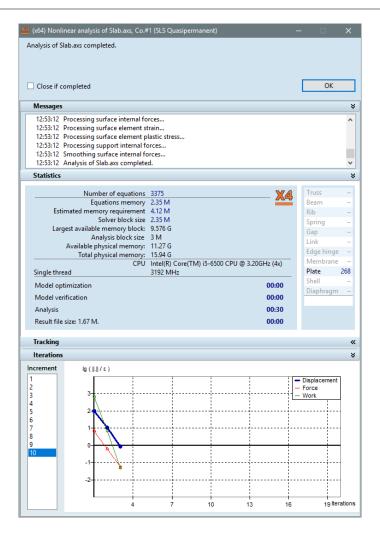
Statistics

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

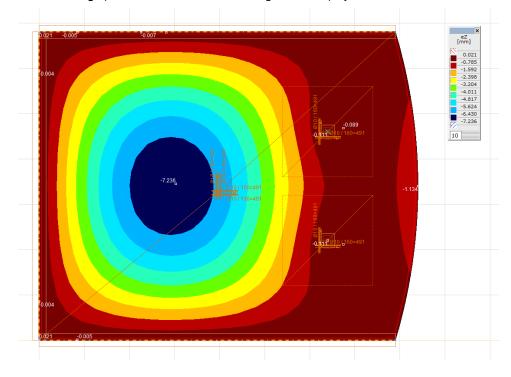


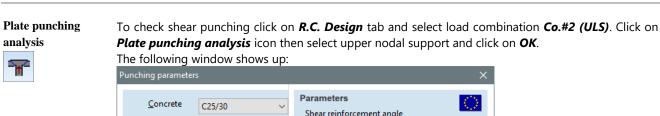
Iterations

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



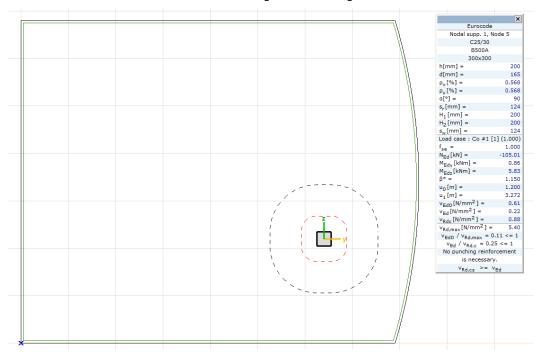
After click on **OK**, then program automatically activates vertical deformations **ez [mm]** in **Isosurfaces 2D** display mode in load combination **Co.#1** with nonlinear analysis. Switch of **Write values** to **Nodes** at **Numbering** speed button, then the following will be displayed:





| <u>C</u> oncrete | C25/30 ~ | Farameters |
|----------------------|-------------------------|--|
| | | Shear reinforcement angle |
| <u>R</u> ebar steel | B500A ~ | α[°] = 90 🗸 |
| | | Radial rebar spacing |
| Total plate thickne | ess | s _r [mm] = 124 ≤ 0,75d = 124 |
| 🗹 By reinforce | ment parameter | Distance of the first perimeter of shear reinforcem |
| h [mm | b] = 200 | r ₁ [mm] = 50 ≤ 0.5d = 83 |
| 🗹 Actual reinfo | prcement | |
| px [%] = 0.568 | в ру [%] = 0.614 | |
| | | Recalculate reinforcement for each control perimeter |
| | α 57 | β factor Calculated according to Eurocode 2 By column position Internal column Edge column Corner column Custom |
| Take soil reaction | on into account | β = |
| Coefficient for seis | mic forces | |
| f _{se} = 1 | | |
| 🕒 Open | | OK Cancel |

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

Design calculations

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

| 24 Design calculations | | | | | |
|--|--------------|--------|-----|----|---|
| Punching analysis result | | | | | ^ |
| Nodal supp. 1, Node 5 | | | | | |
| Input data | | | | | |
| Punching Code: Eurocode | | | | | |
| MaterialsConcrete: C25/30 $f_{ck} = 25$ N/mm²Rebar steel: B500A $f_{ywk} = 500.25$ N/mm² | | | | | |
| Geometry Cross-section: 300x300 Plate thickness: h = 0.2 m Rebar position: $u_x = 0.03 \text{ m}$ $u_y = 0.04 \text{ m}$ Effective thickness: $d_x = h - u_x = 0.2 - 0.03 = 0.17 \text{ m}$ $d_y = h - u_y = 0.2 - 0.04 = 0.16 \text{ m}$ Effective thickness: $d = \frac{d_x + d_y}{2} = \frac{0.17 + 0.16}{2} = 0.165 \text{ m}$ | | | | | |
| | Substitution | 100% ~ | 🖨 🛅 | ОК | |

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

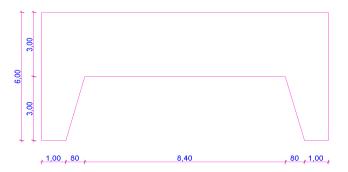
Objective

Start

New

Δ

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

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*.

| New model | | × |
|--------------------------------|---|---|
| Select a view to start with | <u>F</u> older | C:\Users\István T\Desktop\Step by Step\ |
| Y → X Top view | <u>M</u> odel file name: | membrane_1 |
| | Design code Units and formats <u>R</u> eport languagi | EU Change settings |
| Front view | Page header Proje | set |
| Perspective | | comment |
| | Project Analysis by Model: membrane_1.axs | |
| | | OK Cancel |

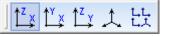
(Starting workplane (**X-Z** plane/front view) can also be set in this window if selected from the list in the left side.)

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 | | eometry tab to define the geometry and structural properties of the beam. On the the available functions are displayed. | tab the |
|-------------------------------------|---------------|--|---------|
| Geometry Elemen | ts Loads Mesh | Static Buckling Vibration Dynamic R. C. Design Steel design Timber design | |
| لـ / • | | ♡ ○ ++ 才 囲 躑 伊 伊 / × ※ ⅍ -+/ キ -*/ | € ⊞ |

Quad division

囲

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

| Quad d | livision | × |
|---------------|----------|------------------------------|
| $N_1 = N_2 =$ | 4 💽 | N_1 2 N_2 N_2 |
| | ОК | Cancel |

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

| Quad c | livisio | n | | × |
|------------------|---------|--------|---------------------|--------|
| N ₁ = | 20 | • | 1 | |
| N ₂ = | 8 | ▲ ▼ | N ₁ 2 | |
| 🗹 Cri | eating | surfac | es | 2 |
| | | ОК | | Cancel |

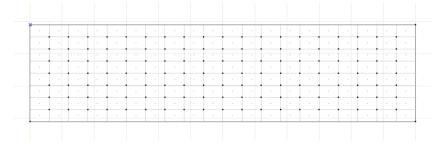
Close panel with **OK** and specify the corner points of the top 'body' of the wall. The nodes can be placed by editing tools or could be determined by their coordinates.

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:

| Х | 12 | у | 0 | Ζ | 0 | <enter></enter> |
|---|-----|---|---|---|---|-----------------|
| Х | 0 | у | 0 | Ζ | 3 | <enter></enter> |
| Х | -12 | у | 0 | Z | 0 | <enter></enter> |

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

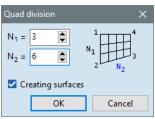


Quad division

囲

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:



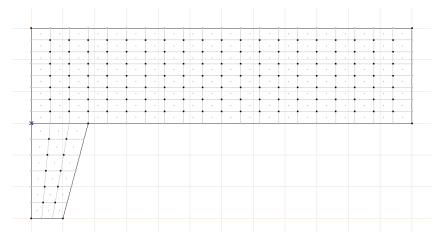
Close the panel with **OK** and specify the four corner nodes of the leg on the left side.

The coordinates are specified as follows:

| Х | 0 | у | 0 | Z | -6 | <enter></enter> |
|---|------|---|---|---|----|-----------------|
| Х | 1 | у | 0 | Ζ | 0 | <enter></enter> |
| Х | 0.8 | У | 0 | Ζ | 3 | <enter></enter> |
| Х | -1.8 | у | 0 | Z | 0 | <enter></enter> |
| | | | | | | |

Press **Esc** to finish data input.

The following geometry can be seen as a result:





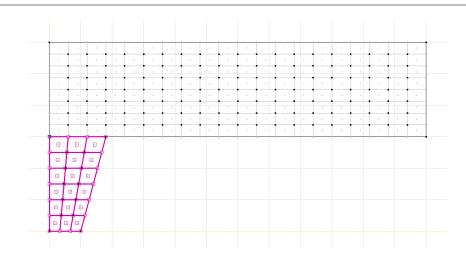
Mirror the previously drawn 'leg' to the symmetry axis of the wall (**X=6** *m*). The mirror function can be found on the vertical toolbar on the left side.

By clicking on the icon, the selecting panel activates:





The selected elements will be highlighted:

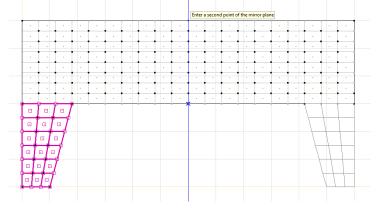


Finish the selection with **OK** and the following dialogue panel will be displayed:

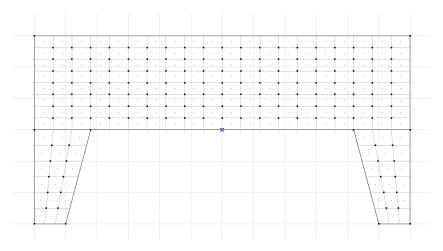
| Mirror | × |
|--|--|
| _ ♦ | |
| Mirror Copy Multiple Move Detach | Nodes to connect None Double selected All |
| Mirror local x-axis of line elements | ✓ Copy elements ✓ Copy loads ✓ Copy nodal masses ✓ Copy dimension symbols |
| ☐ With guidelines ☐ With DXF/PDF layer ☐ Visible layers only | OK Cancel |

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.

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





₩,

Zoom

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.

| Geometry check | \times |
|--|----------|
| Geometry check Tolerance [m] = 0.001 Selection of problematic nodes Selection of intersecting elements Select unattached nodes or lines List deleted nodes | |
| Check domain contours Tolerance [m] = 0.001 | |
| Check all loads Rebuilding of loads may cause loss of results. | |
| OK Cancel | |

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:

| () | Geometry check | |
|----|---|--|
| | Nodes deleted: 0 Lines deleted: 0 Surfaces deleted: 0 | |
| | <u>о</u> к | |

| Elements | | To define Surface elements, change tab to Elements : | | | | | | | | | | |
|----------|----------|--|------|--------|----------|-----------|---------|--------------|--------------|---------------|--------------|--|
| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | |
| | <i>*</i> | | | ő ť | V, E | [🌧 🛲 | • | 戽 ⊢ , | 5 1 1 | مز خل رقي 🆽 | : 14 14 14 (| |

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:

| Surface | elements | | | | × |
|---------|------------------|------------------------|-------------------|--------|------------------------------|
| ۲ | Define O N | lodify | | | |
| | Туре | Membran | e Plate Shell | | $\langle \mathbf{v} \rangle$ |
| | Material | | ~ | ð | \sim |
| | Thickness [mm] : | - ~ | | | |
| | | | | | |
| | Local x refere | nce | × Auto | \sim | × |
| | Local z refere | nce | × Auto | \sim | |
| | | | | | |
| | | | | | |
| | Color | I I By ma I I By ma | | | |
| Pi | ick up | | | ОК | Cancel |

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

| <u>)</u> esign code | <u>M</u> aterials | | C25/30 | | | |
|--------------------------|-------------------|---|--------------------------------------|-----------|--------|-----|
| ⊕ CSA S6-06 [Rev. 2010 ∧ | C12/15 | ^ | Туре | Concrete | | |
| 🕀 DIN (German) | C16/20 | | | Isotropic | | |
| Eurocode | C20/25 C25/30 | _ | E [N/mm ²] | 31500 | | |
| Eurocode [A] | C30/37 | | v | 0.20 | | 100 |
| Eurocode [B] | C35/45 | | α _τ [1/°C] | 1E-5 | | |
| Eurocode [CZ] | C40/50 | | ρ[kg/m ³] | 2500 | | |
| Eurocode [D] | C45/55 C50/60 | | f _{ck} [N/mm ²] | 25.00 | | |
| Eurocode [FIN] | C14 | | | 1.500 | | |
| Eurocode [H] | C16 | | Yc | 1.00 | | |
| Eurocode [NL] | C18 | | α _{cc} | | | |
| Eurocode [PL] | C20 C22 | | Φt | 2.00 | | |
| Eurocode [RO] | C22 | | | | | |
| Eurocode [SK] | C27 | | | | | |
| Eurocode [UK] | C30 | | | | OK | |
| MSZ (Hungarian) | C35 | | | | UK | |
| NBCC 1995 ✓ | C40 C45 | ~ | | | Cancel | |

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.

Material library import



Thickness

| Surface elements | | × |
|---|-------------------|-----------|
| Oefine | | |
| Type Membran | e Plate Shell | Ň |
| Material C25/30 Thickness [mm] = 200 ~ | | ~ |
| Local x reference | × Auto 🗸 | × |
| Local z reference | × Auto 🗸 | |
| <u>S</u> tiffness factors | * | |
| Color 📃 🗹 By ma | | duction |
| Pick up >> | | OK Cancel |



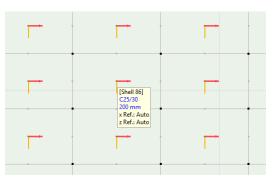


To view the local coordinate system of the *Surface elements*, click on *Display Options* icon in the icons menu bar on the left side. The following dialogue window is displayed:

| Display options | × |
|---|---|
| Symbols Labels Switches | |
| Graphics symbols Node Trusses Beams Virtual beams Ribs Center of circle Domain Surface center Mesh Nodal support Line support Surface support | Local systems Beam Rib Virtual beam Surface Domain Support Spring Gap Link Edge hinge Load panel |
| Joinglay footings Display footings Dimension lines Detailed dimension lines Springs Gap elements Links Rigids Diaphragm Reference Cross-section shape End releases Structural members Reinforcement param. Reinforcement domain Mass | Loads Concentrated Line Surface Temperature Fire Self weight Other Load panel Abutting wall or parapet (for snow loads) / Edges with return corner (for wind loads) Load distribution scheme Derived beam load Moving load phases |
| Thickness reference points | Transparent load diagrams Object contours in 3D |
| Auto refresh Refresh all Save as default | OK Cancel |

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

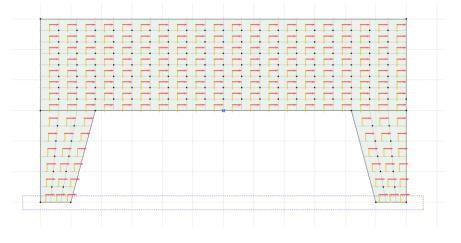
If the *Mesh*, *Node*, *Surface centre* is switched on among the *Graphics Symbols* in the *Display Options*, you can see that the program uses 9-node membrane elements. These 9 nodes are the 4 corners, 4 midpoints and the centre point of surface element. If you move the cursor over the surface centre 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:

| Line supports | | | × |
|--|----------------------|-------------|--------|
| Define | | | |
| Direction Global Beam/Rib relative Edge relative | Nonlinear parameters | z V X | k t |
| $R_{\chi} [kN/m/m] = 1E+7 \lor$ | | | |
| $R_{Y} [kN/m/m] = 1E+7 \vee$ | | | |
| $R_{Z} [kN/m/m] = 1E+7 \sim$ | | | |
| R_{XX} [kNm/rad/m] = 1E+7 \checkmark | | | |
| R_{YY} [kNm/rad/m] = 1E+7 \checkmark | | | |
| R_{ZZ} [kNm/rad/m] = 1E+7 \checkmark | | | |
| | | | |
| Pick up >> | | ОК | Cancel |

To define a pinned support, use the following settings:

| ine supports X |
|---|
| Define O Modify |
| Direction © Global © Beam/Rib relative © Edge relative □ Nonlinear parameters |
| R _X [kN/m/m] = 1E+7 ~~ |
| $R_{Y} [kN/m/m] = 0 \qquad \checkmark$ |
| $R_{Z} [kN/m/m] = 1E+7 \qquad \checkmark$ |
| $R_{XX} [kNm/rad/m] = 0$ |
| R_{YY} [kNm/rad/m] = 0 \checkmark |
| $R_{ZZ} [kNm/rad/m] = 0 \qquad \checkmark$ |
| Pick up >> OK Cancel |

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.



New

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

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**.

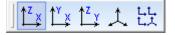
| New model | | | |
|--------------------------------|---|------------------------|---|
| Select a view to start with | | <u>F</u> older | C:\Users\lstván T\Desktop\Step by Step\ \begin{bmatrix} 2 & D & D & D & D & D & D & |
| , | <u>M</u> odel fi | le name: | membrane_2 |
| Top view | | | Eurocode Change settings |
| Front view | <u>R</u> eport language | | 🗱 English 🗸 |
| | Page header | | |
| Perspective | (Company logo) | Projec Analys Co | |
| | Project Analysis by Model: membrane_ | 2.axs | |
| | | | OK Cancel |

(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.

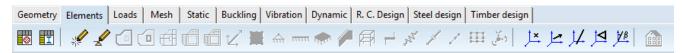
In the following the geometry of the wall structure is created by editing toolbar.

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:

| | 🗢 🛞 🖉 🚺 |
|---------------------|----------------|
| Туре | Shell |
| Material | - - - - |
| Local x orientation | × Auto |
| Local z reference | × Auto |
| Thickness [mm] | 200 |
| | |
| | |



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:

| Warning | |
|---|----------|
| No element material defined. Browse material library | |
| browse material library | 01 |
| | <u> </u> |

Material library import

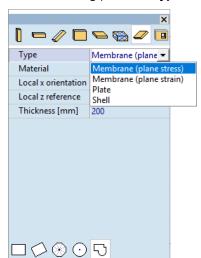
The following, Cross-section editor window shows after clicking.

| esign code | <u>M</u> aterials | | C25/30 | | | |
|---------------------------|-------------------|---|--------------------------------------|-----------|--------|----------|
| 🕀 CSA S6-06 [Rev. 2010 🔺 | C12/15 | ^ | Туре | Concrete | | |
| 🗄 DIN (German) | C16/20 | | | Isotropic | | |
| <mark>⊞ -</mark> Eurocode | C20/25 C25/30 | _ | E [N/mm ²] | 31500 | | |
| Eurocode [A] | C30/37 | | v | 0.20 | 1000 | (4)-13-1 |
| Eurocode [B] | C35/45 | | α _τ [1/°C] | 1E-5 | | |
| Eurocode [CZ] | C40/50 | | ρ[kg/m ³] | 2500 | | |
| Eurocode [D] | C45/55 C50/60 | | f _{ck} [N/mm ²] | 25.00 | | |
| Eurocode [FIN] | C14 | | en - | | | |
| Eurocode [H] | C16 | | γ _c | 1.500 | | |
| Eurocode [NL] | C18 | | α _{cc} | 1.00 | | |
| Eurocode [PL] | C20 | | Φt | 2.00 | | |
| Eurocode [RO] | C22 C24 | | | | | |
| Eurocode [SK] | C24 C27 | | | | | |
| Eurocode [UK] | C30 | | | | | |
| 🗄 MSZ (Hungarian) | C35 | | | | OK | |
| ₩ NBCC 1995 | C40 | | | | Connel | |
| K NBCC 1995 | C45 | ~ | | | Cancel | |

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

Type

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





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 (**d***X*, **d***Y*, etc...).

| × | | dX[m]: | 7.500 | | d r[m] | : | 12.985 |
|---|---|-------------------------------|--------|---|--------|---|--------|
| | Ъ | dY[m] : | 0 | | d a[°] | | |
| | a | dY[m] : dZ[m] : dL[m] : | 10.600 | a | dh[m] | : | 0 |
| | | dL[m] : | 12.985 | | | | |

Continue defining next points of the domain with relative coordinates:

| Х | 1 | У | 0 | Ζ | 0 | <enter></enter> |
|---|-----|---|---|---|----|-----------------|
| Х | 0.8 | у | 0 | Z | 3 | <enter></enter> |
| Х | 8.4 | у | 0 | Ζ | 0 | <enter></enter> |
| Х | 0.8 | у | 0 | Z | -3 | <enter></enter> |
| Х | 1 | у | 0 | Z | 0 | <enter></enter> |
| Х | 0 | у | 0 | Z | 6 | <enter></enter> |
| Х | -12 | у | 0 | Z | 0 | <enter></enter> |
| Х | 0 | у | 0 | Z | -6 | <enter></enter> |

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

The following will be displayed in the main window:



ΔΔ

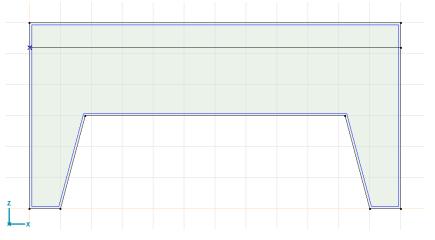
Mesh

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:

| [Domain 1 [membrane]] 200 mm C25/30 44.400 m ² 22200.000 kg x Ref.: Auto, z Ref.: Auto |
|---|
| |

| Mesh | | Click on <i>Mesh</i> tab for domain meshi | | | | | |
|------|----------|---|-------|------|--------|----------|------|
| | Geometry | Elements | Loads | Mesh | Static | Buckling | Vibr |

| Geometry Elements Loads | Mesh | Static Buckling | Vibration Dynar | nic R. C. Design | Steel design | Timber design |
|--|------|-----------------|-----------------|------------------|--------------|---------------|
| $\mathbf{k} = \mathbf{k} = $ | | B 🔨 🕅 | × | | | |

Domain meshing

Ó

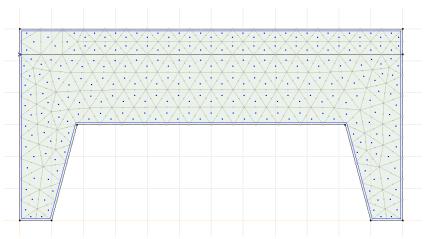
Click on **Domain Meshing** icon. Select the domain with the **All** (*) button and finish selection with **OK**. The following dialogue window will be displayed:

| Meshing parameters | × |
|--|---|
| Define | > Modify |
| Mesh type | |
| Average me | esh element size [m] = 0.70 🗸 |
| IU | Point loads $ \ge [kN] = 0$ ne loads $ \ge [kN/m] = 0$ e loads $ \ge [kN/m^2] = 0$ |
| Adjust mesh to c cutting of mome | olumn heads (to enable nt peaks) |
| Contour division met | ze |
| Smoothing | |
| Create mesh only for u Calculation of domain Keep generated mesh o | |
| | OK Cancel |

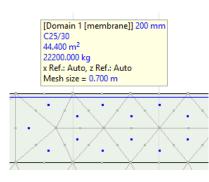
Type in **0.7** [m] for the *Average mesh element size*. After closing the dialog with *OK*, the automatic mesh generation starts. The progress of *Mesh generation* is shown in the window:

| Mesh generation | |
|-----------------|-------|
| 15% | |
| | |
| | Abort |

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:



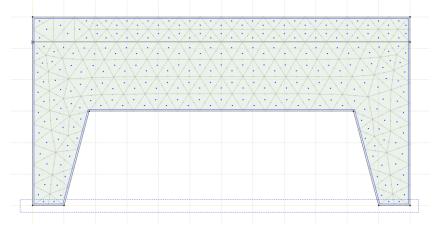
Elements

Return to **Elements** tab:

| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | |
|----------|----------|-------|------|--------|----------|-----------|---------|--------------|--------------|---------------|---------------------|--|
| | <i>*</i> | | • | í í | Ľ | 🌧 🚥 | • 🗢 🌈 | 骨子, | 5 1 1 | ⊞ 渉 九 | ا <u>مبر احر با</u> | |

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.



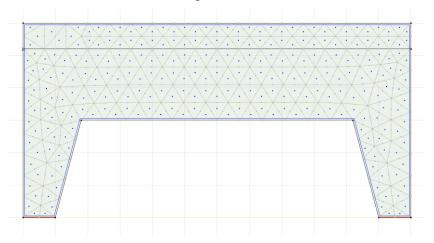
Press **OK** to go on, the following window shows up:

| Line supports | × |
|---|----------------------|
| Define O Modify | |
| Direction ● Global ◇ Beam/Rib relative ◇ Edge relative | Nonlinear parameters |
| $R_{\chi} [kN/m/m] = 1E+7 \lor$ | |
| R _Y [kN/m/m] = 1E+7 ∨ | |
| $R_{Z} [kN/m/m] = 1E+7 \sim$ | |
| R_{XX} [kNm/rad/m] = 1E+7 \checkmark | |
| R_{YY} [kNm/rad/m] = 1E+7 \sim | |
| R_{ZZ} [kNm/rad/m] = 1E+7 \checkmark | |
| Pick up >> | OK Cancel |

To define a pinned support, use the following settings:

| Line supports | | | | | × |
|---|-------------|----------------------|-------------|---|-------|
| Define | O Modify | | | | |
| Direction Global Beam/Rib relative Edge relative | ative | | z V X | t | 4 |
| | | Nonlinear parameters | | | |
| R _X [kN/m/m | i] = 1E+7 ~ | | | | |
| R _Y [kN/m/m |] = 0 ~ | | | | |
| R _Z [kN/m/m | i] = 1E+7 ~ | | | | |
| R _{XX} [kNm/rad/m | i] = 0 ~ | | | | |
| R _{YY} [kNm/rad/m | i] = 0 ~ | | | | |
| R _{ZZ} [kNm/rad/m | •] = 0 ~ | | | | |
| Pick up » | | | OK | C | ancel |

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

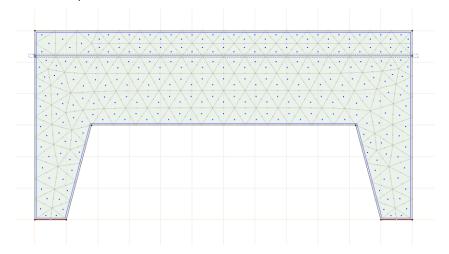


With this final step, the finite element definition of the wall structure has been completed. In the following, we proceed the analysis with this model (*Geometry definition using domains*).

| Loads | ٦ | The next | t step i | s to spe | cify the | loads on | the wall | structure. C | Click on Loa | ds tab: | | | |
|-----------------|----------|----------|----------|------------|-----------|----------|--------------|--------------------------|---------------------|----------------|-----|------------------|--------|
| Geometry Elemen | ts Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | | | |
| 프 - 블 (수 | • 📥 🕭 | - 4 | , 🌆 🍯 |) <i>@</i> | 87 | ¥* 🛪 🕯 | G ALP | н <mark>Е н Дат</mark> 🌡 | 🗊 🚣 🛲 🖉 | 1 🗠 📐 | 亚国士 | <u>&</u> 2 | ™ ● |

Surface edge loads

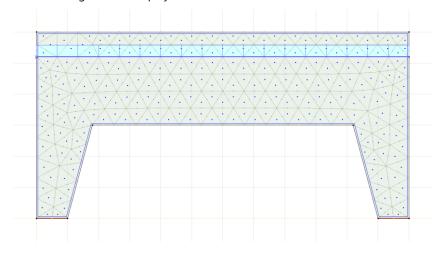
Assume a **50** *kN/m* vertical distributed load. Click on **Surface Edge Load** icon, then select the second line from top (created with the translate command before):



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

| Edge load on membranes | × |
|---|-----------|
| Oefine | |
| Direction Global on surface Global projective Local Overwrite Add | |
| p _x [kN/m] = 0 ~ | |
| p _y [kN/m] = 50 ∨ | |
| p _z [kN/m] = 0 ~ | |
| Pick up >> | OK Cancel |

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



| Static | Т | he nex | t step i | s the and | alysis and | d post pi | rocessing. (| Click on St | atic tab: | |
|-------------------|-------|--------|----------|-----------|------------|-----------|--------------|--------------------|------------------|--|
| Geometry Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | |
| | | | | - | | | - | | ~ | |

Linear static



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:

| | Nodal DOF | |
|---|---|--|
| • | (Membrane in X-Z plane) | |
| | Set the nodal degrees of freedom automatically? Save model with these settings | |
| | <u>Y</u> es <u>N</u> o | |

Click on Linear static analysis icon to start analysis.

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:

| 🤽 (x64) Linear analysis of membrane_2.axs | | | | × |
|---|--|---------------|---------|-----|
| Writing model data to disk | | | | |
| | | | | |
| | 7% | | | |
| Close if completed | | | Cancel | |
| | | | | |
| Messages | | | | « |
| Statistics | | | | ≽ |
| Number of equations 175 | 755 | Truss | | - |
| Equations memory 710 | | Beam | | - |
| Estimated memory requirement 2.1 | 13 M | Rib | | - |
| Solver block size - | | Spring | | - |
| Largest available memory block: 9.49 Analysis block size - | 492 G | Gap | | - |
| Available physical memory: 11. | .17 G | Link | | - |
| Total physical memory: 15.8 | | Edge h | - | - |
| | tel(R) Core(TM) i5-6500 CPU @ 3.20GHz (4x) 92 MHz | Memb Plate | rane | 260 |
| | | Shell | | _ |
| Model optimization | 00:00 | Diaphr | agm | - |
| Model verification | 00:00 | Load c | ase | 1 |
| Analysis | | Combi | ination | - |
| Result file generation | | | | |
| | | | | |

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

| 🧯 (хб4) Linear analysis of membrane_2.axs | : | - 🗆 | × |
|---|--|-------------|-----|
| Analysis of membrane_2.axs completed. | | | |
| Close if completed | | ОК | |
| Messages | | | « |
| Statistics | | | × |
| Number of constitutions | 1755 | Truss | |
| Number of equations Equations memory | | Beam | - |
| Estimated memory requirement | | Rib | - |
| Solver block size | | Spring | - |
| Largest available memory block: | | Gap | - |
| Analysis block size | | Link | - |
| Available physical memory: | | Edge hinge | - |
| Total physical memory: | Intel(R) Core(TM) i5-6500 CPU @ 3.20GHz (4x) | Membrane | 260 |
| Single thread | 3192 MHz | Plate | - |
| | 00:00 | Shell | - |
| Model optimization | | Diaphragm | - |
| Model verification | 00:00 | Load case | 1 |
| Analysis | 00:00 | Combination | n – |
| Result file size: 262 k. | 00:00 | | |
| | | | |

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

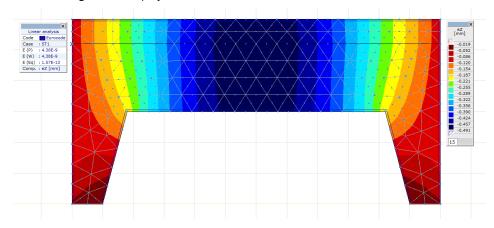


⇔

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:

| ⇒ Displacement - eX [mm] - eY [mm] - eY [mm] - fZ [rad] - fY [rad] - fZ [rad] - eR [mm] - fR [rad] Surface internal forces - nx [kN/m] - nx [kN/m] - nx [kN/m] - n2 [kN/m] - nxD [kN/m] - nyD [kN/m] Surface stresses ⇒ Line support internal forces | eZ [mm] 🔹 | lsosurfac |
|---|------------------------|-----------|
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | Displacement | |
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | eX [mm] | |
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | eY [mm] | |
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | eZ [mm] | |
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | fX [rad] | |
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | fY [rad] | |
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | fZ [rad] | |
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | eR [mm] | |
| Surface internal forces nx [kN/m] ny [kN/m] nxy [kN/m] vEd [kN/m] n1 [kN/m] n2 [kN/m] nx [kN/m] nxD [kN/m] nxD [kN/m] Intensity variation Surface stresses | … fR [rad] | |
| Intensity variation Surface stresses | Surface internal force | 5 |
| Intensity variation Surface stresses | <u>nx [kN/m]</u> | |
| Intensity variation Surface stresses | ny [kN/m] | |
| Intensity variation Surface stresses | nxy [kN/m] | |
| Intensity variation Surface stresses | vEd [kN/m] | |
| Intensity variation Surface stresses | n1 [kN/m] | |
| Intensity variation Surface stresses | n2 [kN/m] | |
| Intensity variation Surface stresses | an [°] | |
| Intensity variation Surface stresses | … nxD [kN/m] | |
| E Surface stresses | nyD [kN/m] | |
| T | Intensity variation | |
| Line support internal forces | Surface stresses | |
| | Line support internal | forces |
| | | |
| | | |

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

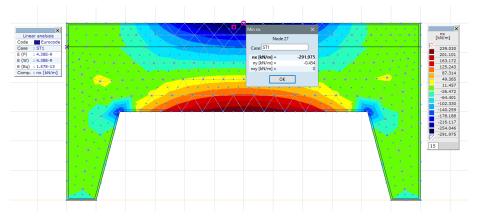


Min, max values

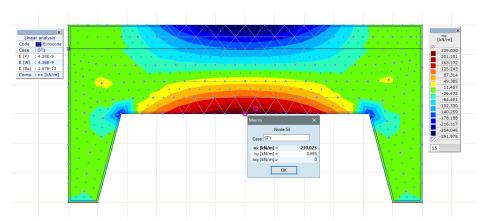
To find location of minimum and maximum values use *Min, Max values* function. By clicking on its icon shows following window:

| Aodel extremes | | × |
|--------------------|-------------------|------------------------------|
| Surface forces | | |
| nx [kN/m] | n1 [kN/m] | nxD [kN/m] |
| ny [kN/m] | n2 [kN/m] | nyD [kN/m] |
| nxy [kN/m] | an [°] | Total |
| тх [kNm/m] | m1 [kNm/m] | mxD+ [kNm/m] |
| my [kNm/m] | m2 [kNm/m] | mxD- [kNm/m] |
| тжу [kNm/m] | am [°] | Total |
| vxz [kN/m] | | myD+ [kNm/m] |
| vyz [kN/m] | | myD+ [kNm/m] myD- [kNm/m] |
| vEd [kN/m] | | ' |
| | ОК | Cancel |

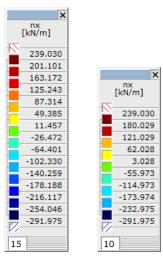
Select one of the *Surface force* components. Confirm with *OK*, and the program shows the negative maximum value and its location as well.



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.

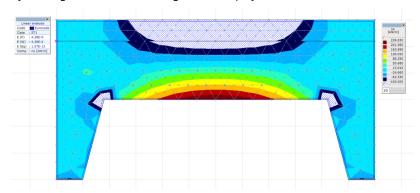


Color legend

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**:

| nx Color legend setup | | \times |
|--|--|----------|
| 239.030 | Values Colors | |
| 180.029 121.029 | Levels 10 | |
| 62.028 3.028 -55.973 -114.973 -173.974 -232.975 -100 | Limits ☐ Round calculated values ○ Min, Max of model -291.975 239.030 ○ Min, Max of parts -291.975 239.030 ○ Absolute max. of model -291.989 291.989 ○ Absolute max. of parts -291.989 291.989 ○ Custom ● ● Auto interpolate ● ● By step value Δ = 1 ✓ Save as ✓ Loor gradient direction Normal Reverse □ Isosurface contours □ Isoline labels | |
| | ✓ Auto refresh □ Refresh all | |
| | OK Cance | : |

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.

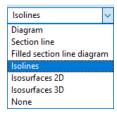


By clicking on **OK**, the following will be displayed:

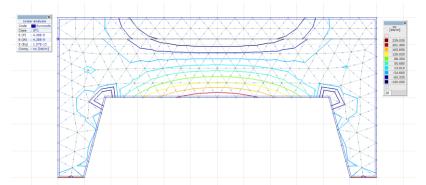
Areas with *nx surface force* exceeding -100 kN/m are hatched in blue.

Isolines

After, let us have a look at the internal forces in *Isolines Display mode*. Click on arrow right to *Isosur-faces 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.

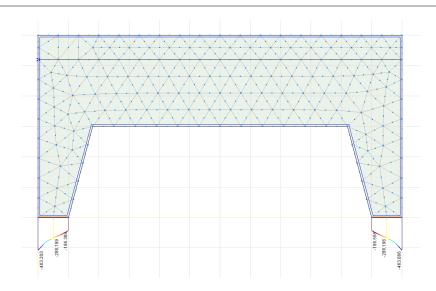
Result display param-



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

| Display parameters | | |
|--|---|---------------|
| Linear analysis Nonlinear analysis Dynamic analysis Case Envelope Critical | | |
| Load case III ST1 v | Component Rz [kN/m] Scale by 1 Display mode Diagram Display shape Undeformed Write values to Nodes C Lines Surfaces Surfaces Min./Max.only Miscellaneous settings C Cut moment peaks over columns | Section lines |
| | Refresh all | OK Cancel |

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:

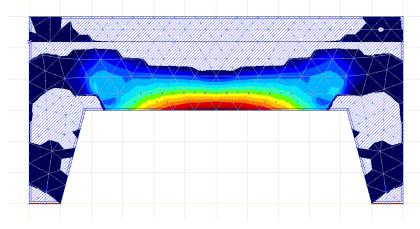
| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Desigi | Steel design | Timber design | n | | |
|----------|----------|-------|------|--------|----------|-----------|---------|--------------|--------------|---------------|---------------|-----------|--|
| | | Ξ +++ | ST1 | | ▼ a | xb [mm2/m | ו] | ▼ No | ne | ~ 1 | ×max ▼ min | I – T V 🗸 | |

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:

| Surface rein | forcement param | eters (Euro | code) | | | × |
|--------------|--------------------------------|-------------|-------------------|-------------------------|--------|---|
| Materials | Reinforcement | Cracking | Shear | | | |
| Materia | ıls Maximum agg | regate size | | C25/30 30 ~ B500A | ~ | |
| | | Ŧ | | tural class | S3 ~ | |
| - | tion class Humid, seldom di | Top su v | пасе | | ~ | |
| | | ////// | | | | |
| XC2 H | Humid, seldom di | - | | | ~ | |
| | | Bottom s | urface | | | |
| Coeffi | cient for seismic f | orces | f _{se} = | 1 ~ | | |
| | | | | | | |
| | ent settings as de | fault | | | | |
| Pick u | р »> | | | ОК | Cancel | |



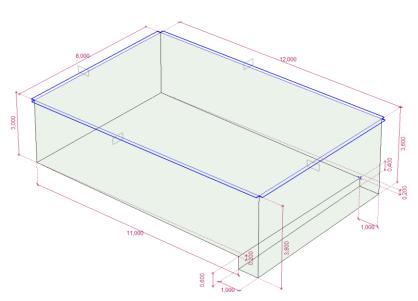
Close dialog window with **OK** and the **axb [mm²/m]** diagram is displayed in **Isosurfaces 2D** view:

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 Euro-code 2 for design.

Start X4

New

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

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*.

| New model | | | |
|--------------------------------|--|--------------------------------|--|
| Select a view to start with | <u>M</u> odel fil | <u>F</u> older le name: | C:\Users\lstván T\Desktop\Step by Step\ v reservoir |
| Top view | <u>U</u> nits and | gn code formats language | Eurocode Change settings EU Change settings |
| Front view Z Perspective | Page header (Company logo) | Projec Analys Co | |
| | Project Analysis by Model: reservoir.ax : | s | |
| | | | OK Cancel |

Options

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

| Options | × |
|---|---|
| Grid & Cursor Editing Drawing | |
| Grid ✓ Display ΔX [m] = 1.000 ΔY [m] = 1.000 ΔZ [m] = 1.000 Iype Grid lines ✓ | Cursor step Mouse snap $\Delta X [m] = 0.100$ $\Delta Y [m] = 0.100$ $\Delta Z [m] = 0.100$ <u>C</u> trl x 0.1 |
| ✓ Auto refresh ☐ Save as default | OK Cancel |

Replace each value under *Cursor step* to 0.2.

| Options | | | × |
|--------------------------------|---------|----------------|--------|
| Grid & Cursor Editing | Drawing | | |
| Grid Display | | Cursor step | ар |
| ΔX [m] = 1.000 | | ΔX [m] = | 0.2 |
| ΔY [m] = 1.000 | | ΔY [m] = | 0.2 |
| ΔZ [m] = 1.000 | | ΔΖ [m] = | 0.2 |
| <u>Type</u> Grid lines | ~ | <u>C</u> trl x | 0.1 |
| ✓ Auto refresh Save as default | | ОК | Cancel |

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.

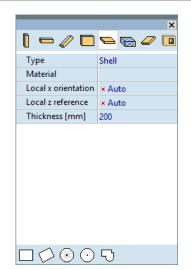


Elements



Draw objects directly

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





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

| Warning No element material defined. Browse material library | |
|--|---|
| <u>о</u> к |] |

Material library import

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

| esign code | <u>M</u> aterials | | C25/30 | | | |
|---------------------------|-------------------|---|--------------------------------------|-----------|----------------|---------|
| 🕀 CSA S6-06 [Rev. 2010 🔺 | \$350GD+Z275 | ^ | Туре | Concrete | | |
| 🕀 DIN (German) | C12/15 | | | Isotropic | | |
| <mark>⊞ -</mark> Eurocode | C16/20 C20/25 | | E [N/mm ²] | 31500 | | |
| Eurocode [A] | C25/30 | | v | 0.20 | and the second | 49-15-5 |
| Eurocode [B] | C30/37 | | α _τ [1/°C] | 1E-5 | | |
| Eurocode [CZ] | C35/45 | | ρ[kg/m ³] | 2500 | | |
| Eurocode [D] | C40/50 C45/55 | | f _{ck} [N/mm ²] | 25.00 | | |
| Eurocode [FIN] | C50/60 | | CN | 1.500 | | |
| Eurocode [H] | C14 | | γ _c | | | |
| Eurocode [NL] | C16 | | α _{cc} | 1.00 | | |
| Eurocode [PL] | C18 | | φ _t | 2.00 | | |
| Eurocode [RO] | C20 C22 | | | | | |
| Eurocode [SK] | C24 | | | | | |
| Eurocode [UK] | C27 | | | | 01/ | |
| 🗄 MSZ (Hungarian) | C30 | | | | OK | |
| • NBCC 1995 | C35 | | | | Cancel | |
| < > | C40 | ¥ | | | Cancel | |

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

Firstly, let us draw the side wall of the reservoir in the X-Z plane.

Thickness

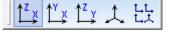
Set **Thickness [mm]** to **250**.

Complex slab





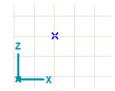
Change view to **Front view** (**X-Z** plane):



can be defined by coordinates.

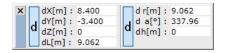
Choose the global origin as the first point of the polygon. It is at the bottom left, where the horizontal and vertical brown lines (representing the global X and Z axes) intersect. The blue cross (rotated 45°) shows the current origin of the actual coordinate system.

Click on Complex slab icon right to the first. The contour line can be drawn directly on the screen or



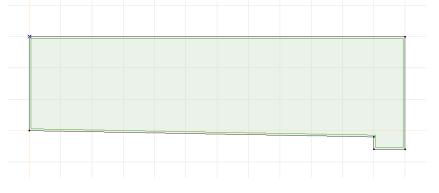
Nodes in relative coordinate system

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:





Change view to **Perspective**:

t^z x t^y x t^z y t Et

H 30.0; V 320.0; P 0.0.

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:

× + - ½ | * ← 府 iiii | OK Cancel

Click on **X** button at top right corner to close window.

Specify the following values in the *Perspective settings* window:

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

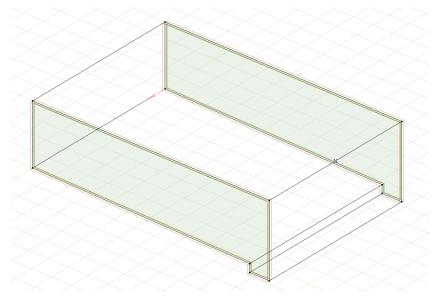
| Mirror | × |
|--|--|
| (¹) | |
| Mirror Copy Multiple Move Detach | Nodes to connect None Double selected All |
| Mirror local x-axis of line elements | ✓ Copy elements ✓ Copy loads ✓ Copy nodal masses ✓ Copy dimension symbols |
| With guidelines With DXF/PDF layer Visible layers only | OK Cancel |

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.

Press Esc once to close the command.

The following will be displayed:





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





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:

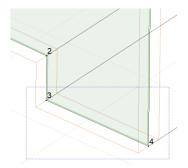
| ~ | Node |
|---|-----------|
| | Domain |
| | Material |
| | Thickness |
| | Reference |
| | Units |

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.

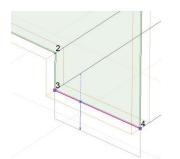
Translate / Copy To specify a slope for the watercourse, move the line between **node 3** and **4** down by **0.2** *m*.

| Geometry | у | Ch | ange t | ab to G | Geometr | y : | | | | |
|----------|----------|-------|--------|----------------|----------|------------|---------|--------------|--------------|----------------|
| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design |
| • / | لـ ` |) 🖾 (| ЭØ | $ \odot$ | O +⁄ | 4 津田 | ₩3 ₽ | | < 🗶 🍕 | -≯≁-*×⊗⊗⊗≥√≤≒⊞ |

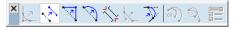
Select the line between *nodes 3* and *4* by a selection rectangle:



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:

| Х | 0 | У | 0 | Ζ | -0.2 | <enter></enter> |
|---|---|---|---|---|------|-----------------|
|---|---|---|---|---|------|-----------------|



⋕

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

| Geometry check | > |
|---|----|
| Geometry check Tolerance [m] = 0.001 Selection of problematic nodes Selection of intersecting elements Select unattached nodes or lines List deleted nodes | |
| Check domain contours Tolerance [m] = 0.001 | |
| Check all loads Rebuilding of loads may cause loss of results. | |
| OK Can | el |

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:

| 6 | Geometry check | |
|---|---|------------|
| | Nodes deleted: 0 Lines deleted: 0 Surfaces deleted: 0 | |
| | | <u>O</u> K |

Elements

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

| Geometr | / Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | |
|---------|------------|-------|------|--------|----------|-----------|---------|--------------|--------------|---------------|-------------|---|
| 0 1 | 1 🦊 🖌 | | 0 付 | í í | 12 🖩 | [🌧 🛲 | • 🗢 🎸 | 骨一, | x / / | 표 🕹 🗍 | × 년 🏹 💆 🏄 🏫 | I |

Reference point

x

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 inplane 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 $\frac{1}{2}$. Press **Esc** to exit from function.



⊴

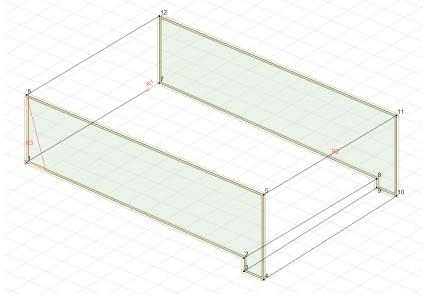
Move the cursor over the *Numbering* icon on the *Speed buttons* toolbar. Turn on the *Reference* checkbox. Now, an *R2* label appears beside the reference symbol.

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.



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:

12-66-11-77-121-71-22-87-85-115-44-1010-11

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

| Domains | | | | × |
|---|--|-------------|-------------------------|-------------------|
| Define | odify | Normal | | \sim |
| Туре | Membrane I | Plate Shell | | $\langle \rangle$ |
| Material | C25/30 | | ð | ~~ |
| Thickness [mm] = Eccentricity [mm] = | | | | |
| Local x reference | :e » | Auto | ~ | × |
| Local z reference | ie 💌 | Auto | \sim | |
| <u>Stiffness fa</u> | ictors | * | | |
| Color | ✓ By materi ✓ By materi | ai | tiffness redu k = [1 | |
| Pick up >> | | | OK | Cancel |



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

Reference

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

| Domair | 15 | | | | × |
|--------|---------------------------------|--------------|------------------------|-------------------------|--------|
| ۲ | Define | O Modify | Norm | al | \sim |
| | Туре | Membrane | Plate Sh | ell | |
| | Mate | erial C25/30 | | - (| |
| | Thickness (m Eccentricity (m | | ; | 1 | |
| | Local x re | ference » | × Auto | \sim | × |
| | Local z re | ference » | × Auto × R1 (0; -1 | 000:0) | |
| | Stiff | ness factors | | 1.000; 0) | |
| | Color | By mate | | Stiffness redu k = 1 | |
| P | ick up » | | | OK | Cancel |

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 x reference**, finally press **OK**.

| Domains | | | | × |
|---|------------------------|-------------|---------|--------|
| ● Define ○ N | lodify | Normal | | \sim |
| Туре | Membrane I | Plate Shell |] | Ì |
| Material | C25/30 | | ð | |
| Thickness [mm] = Eccentricity [mm] = | | | | |
| Local x referen | ice » | R3 | ~ | × |
| Local z referen | ice » | R2 | ~ | |
| Stiffness | factors | * | | ~ |
| Color | By materi By materi | ai - | k = 1.0 | |
| Pick up » | | | OK | Cancel |



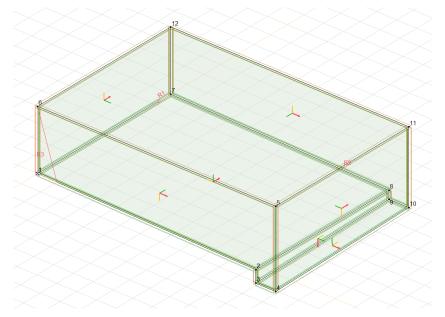
Repeat creating shell elements, activate again *Domain* function, then select lines between the given nodes:

3 - 9 3 - 4 4 - 10 9 - 10,

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

| ۵ | omain | | | | | | | × |
|---|-------|-------------------------------|-------------------------------|----------------------|--------------------------|---|----|-------------------|
| | ۲ | Define | ○ Modify | | Normal | | | > |
| | | Туре | Mem | brane | Plate <mark>Shell</mark> |] | | $\langle \rangle$ |
| | | Ma | terial C25/3 | 0 | | ð | | ~~ |
| | | Thickness [Eccentricity [| mm] = 250 mm] = 0 | ~ ~ | | | | |
| | | Local x | reference » | | Auto | ~ | | × |
| | | Local z | reference » | | Auto | ~ | < | |
| | | | rsion = 1.000 hear = 1.000 | ~ | | | | |
| | | Color | | / materi / materi | | | | |
| | Pi | ck up » | | | | (| DK | Cancel |

The following will be displayed:



Speed buttons

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

Line elements

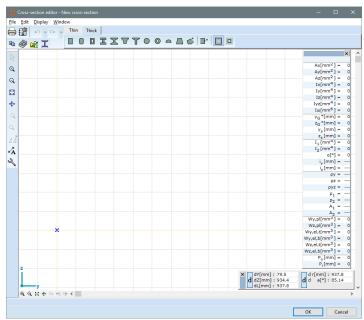
To define ribs on the upper edges, click on *Line elements* on the *Elements* tab. The *Selection palette* appears, click on the 4 edges and close the selection with *OK*, then *Line elements* dialog window appears:

| ine elements X | < |
|--|---|
| Define O Modify | |
| Truss Beam Rib | |
| Material properties Material C25/30 | |
| Cross-section Variable cross-section Cross-section 300x600 V T | |
| Local coordinate system Local x orientation Auto v Local z reference >> X Auto v | |
| End releases Startpoint Endpoint | |
| | |
| ColorStiffness reduction \checkmark By material $k_A = 1.000 \lor$ \checkmark By material $k_I = 1.000 \lor$ | |
| Pick up » OK Cancel | |

Cross-section

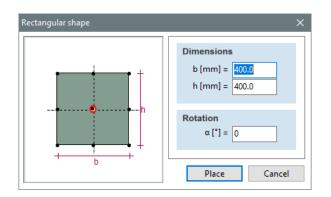


To define the cross-section of the rib, click on **Cross-section editor** icon. The following window shows up:

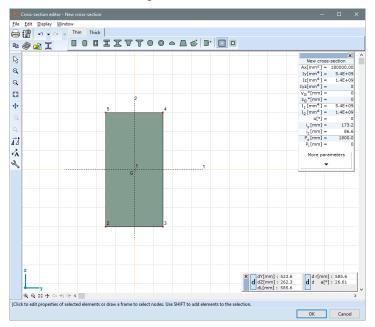


Rectangular shape

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.

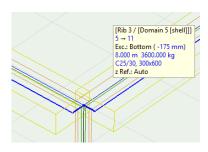
Click **OK** to close **Cross-section editor**, in the dialog window change default name of cross-section if required. Click on **OK** again to confirm.

| 2 | New cross-section Enter cross-section name. |
|---|--|
| | Name 300x600 |
| | Remember cross-section position |
| | <u>O</u> K <u>C</u> ancel |

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:





ß



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.

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.

Restores the previous view



22

ß

Zoom to fit

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

Restore previous view with icon between Zoom toolbar.

Wireframe

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

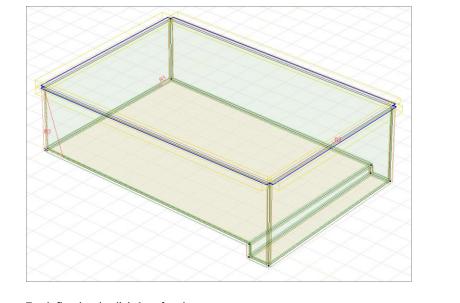
up:



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

| Surface support for dom | ains | | × |
|---|----------|----------------------|------------|
| Define | O Modify | | |
| | | Nonlinear parameters | \diamond |
| R _x [kN/m/m ²] : R _y [kN/m/m ²] : R _z [kN/m/m ²] : | = 1E+4 ~ | | |
| Pick up » | | | OK Cancel |

Change R_x and R_y to 1E+3, then click OK. The following will be displayed in the main window:



Loads

To define loads click **Loads** tab:

| Geometry | Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design | Steel design | Timber design | | |
|----------|--------------|----------|------|--------|----------|-----------|---------|--------------|--------------------|---------------|---------|-----|
| 표 + 분 | <u>።</u> ተ | <u>₩</u> | ₫ 🧃 | • 🚈 (| è 🖉 | 🛛 🏷 🕈 | ** 😤 🛙 | G 🛱 | . <u>Е</u> . Дат 🖟 | v 🚑 🛲 🚽 | した 東西 🚽 | 👋 💍 |

Load cases and load groups

Щ

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

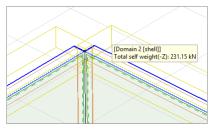
| M Load groups and load cases | | - | | × |
|------------------------------|--|--|--------|-----------------------------|
| ⊡• Ungrouped STI (⊖) | Load case Duplicate ST1 Conversion Load group Vingrouped | Add H Add Add Add Add Add Add Ad | ₩ - | |
| | Eurocode | | | |
| | Load group | | Add |))] } |
| × Delete | Critical load group combinations OI | K | Car | ncel |

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

Self weight

G

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.

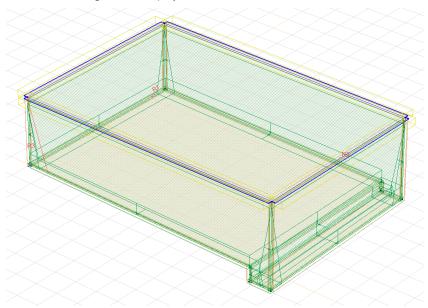
Fluid loads



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:

| Fluid load | × |
|---|--|
| Define OModify | |
| Direction of load variation | $p(Z_1) \rightarrow p(Z_2) \rightarrow p$ |
| $Z_1 [m] = 3.000$ $Z_2 [m] = -0.800$ | $p(Z_1) [kN/m^2] = 0$ $p(Z_2) [kN/m^2] = 0$ |
| Pick up >> | OK Cancel |

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 $p(Z_2)$ [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:



| MODEL DATA | 1 | × 🖻 💼 🏭 🖶 | ज मिल मह | | | | |
|---|------|---------------------------|-----------|-------------|-------|---------|--|
| Materials (1) | | | | | | | |
| Rebar steel grades (1) | Cust | om load combinations by l | oad cases | | | | |
| - Cross-sections (1) | | Name | Туре | SELF WEIGHT | WATER | Comment | |
| XLAM timber panels References (3) | | | | | | | |
| - Nodes (12) | | | | | | | |
| Elements | | | | | | | |
| E Loads | | | | | | | |
| SELF WEIGHT (11) | | | | | | | |
| WATER (7) | | | | | | | |
| - Load cases (2) - Critical load group combinations (1) | | | | | | | |
| Cutton load group combinations (1) Custom load combinations | | | | | | | |
| By load cases | | | | | | | |
| - By load groups | | | | | | | |
| Functions | | | | | | | |
| Weight report | | | | | | | |

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 cre | ate finit | e eleme | nt mesh o | change tal | o to Mesh | • |
|-------------------|--------|-----------|---------|-----------|------------|------------------|--------------|
| Geometry Elements | Loads | Mesh | Static | Buckling | Vibration | Dynamic | R. C. Design |
| \mathbb{Z} | | | | | × | | |

Speed buttons

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

Domain meshing

ΠΟ

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*.

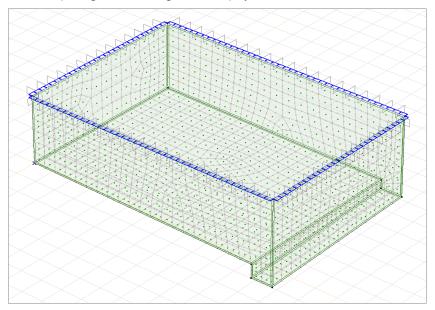
Steel design | Timber design

| Meshing parameters X |
|---|
| Define O Modify |
| Mesh type |
| Average mesh element size [m] = 0.60 ~ |
| Fit mesh to loads |
| $ $ Point loads $ \ge [kN] =$ 0 $ $ Line loads $ \ge [kN/m] =$ 0 $ $ Surface loads $ \ge [kN/m^2] =$ 0 |
| Adjust mesh to column heads (to enable cutting of moment peaks) |
| Contour division method ① Uniform mesh size 〇 Adaptive mesh size |
| Smoothing |
| Create mesh only for unmeshed domains Calculation of domain intersections Keep generated mesh guidelines if meshing fails |
| OK Cancel |

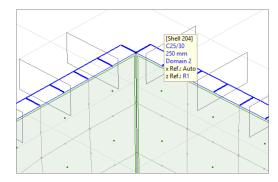
Click on **OK** to close dialog. Meshing process is can be followed on the status bar:

| Mesh generation | |
|-----------------|-------|
| 15% | |
| | |
| | Abort |

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.

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

| Geometry Elements Loads Mesh | Static Buckling Vibra | ion Dynamic R. C. Design Steel desi | ign Timber design | |
|------------------------------|-----------------------|-------------------------------------|---------------------------------------|--|
| | · | _ | → = ^{max} = ^{ty} F. | |



Static

۰u

Click on Linear static analysis to start analysis.

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

| 🚧 (x64) Linear analysis of reservoir.axs | | - 🗆 | × |
|--|---|--|--------------------|
| Stiffness matrix evaluation | | | |
| | 56% | | |
| | 18% | | |
| Close if completed | | Cance | I |
| Messages | | | « |
| Statistics | | | × |
| | 29.2 M 49.3 M 29.2 M 9.292 G 30 M 10.93 G 15.87 G Intel(R) Core(TM) i5-6500 CPU @ 3.20GHz (4x) | Truss Beam Rib Spring Gap Link Edge hinge Membrane | 68 |
| Single thread | 3192 MHz | Plate | 774 |
| Model optimization Model verification | 00:00 | Diaphragm | - |
| Analysis | 00:00 | Load case Combination | 2 |
| Result file generation | | | |

Messages, Statistics

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

| lateQ9 element strain evaluation | | |
|--|------------------|----|
| 58% | | |
| 82% | | |
| | | |
| Close if completed | Canc | el |
| Messages | | |
| 10:00:18 Rib element equilibrated load assembly | | - |
| 10:00:18 MembraneO8 element strain evaluation | | |
| 10:00:18 MembraneO8 element stress evaluation | | |
| 10:00:19 MembraneQ8 element equilibrated load evaluation | | |
| 10:00:19 MembraneO8 element equilibrated load assembly | | |
| 10:00:19 PlateQ9 element strain evaluation | | |
| Statistics | | |
| | Truss | |
| Number of equations 16668 | X4 Beam | |
| Equations memory 29.2 M Estimated memory requirement 49.3 M | Rib | |
| Solver block size 29.2 M | | (|
| Largest available memory block: 9.213 G | Spring | |
| Analysis block size 30 M | Gap | |
| Available physical memory: 10.84 G | Link | |
| Total physical memory: 15.87 G | Edge hinge | |
| CPU Intel(R) Core(TM) i5-6500 C | | |
| Single thread 3192 MHz | Plate | |
| Model optimization | 00:00 Shell | 7 |
| Model verification | 00:00 Diaphragm | |
| | Load case | |
| Analysis | 00:02 Combinatio | |

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:

| (x64) Linear analysis of reservoir.axs | | - 🗆 | × |
|---|----------------|-------------|-----|
| Analysis of reservoir.axs completed. | | | |
| | | | |
| | | | |
| Close if completed | | OK | |
| F | | | |
| Messages | | | |
| 10:00:45 Calculating surface support force combinations | | | 1 |
| 10:00:45 Saving surface support force combinations | | | |
| 10:00:45 Calculating smoothed surface support force combinations | | | |
| 10:00:45 Saving smoothed surface support force combinations | | | - 1 |
| 10:00:45 Calculating support element internal force combinations 10:00:45 Analysis of reservoir.axs completed. | | | |
| 2 1 | | | _ |
| Statistics | | | |
| Number of equations 16668 | V.A | Truss | _ |
| Equations memory 29.2 M | <u> </u> | Beam | - |
| Estimated memory requirement 49.3 M | | Rib | 68 |
| Solver block size 29.2 M | | Spring | - |
| Largest available memory block: 9.219 G | | Gap | _ |
| Analysis block size 30 M | | Link | - |
| Available physical memory: 10.85 G | | Edge hinge | _ |
| Total physical memory: 15.87 G CPU Intel(R) Core(TM) i5-6500 CPU @ | 2 20CH= (Av) | Membrane | |
| Single thread 3192 MHz | 9 5.200H2 (4X) | Plate | - |
| | | Shell | 774 |
| Model optimization | 00:00 | Diaphragm | - |
| Model verification | 00:00 | Load case | 2 |
| Analysis | 00:03 | Combination | n 1 |
| Result file size: 3.17 M. | 00:00 | | |

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.

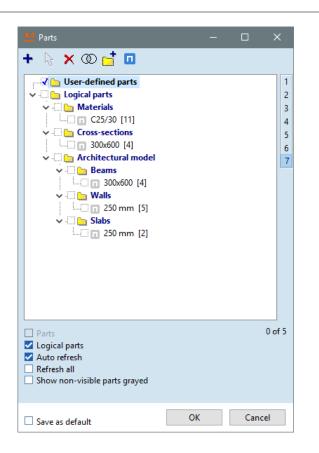
To see the result for the **WATER** load case, click on the dropdown button of the combo box displaying **SELF WEIGHT** and select **WATER**:

| 111 SELF WEIGHT | ▼ eZ [mm] |
|---------------------|-----------|
| 🖃 🕒 Linear analysis | |
| SELF WEIGHT | |
| WATER | |
| (+) Co.#1 | |
| 🕀 🛄 Envelopes | |

Select ey [mm] result component to display.



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:





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

| | New part definition |
|---|---|
| • | Part name |
| | Select from the entire model Turn off all the other parts |
| | Activate this part on all drawings in the Drawings Library |
| | <u>Q</u> K <u>C</u> ancel |

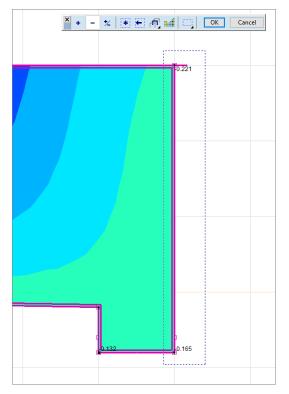


Change view to X-Z plane!

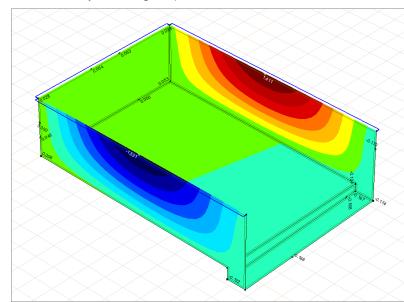


Press and hold wheel of the mouse, to pan the view, so the right part of model is fully visible. Use the bottom slider if your mouse does not have a wheel button.

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**.

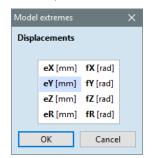


Restore the previous view with clicking the icon among **Zoom** toolbar – or activate **Perspective** view with **Ctrl+4** keys. Defining the part, the front wall is not visible in the next view:

Min, max values

🖛 min

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



Restore the previous view

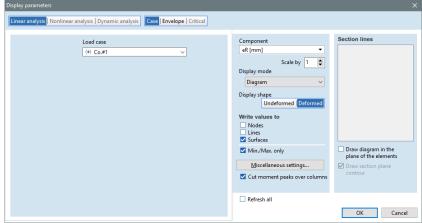
and its location. 7.53 Node 27 Case: WATER 0.456 eX [mm] = eY [mm] = -1.531 eZ [mm] = -3.168 0.00030 fX [rad] = fY [rad] = 0.00004 fZ [rad] = -0.00001 eR [mm] = 3.548 0.00030 fR [rad] = ОК

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

Max eY Node 1141 Case: WATER eX [mm] = 0.594 eY [mm] = 1.411 eZ [mm] = -3.151 fX [rad] = -0.00030 fY [rad] = 0.00003 0.00001 fZ [rad] = eR [mm] = 3.503 fR [rad] = 0.00030 OK

Close the window with **OK**.

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

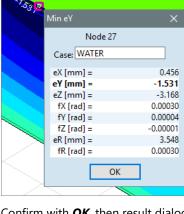


Click on Result display parameters icon, set Display shape to Deformed, Display mode to Diagram and Scale by to 2:

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

Close window with **OK**.





Hidden line removals

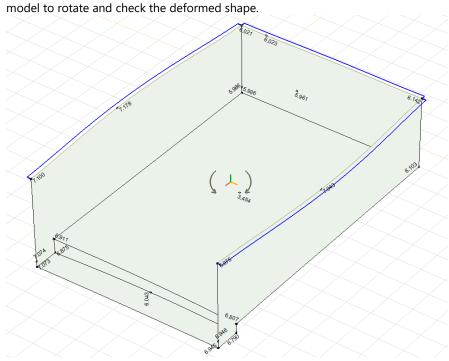


Go to the bottom right of the screen, turn on the **Mesh display** speed button and at the icon bar on the left change the **Display mode** to **Hidden line removal**:

Click on Rotate view icon in the Zoom toolbar at the bottom left corner of the main window. Drag the



Rotate



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

Press **Esc** to exit.

Restore the previous view



| Result dis | play |
|------------|------|
| parameter | s |

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**:

| | × |
|-----------------|-----------------|
| | mxD+ [kNm/m] |
| | 29.267 |
| | 27.874 |
| | 26.480 |
| | 25.086 |
| × | 23.693 |
| mxD+ [kNm/m] | 22.299 |
| | 20.905 |
| 29.267 | 19.511 |
| 27.177 | 18.118 |
| 25.086 | 16.724 |
| 22.996 | 15.330 |
| 20.905 | 13.937 |
| 18.815 | 12.543 |
| 16.724 | 11.149 |
| 14.634 | 9.756 |
| 12.543 | 8.362 |
| 10.453 | 6.968 |
| 8.362 | 5.575 |
| 6.272 | 4.181 |
| 4.181 | 2.787 |
| 2.091 | 1.394 |
| 0 | 0 |
| | 1/2 |
| 15 | 22 |
| | |

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





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:

| Section lines | × |
|---|-----------|
| Section segments Section planes Section lines | |
| 0 of 0 | |
| Ne <u>w</u> section segment | |
| New section segment group | |
| New section plane | |
| N <u>e</u> w section line | |
| <u>M</u> odify | |
| <u>D</u> elete | |
| Section lines Draw section plane contour | |
| ☐ Refresh all ✓ Auto refresh | OK Cancel |

150

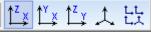
To define a new section plane, click on **New section plane** button in the dialog. The following message box will be displayed to enter name of the section:

| | New section pla | ne | | |
|---|--------------------|----|------------|----------------|
| • | Section plane name | | | |
| | | | OK | Canaal |
| | | | <u>U</u> K | <u>C</u> ancel |

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



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



Specify a section plane in the middle of the reservoir.

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

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.



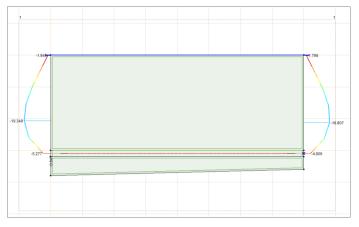
| t^zx t^yx t^zy 大 技

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

Numbering

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

The result is the following:

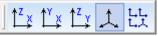


Speed buttons

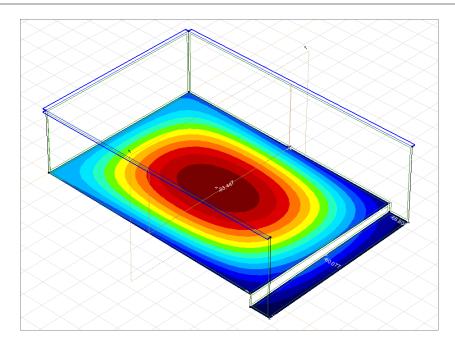
Turn off Section Line Display mode.



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**:

| Geometry Elements Loads Mesh | Static Buckling Vibration Dynamic | R. C. Design | Steel design Timber design | |
|------------------------------|-----------------------------------|--------------|----------------------------|----------------|
| 🌘 🧾 🛄 🔛 📰 (+) Co.#1 | ▼ axb [mm2/m] | ▼ None | ~ 1 | 📲 🖬 🖪 Tr 😔 🥪 🌋 |

Reinforcement parameters

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

| Surface reinforcement parameters (Eurocode) | × | | |
|---|---|--|--|
| Materials Reinforcement Cracking Shear | | | |
| Materials | | | |
| Concrete C25/30 V | | | |
| Maximum aggregate size [mm] = 30 🗸 🗸 | | | |
| Rebar steel B500B 🗸 | | | |
| | | | |
| Structural class S3 🗸 | | | |
| Exposition class Top surface | | | |
| XC2 Humid, seldom dry 🗸 🗸 | | | |
| | | | |
| | | | |
| XC2 Humid, seldom dry | | | |
| Bottom surface | | | |
| Coefficient for seismic forces $f_{se} = 1$ | | | |
| Nonlinear analysis | | | |
| Take into account concrete tensile strength | | | |
| • f_{ctm} \bigcirc $f_{ctm,fl}$ ϵ_{cs} [%] = 0.473 | | | |
| Nonlinearity | | | |
| ○ε-N (Wall)κ-M (Slab)ε-N;κ-Μ | | | |
| | | | |
| Set current settings as default | | | |
| Pick up >> OK Cancel | | | |

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:

Speed buttons

| Surface reinforcement parameters (Eurocode) |
|---|
| Materials Reinforcement Cracking Shear |
| Calculate with actual thickness Thickness (h) [mm] = 250 |
| Unfavorable eccentricity (N > 0) = $0 $ \checkmark h |
| Unfavorable eccentricity (N < 0) = 0 |
| Concrete cover Diameter (mm) Direction |
| $c_{T} \text{ [mm]} = 30 \checkmark \geq 30 \qquad \qquad \emptyset = 12 \checkmark \boxed{x \mid y}$ |
| z h |
| $\emptyset = 12 \checkmark x y$ |
| $c_{B} \text{ [mm]} = 30 \checkmark \geq 30 \qquad \qquad \emptyset = 12 \checkmark \boxed{x \mid y}$ |
| Apply minimum cover |
| Load transfer Two-way slab One-way slab |
| O In local x direction O In local y direction |
| Take into account the required Top reinforcement minimum reinforcement Bottom reinforcement |
| Reinforcement directions ● Local x, y ○ Custom |
| Set current settings as default |
| Pick up » OK Cancel |

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**.

be displayed:

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 | Click on Actual reinforcement icon, the following window shows up: | |
|---------------|--|----------|
| reinforcement | Actual reinforcement | \times |
| | Parameters (Eurocode) Reinforcement | - |
| | Min. thickness (h) [mm] = 250 | |
| | Primary direction of reinforcement Top surface $c_T [mm] = 30 \rightarrow \geq 30$ $v \land y$ $v \land y$ Bottom surface $c_B [mm] = 30 \rightarrow \geq 30$ | |
| | Apply minimum cover Use this rebar steel and concrete cover by default | |
| | ✓ Auto refresh | |
| | Pick up >> k_0 m | |

Selection

6

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:

| Actual reinforcement | | | × |
|-----------------------|--|---|--------|
| Parameters (Eurocode) | Reinforcement | | |
| | | Min. thickness (h) [mm] = 250 | |
| | Primary direction of re Top surface | c _T [mm] = 30 r_{T} [mm] = 30 r_{T} c _B [mm] = 30 | |
| Use this rebar steel | l and concrete cover by default | Apply minimum cover | |
| Auto refresh | | | |
| Pick up » | | ОК | Cancel |

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:

| Parameters (Eurocode) Reinforcement | *# | Rebars Type Ribbed \checkmark Ø [mm] = 16 \checkmark Spacing [mm] = 200 \checkmark Rebar position [mm] = 30 \checkmark As [mm²/m] = 1005 Image: Calculate rebar positions Image: Calculate rebar positions Add Delete Maximum of calculated reinforcement for the selected elements | |
|---|----|---|--|
| Min. thickness (h) [mm] = 250 c_{T} [mm] ≥ 30 c_{B} [mm] ≥ 30 Auto refresh Pick up >> | | axt [mm ² /m] = 914 axb [mm ² /m] = 510 ayt [mm ² /m] = 362 ayb [mm ² /m] = 414 OK Cancel | |

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:

| Actual reinforcement | × |
|--|---|
| Parameters (Eurocode) Reinforcement | |
| x Direction Top reinforcement (P) = 754 12 mm / 150 mm (36 mm) [R] Bottom reinforcement (P): None y Direction Top reinforcement: None Bottom reinforcement: None | Rebars Type Ribbed \checkmark Ø [mm] = 12 \checkmark Spacing [mm] = 150 \checkmark Rebar position [mm] = 36 \checkmark A _s [mm ² /m] = 754 Calculate rebar positions <u>Add</u> <u>Delete</u> Maximum of calculated reinforcement |
| $\begin{array}{c c} \mbox{Min. thickness (h) [mm] = } & 250 \\ c_{T} \mbox{ [mm] } & \geq 30 \\ c_{B} \mbox{ [mm] } & \geq 30 \end{array}$ | for the selected elements axt [mm²/m] = 914 axb [mm ² /m] = 510 ayt [mm ² /m] = 362 ayb [mm ² /m] = 414 |
| Auto refresh | OK Cancel |

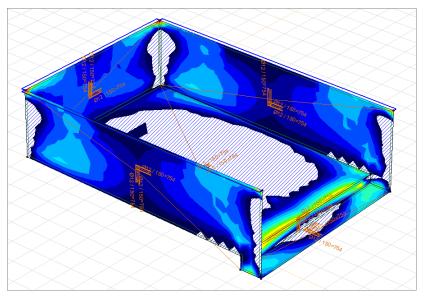
Specify the other reinforcement layers by using the above method.

Finally, the following will be the result:

| Actual reinforcement | | × | | | |
|-------------------------------------|---|-------------------------------------|--|--|--|
| Parameters (Eurocode) Reinforcement | | | | | |
| | | Rebars | | | |
| 🖃 x Direction | | Type Ribbed V | | | |
| Top reinforcement (P) = 754 | | Ø [mm] = 12 🗸 | | | |
| 12 mm / 150 mm (36 mm) [R] | æ | | | | |
| Bottom reinforcement (P) = 754 | | Spacing [mm] = 150 🗸 | | | |
| | | Rebar position [mm] = 48 🗸 🗸 | | | |
| Top reinforcement = 754 | | $A_{e} [mm^{2}/m] = 754$ | | | |
| 12 mm / 150 mm (48 mm) [R] | | 2. | | | |
| Bottom reinforcement = 754 | | | | | |
| 12 mm / 150 mm (48 mm) [R] | | Calculate rebar positions | | | |
| | | Add <u>D</u> elete | | | |
| | | Maximum of calculated reinforcement | | | |
| | | for the selected elements | | | |
| | 1 | axt [mm ² /m] = 914 | | | |
| Min. thickness (h) [mm] = 250 | | axb [mm ² /m] = 510 | | | |
| c _T [mm] ≥ 30 | | ayt [mm ² /m] = 362 | | | |
| c _B [mm] ≥ 30 | | ayb [mm ² /m] = 414 | | | |
| | | | | | |
| ✓ Auto refresh | | | | | |
| | | | | | |
| Pick up >> OK Cancel | | | | | |

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:

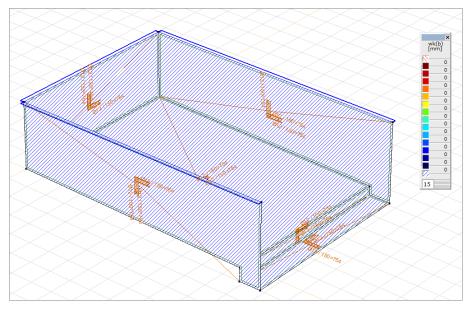


The thick brown lines and their titles indicates the given actual reinforcement. If only one direction of reinforcement is visible on the screen, then open **Display options**. On **Label** tab switch off **According to the displayed result component** function.

| Display options X | | | | |
|---|---|--|--|--|
| Symbols Labels Switches | | | | |
| Numbering 12 Truss Beam Rib Virtual beam Surface Domain Domain Support Links Rigid Diaphragm Spring Gap Material Cross-section Design optimization group Load panel Reference Use finite element numbers Story center of gravity Story shear center Labels on lines seen from axis direction Transparent labels Prevent labels from overlapping | Properties ABC Nodal coordinates Material name Cross-section name Bolted joint Column reinforcement Beam reinforcement Beam reinforcement Beam length Thickness Domain area Stiffness reduction COBIAX labels Load value Concentrated Line Surface Temperature Fire Self weight Other Mass value Units V Actual reinforcement Suppose Y axb Axb Y ayb Ayb Y ayt Y ayt Labels Rebars + Reinforcement values Rebars + Quantity x (Length) According to the displayed result | | | |
| ✓ Auto refresh Refresh all | component | | | |
| Save as default | OK Cancel | | | |

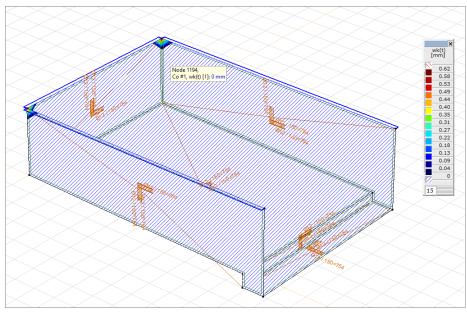
Crack width

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 co-ordinate 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:

| Narning | | × |
|---|--|------------|
| combina combina Therefor recomm | ement amounts must be calculated form ULS tions, while cracking analysis requires SLS tions. e to perform a full check of reinforced beams it is ended to select an apropriate envelope or critical tion from the list. | |
| 🗌 Do no | ot display this message 0 | K |
| Rib internal forces Selected rib elements are connected to shells with an eccentricity. In such cases rib design moments will be calculated as Mdesign = M + N * exc. | | |
| | | <u>O</u> K |

Beam By closing warning message, the **Beam reinforcement parameters** window shows up. reinforcement parameters Set the parameters as shown in the next figures: Image: Set the parameters - Eurocode X Image: Cross-section Parameters Parameters

| Cross-section Parameters |
|---|
| Concrete C25/30 V D _{max} [mm] = 16 V Structural class S3 V Vz - My |
| $\begin{array}{c} 300x600 \\ h \\ \\ \end{array} \\ \begin{array}{c} b_w \\ h \\ \\ \end{array} \\ \begin{array}{c} b_w \\ b_w \\ \\ b_w$ |
| Environment classes, concrete covers Apply minimum cover |
| $\begin{array}{c c c c c c c c c c c c c c c c c c c $ |
| Stirrup B500B \checkmark Longitudinal rebars B500B \checkmark Stirrup legs = 2 $$ Type Ribbed \checkmark $\emptyset_s [mm] = 8 \checkmark \emptyset_t [mm] = 16 \checkmark$ |
| |
| Use this rebar and stirrup steel by default OK Cancel |

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

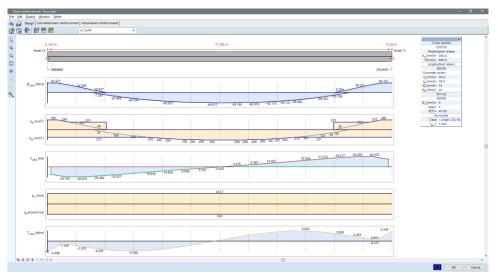
Use XC2 Environment class on all four sides of the rib and check Apply minimum cover.

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:

| Beam reinforcement parameters - Eurocode | × | | | | |
|--|--|--|--|--|--|
| Cross-section Parameters | | | | | |
| Design internal forces Vz - My Vy - Mz Shear force reduction at supports | Angle of the concrete compression strut • 45° • Variable • Custom $\theta = 45^{\circ}$ 22° 45° | | | | |
| Cracking ✓ Increase reinforcement according to limiting crack width Top crack width [mm] = 0.30 Bottom crack width [mm] = 0.30 ✓ Take into account concrete tensile strength Load duration | | | | | |
| Check allowed deflection Beam: L / 300 Deflection check will be performed only if the actual concrete grade and cross-section is set. Cantilever: L / 400 | | | | | |
| Nonlinear analysis Image: Take into account concrete tensile strength Image: fctm fctm,fil ϵ_{cs} [%] = 0.422 | | | | | |
| Coefficient for seismic forces | f _{se} = 1 | | | | |
| | | | | | |
| Use this rebar and stirrup steel by default | OK Cancel | | | | |

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