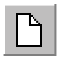
1. PLATE MODEL

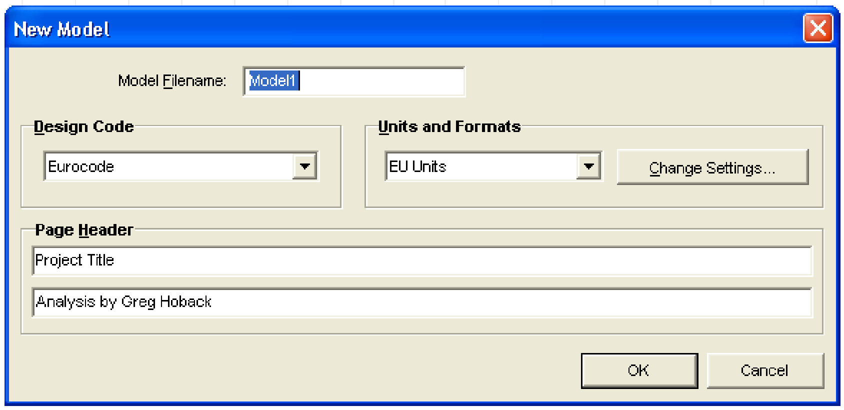
Start

Start AxisVM by double-clicking the AxisVM icon in the AxisVM folder, found on the Desktop, or in the Start, Programs Menu.

New

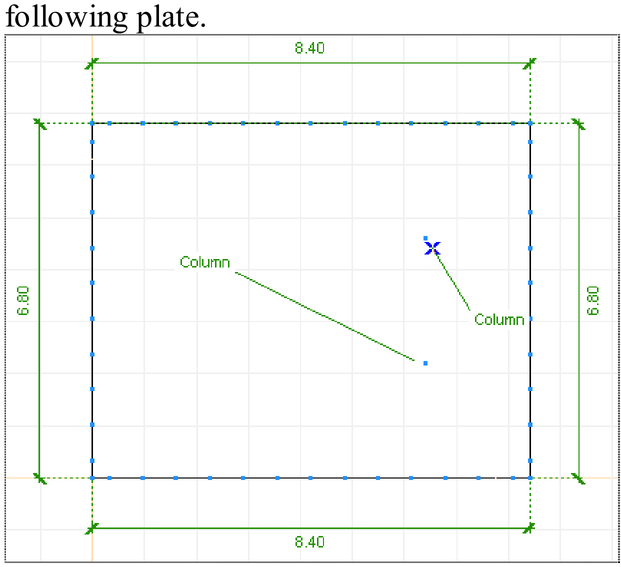


Create a new model with the New Icon. In the dialog window that pops up, **replace the Model Filename** with "Plate", and in the Design Code panel **select Eurocode**.



Objective

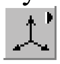
The objective of the analysis is to determine the maximum deflection, bending moments and required reinforcement of the following plate.



Lets suppose the plate thickness is 20 cm, the concrete is of C20/25, and the reinforcement is computed according to Eurocode-2.
The first step is to create the geometry of structure.

Coordinate

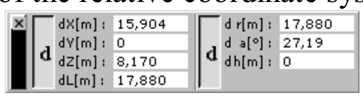
In the lower left corner of the graphics area is the global

System


coordinate system symbol. The positive direction is marked by the corresponding capital letter (X, Y, Z). The default coordinate system of a new model is the X-Z coordinate system. It is important to note that unless changed the gravity acts along the – Z direction.

In a new model, the global coordinate ***default location*** of the cursor is the bottom left corner of the graphic area, and is set to X=0, Y=0, Z=0.

The location of the cursor is defined as a relative coordinate. You can change to the relative coordinate values by pressing the 'd' labeled button on the left of the Coordinate Window. (Hint: In the right column of the coordinate window you can specify points in cylindrical or spherical coordinate systems). The origin of the relative coordinate system is marked by a thick blue X.

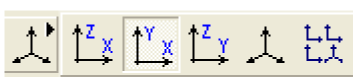


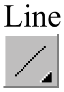
Geometry

If not already selected, **activate** the Geometry tab. Under it appears the Geometry Toolbar.

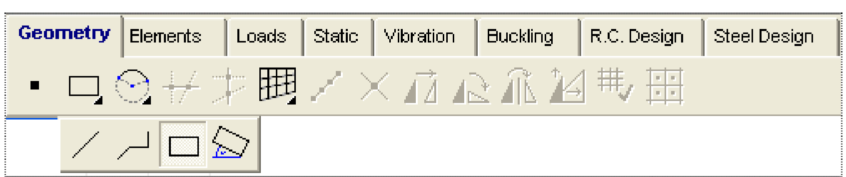
View


Click the Y-X view from the View Icon Bar.

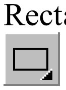


Line


Create the geometry of plate using the **Rectangle** command. **Holding down the left mouse button** on the Line Icon can access it.



Note: When the a line type is chosen, the Relative coordinate system automatically changes to the local system ('d' prefix)

Rectangle


The corners of the rectangle can be specified graphically or by entering the coordinates. Lets **enter them** with coordinates:

Set the first corner (node) of the rectangle by **typing in these entries**:

X=0

Y=0
Z=0

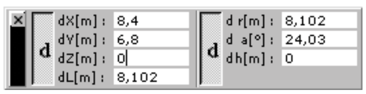
Finish specifying the first corner point by **pressing Enter**. The first node of the plate model is now also the global coordinates origin point.

Relative Coordinates

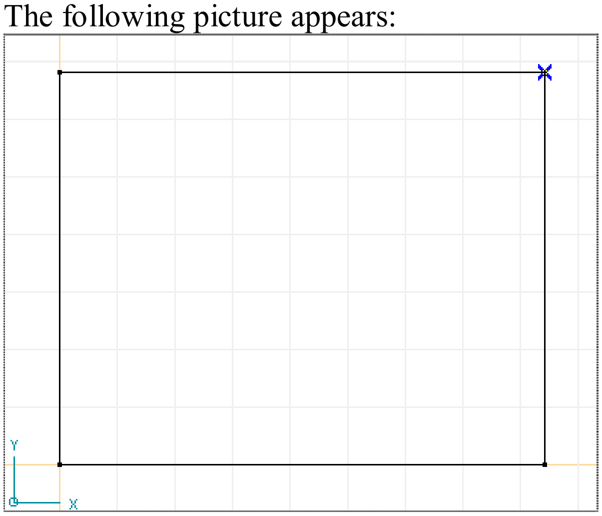
Lets specify the relative coordinates of the next corner. **Type in** the following sequence of keys:

X=8.4
Y=6.8
Z=0

Finish specifying the second corner point by **pressing Enter**. (Note: If the decimal separator on the computer is set to comma, then instead of the 'dot' you have to uses the comma.)



The following picture appears:



Lets move the relative origin to the lower left corner of the rectangle. For this **move the cursor** over the lower left node and **left Click**.

Exit from rectangle line command by pressing **Esc**.

Node Icon

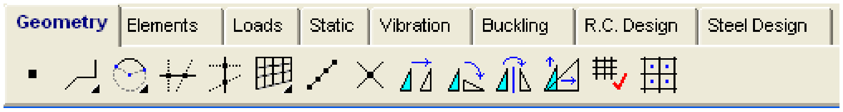
Click the Node Icon, then **type in** the following sequence:

X=6.4
Y=2.2, Enter
X=0
Y=2.4, Enter

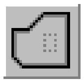
These nodes have added columns to support the plate. **Exit** from the Node command with **Esc**.

Elements

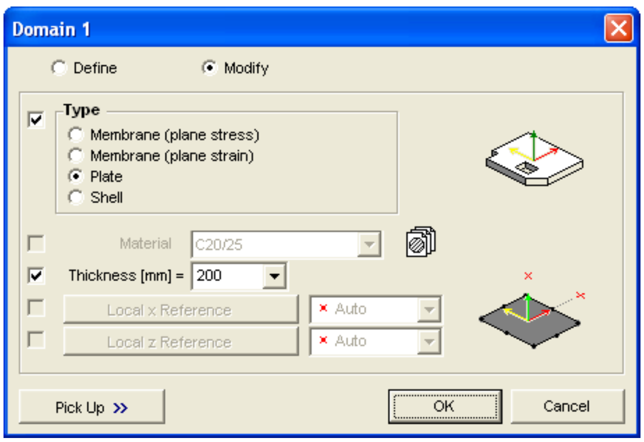
The next step is to define the finite elements. **Click the Elements tab.**



Domain

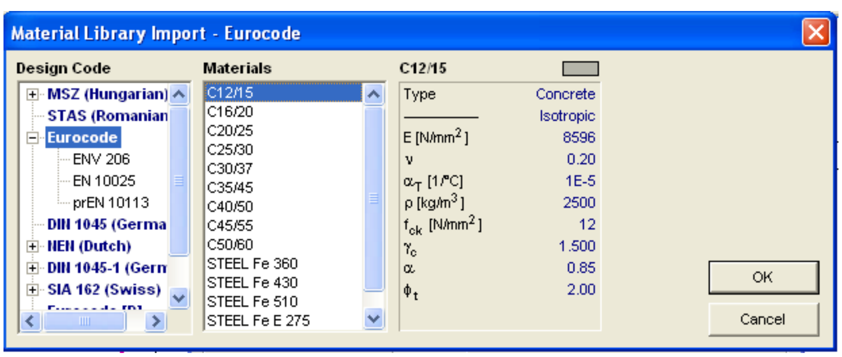


Click the Domain Icon, then click on one line and the whole rectangle will be selected. Finish the selection with **Ok**, and the following dialog window appears.



Material Library Import

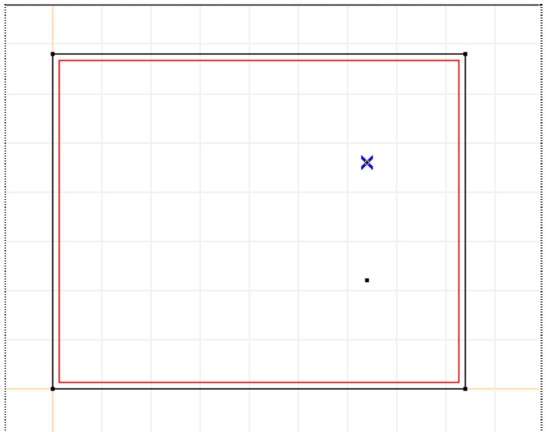
Click the Material Library Import Icon in the row of Material, and the following dialog window appears:



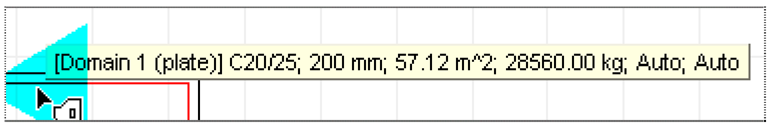
Choose C20/25 from Materials List Box, using the scroll bar if necessary. Close the Material Library Import dialog window with **Ok**.

Thickness

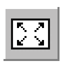
Type in the thickness combo box the value 200 [mm], then close the dialog window with **Ok**. The following picture appears:
Note the red line on the inner contour of the domain



This is the symbol of a (plate) domain. If you move the mouse on this contour, the properties of the domain will appear in a hint window.

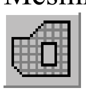


Zoom to Fit

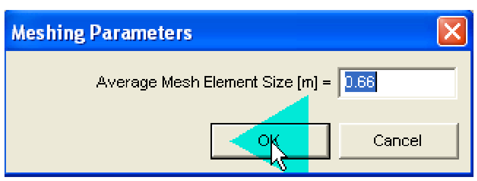


For a better view let's **click** the Zoom to Fit Icon on the Zoom Icon bar.

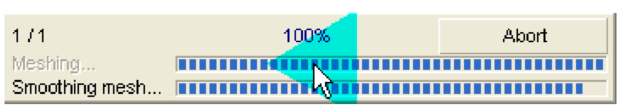
Domain Meshing



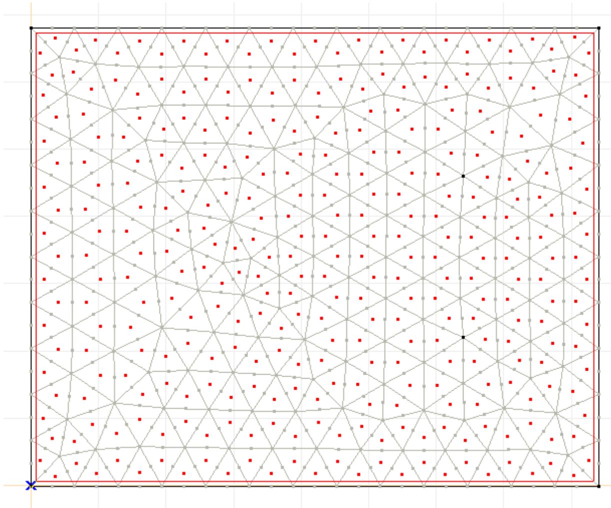
Click the Domain Meshing Icon. Use the select All command (the asterisk) and finish the selection with **OK**. The following dialog window appears:



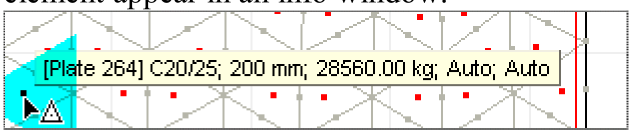
Type in the Average Mesh Element Size edit box the value 0.66 [m], then press **Ok**. An automatic mesh generation will start. Its progress is showed in the following window.



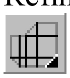
When the mesh generation finishes, the following picture appears:



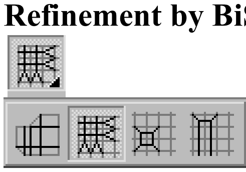
The surface element symbol is a solid red square in the center of the element. If you move the cursor over it, the properties of the element appear in an info window.



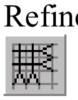
Refinement



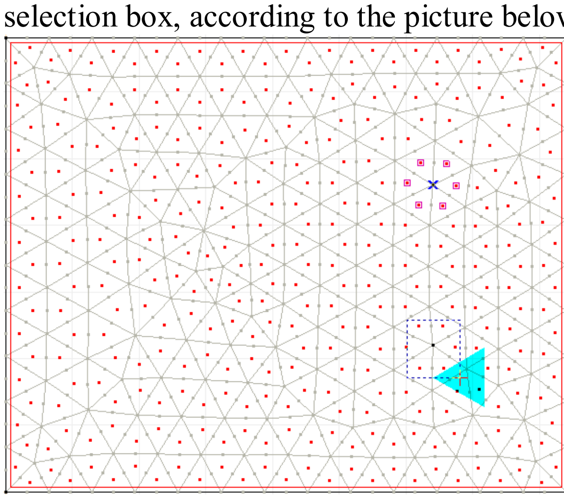
Let's refine the mesh around the two nodal supports. **Depress the left mouse button** over the Refinement Icon, and **click the Refinement by BiSection Icon** that appears.



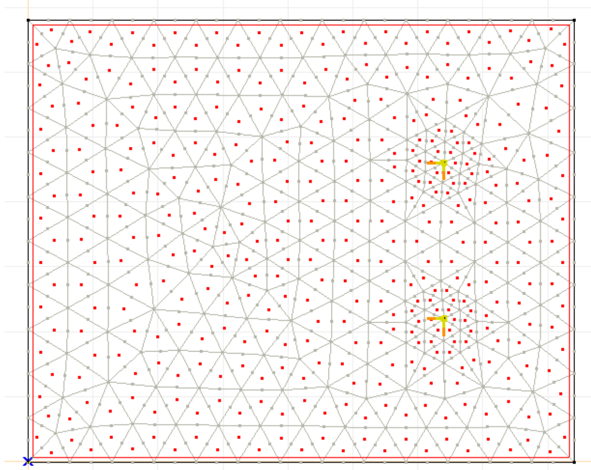
Uniform Refinement



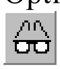
Select the surface elements around the nodal supports with a selection box, according to the picture below:



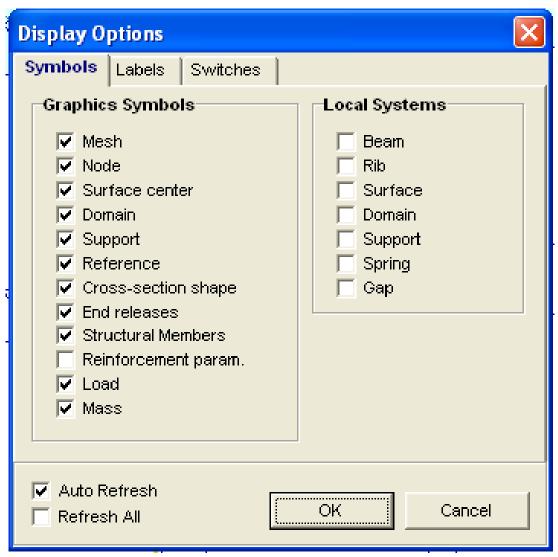
Finish the selection with **Ok** and **accept** the offered Maximum Side Length. The result of the refinement is shown in the following picture:



Display Options

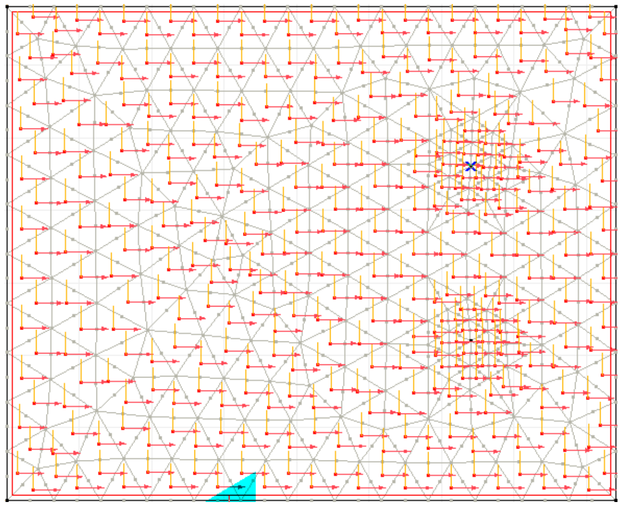


Let's view the local coordinate system of the surface elements. **Click the Display Options Icon** in the Icons Menu (left side).

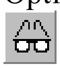


Activate the Symbols tab, then on the Local System Panel **check the Surface box.**

Close the dialog window with **Ok**.
A red line shows the local -x direction, a yellow line the local -y direction and a green line the local -z direction:

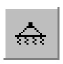


Display Options

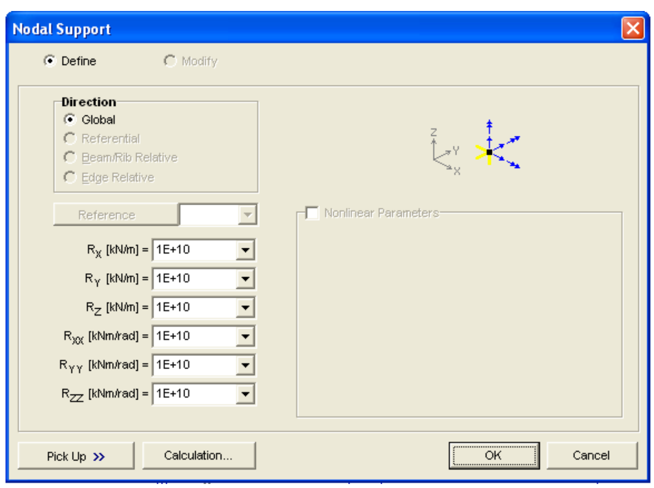


Select Display Options Icon to **switch off the Surface box** on the Local System panel.

Nodal Support

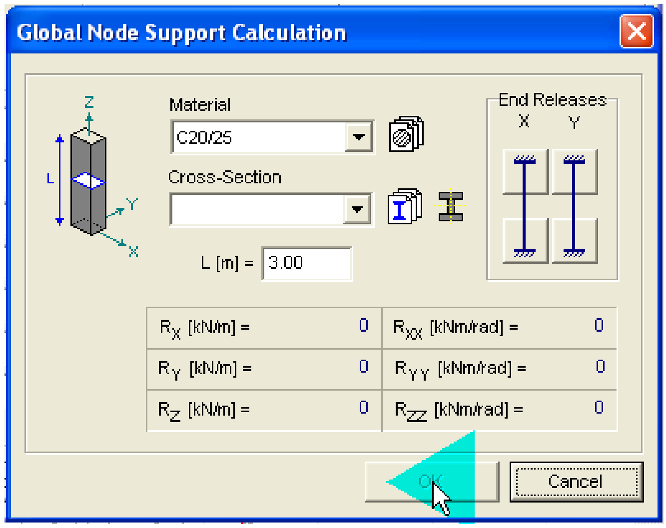


Let's specify the supports of the structure. **Click on the Nodal Support** command then select the two nodes in the center of the columns and finish the selection with **Ok**. The Nodal Support Window appears.



Calculations

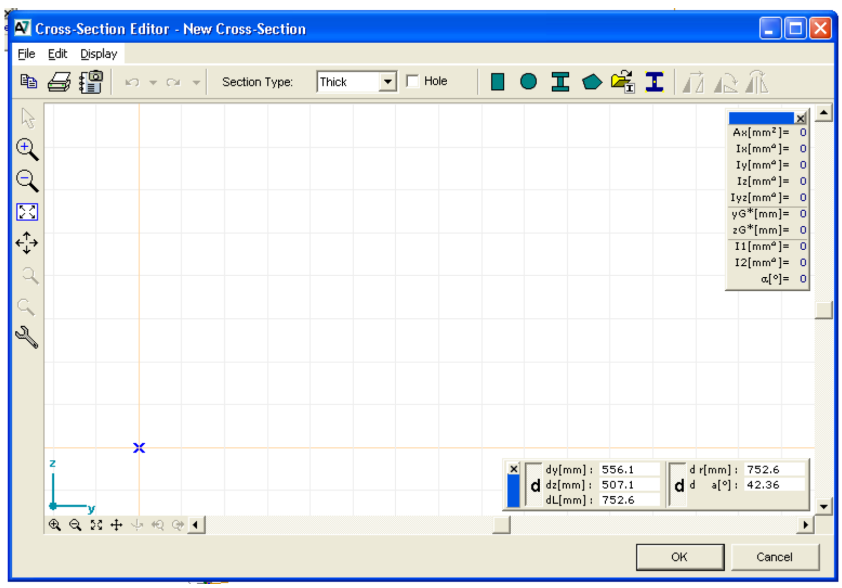
Click the Calculations button. The following dialog window appears:



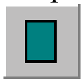
In this dialog window you can specify the support stiffness for the column type support.

New Cross Section

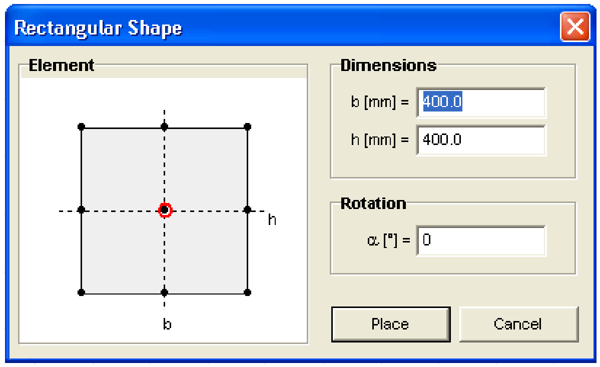
Click the New Cross Section Icon. The following dialog window appears:



Rectangle Shape

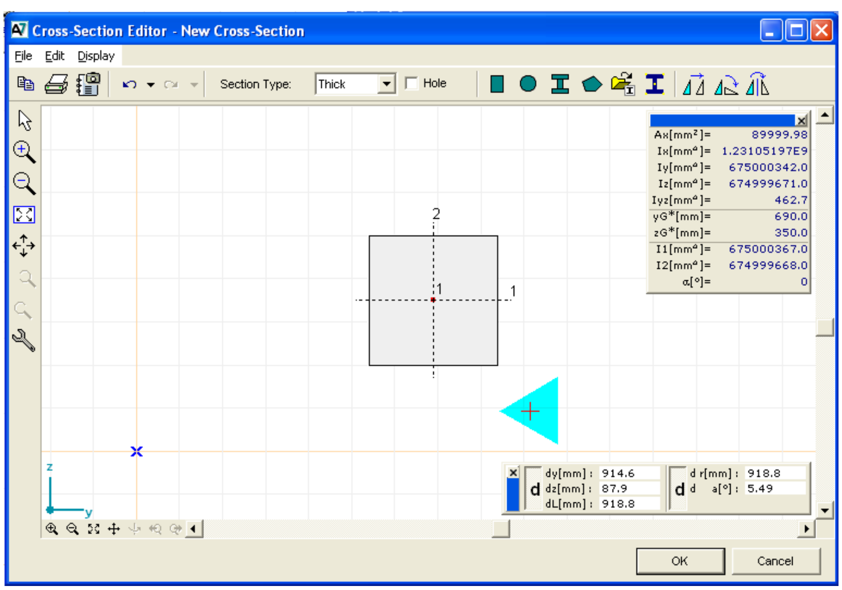


Click the Rectangular Shape Icon. The following dialog window appears:

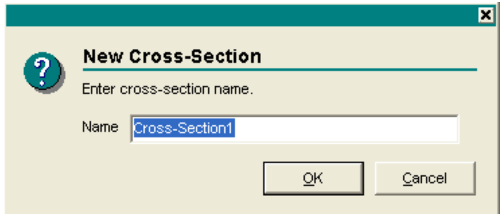


Type 300 [mm] in the upper two edit boxes, as the dimensions of cross section, and click Place. **Click** in the Cross Section Editor Drawing Area to place the rectangle. The location where the rectangle is placed is unimportant.

A following picture appears:

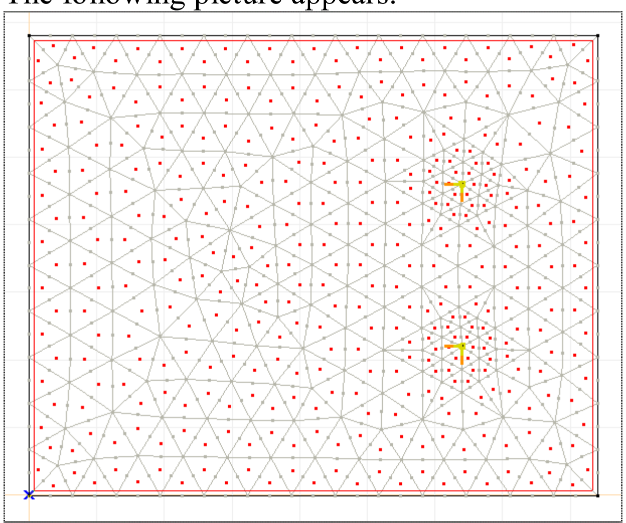


Close the Cross Section Editor with **Ok**. A dialog window asks for the name of the new cross-section.

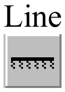


Type in 300x300, then close the dialog window with **Ok**. The Global Node Support Calculation dialog window's stiffness values will take into account this cross-section's properties. Accepting the remaining settings click **Ok**. The stiffness values displayed in the Global Node Support Calculation dialog window will be copied in the Nodal Support dialog window. Close the dialog window with **Ok**, and the two supports are created.

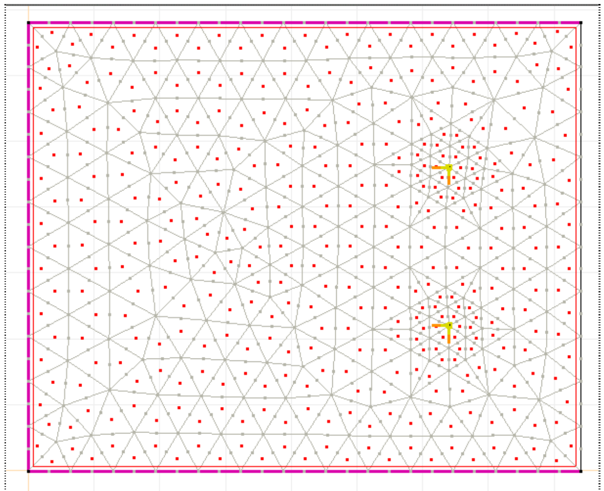
The following picture appears:



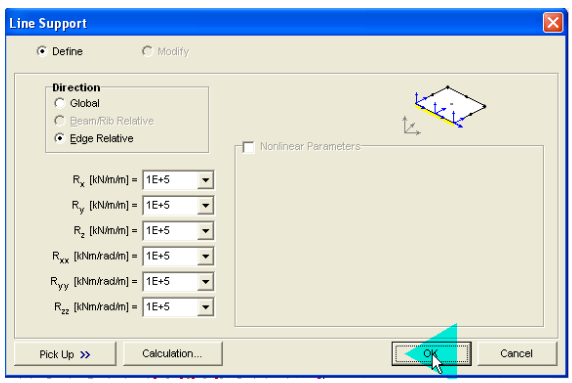
Line Support



Let's create the line supports on the contour of the domain. **Click the Line Support Icon**, and select the four contour lines of the domain. They represent walls on the edges of the plate.

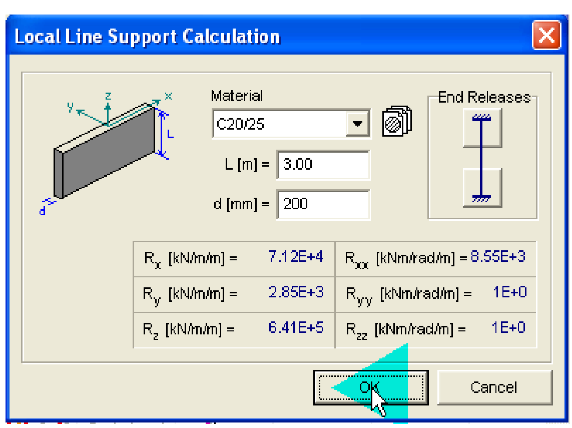


Finish the selection with **Ok**, and the following dialog window appears.

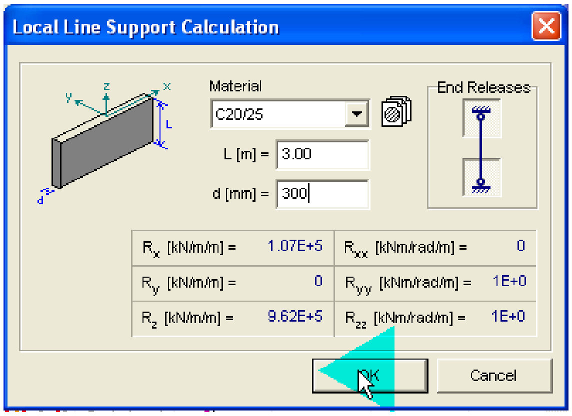


Calculation

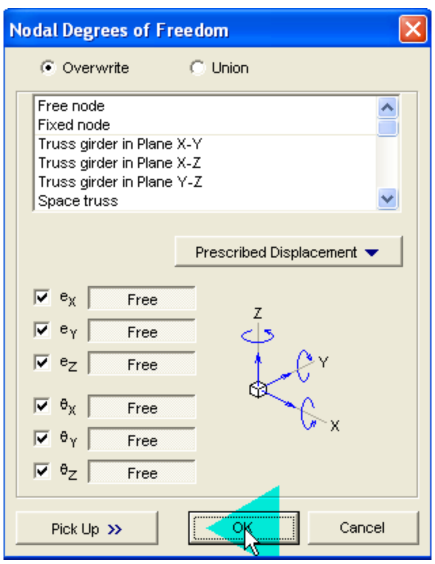
Click the Calculation button. Here you can calculate the line support stiffness due to a wall support. **Type in** the thickness of wall edit box 300 [mm]. You can see that the height of the wall is 3.0m, and the wall stiffness is also shwon in this dialog box.



Depress both the upper and lower End Release Icons. Close with **Ok** the dialog windows.



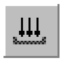
Nodal DOF 	**Click the Nodal DOF Icon**. **Select all nodes** with the All command (the asterisk), then finish the selection with **Ok**. In the Nodal Degrees Of Freedom dialog box **select Plate in X-Y** from the list.



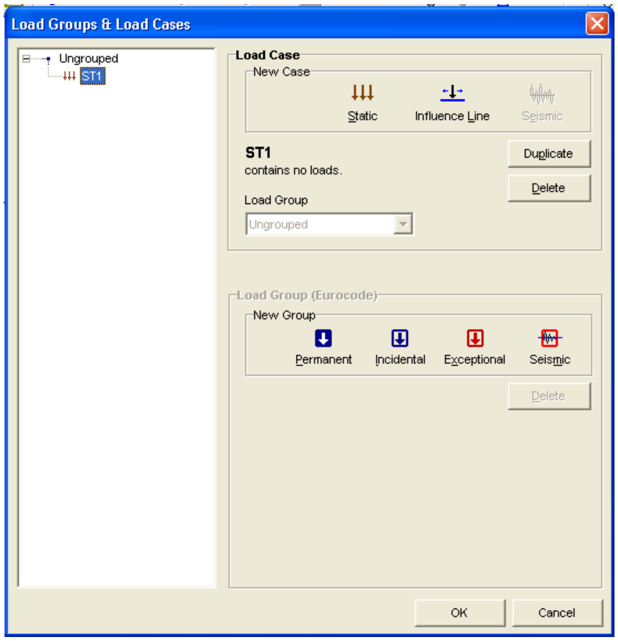
Accepting this will constrain the degree of freedom to vertical displacements and rotations about axes in the plane of the plate.

Loads	The next step is to apply the loads. **Click the Loads tab**.

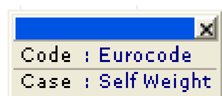
Load Cases & Load Groups



It is useful to group the loads into load cases. To manage the load cases **click the Load Cases & Load Groups Icon**. The following dialog window appears:

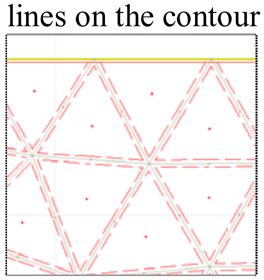


ST1 in the upper left corner of the window is the first load case (created by default). **Click it and rename it** to Self-Weight. Closing the dialog window it will be the active load case. It can be seen on the Info Window:

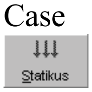


Self Weight

Click the Self Weight Icon, and **select all elements** with the All command. Finish the selection with **Ok**, and the self-weight load will be applied to all elements. This can be seen by the red dashed lines on the contour of elements.



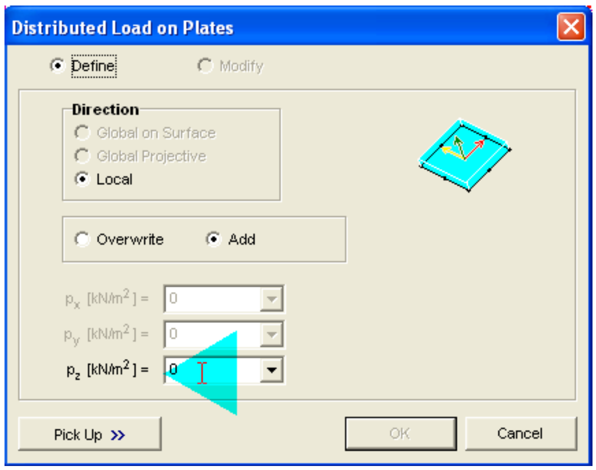
New Load Case



Click the Load Cases & Load Groups Icon again, and **create** a new load case with the Static Icon. **Name it Permanent Load**. This load case contains the dead loads on the plate. Let's assume it is 2.5 kN/m2 distributed load.

Distributed Surface Load

Click the Distributed Surface Load Icon and **select all elements** with the All command. Finish the selection and **type in** the -pz input box the value -2.5 kN/m2. The negative value means a load acting in opposite direction to the local z-axis of the surface element. This is a load on the surfaces of the plate.

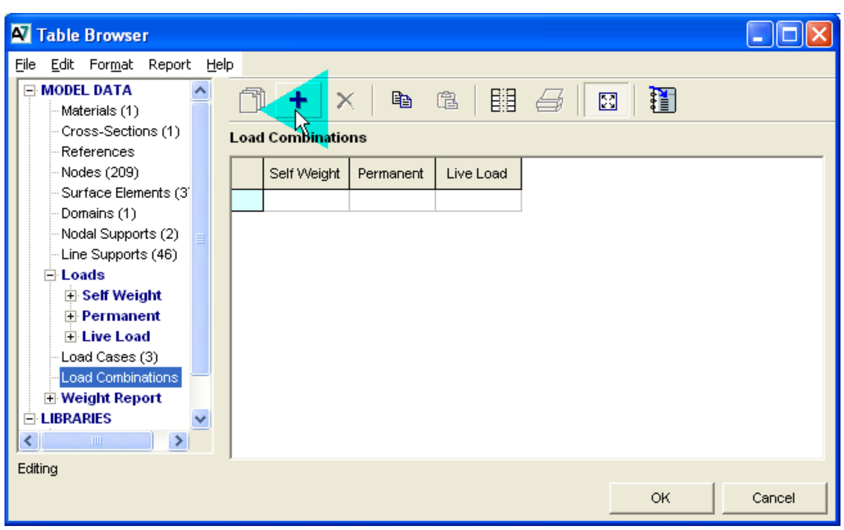


New Load Case

Create a new load case and **name** it **Live Load.** It will contain the variable loads. **Click the Distributed Surface Load Icon** and **select all elements** with the All command. Finish the selection and **type in**
-pz=-1.5 kN/m2.

Load Combinations

Now, that all loads have been applied to the structure, the load combinations can be created. There will be only one load combination, containing all load cases. **Click the Load Combinations Icon**. The following dialog window appears:



New Row

Create a new load combination by using the **New Row** command. You can apply load factors to load cases by using a load combination. In this example the factors of the Eurocode2 will be used:

Self Weight	1.35
Dead Load	1.35
Variable	1.50

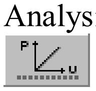
Type in these values in their columns. You can move to the next column by pressing Enter. When finished press **Ok**, and the new load combination is created.

Now all the model data is available for the analysis.

Static

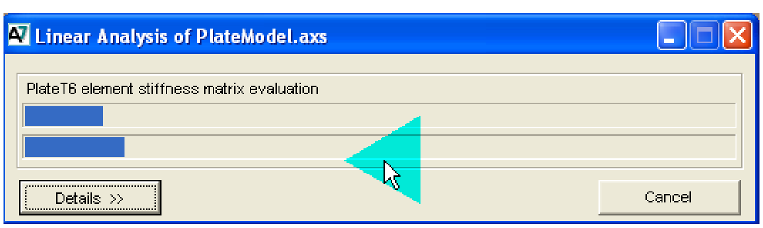
The next step is the analysis and postprocessing. **Click the Static tab.** Here you can start the static analysis and visualize the results.

Linear Static Analysis



Click the Linear Static Analysis Icon. If till this point the model wasn't saved, the program will ask to save. **Accept Save**, and a Save dialog window appears, where you can specify the model file name and path.
The analysis process will start.

During the analysis the following window appears:

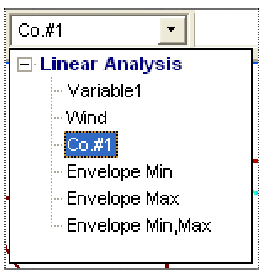


If you click the details button, the topmost bar shows the progress of the current computation step. The bar below it shows the global analysis progress. The estimated memory requirement is the amount of virtual memory that must be available. If the size of the operating systems virtual memory is limited to a lower value, an error message will appear, showing the required virtual memory. When the analysis has finished, the progress bars will disappear.

Static

Closing the Linear Analysis window with **Ok** the postprocessor will start by default with the first load case (Self-Weight in this case), the result component is ez displacement and the display mode is isosurface 2D. This shows the vertical displacements from the first load case.

Click the Case Selector combo box, and **select Co.#1** to view the results from the load combination.

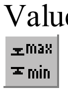


The Color Legend Window shows that the displacements are negative, because they are in an opposite direction with the local z-axis of the elements. This is the top view of a surface load.

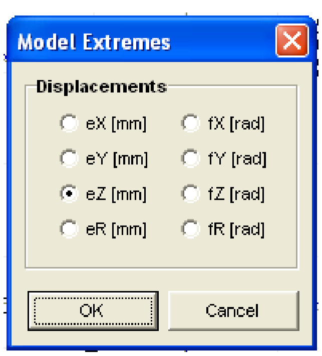
Display Options

Click the Display Options Icon on the Icons Menu in the left side. Under the Symbols tab, in the Graphics Symbols Panel **switch off** the **Load and Surface Center options**.

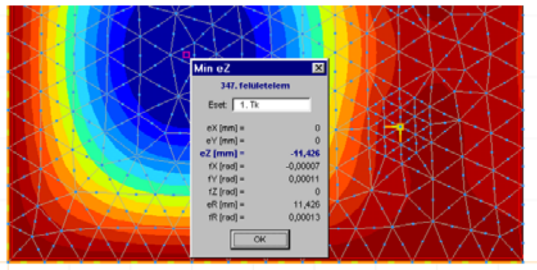
Min/Max Values



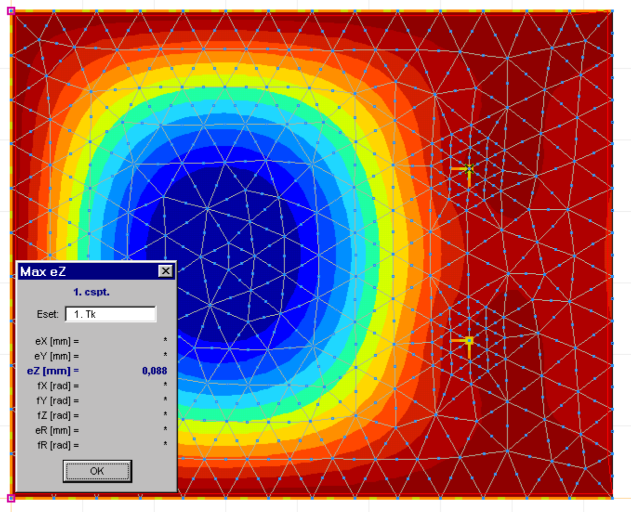
Let's find the maximal displacements. **Click the Min, Max Values Icon**. The following dialog window appears:



Here you can select the displacement component extremities. **Accept ez,** and a window pops up, showing the location and value of maximum negative displacement

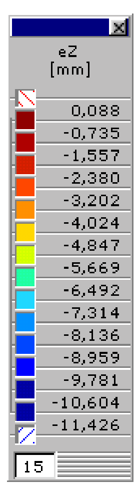


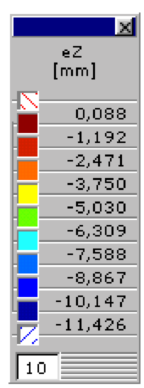
Click **Ok**, and another window pops up, showing the location and value of maximum positive displacement.



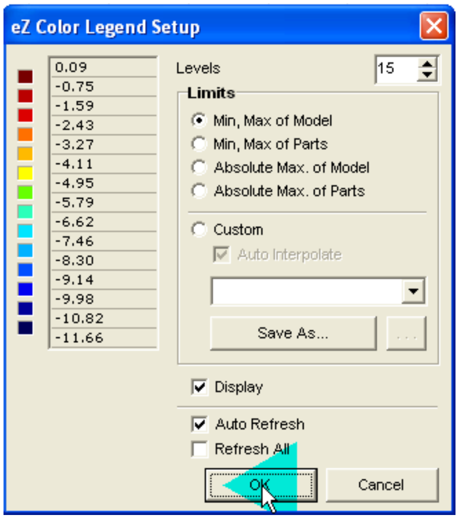
Color Legend

The Color Legend Window shows the color ranges. You can change the number of colors by dragging the handle beside the level number edit box or entering a new value.

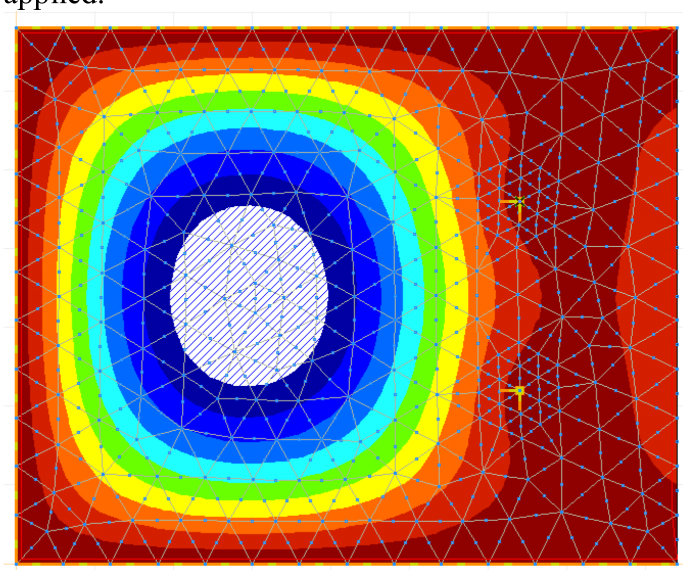




Let’s find the ranges with a displacement larger than 10 mm. **Click** on the values in the Color Legend Window. In the Color Legend Setup dialog window **check Auto Interpolate**, then **click** on the bottom value in the left column, and **replace -11.4 with -10**.



Close the dialog window with **OK**, and the new ranges will be applied.

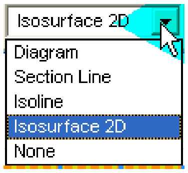


The ranges with a displacement larger than 10 mm are shown by the inclined hatching.

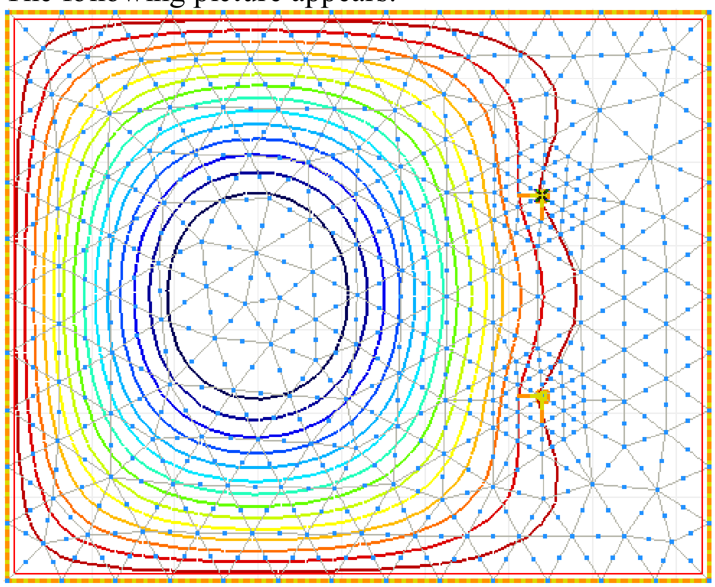
Display Mode

Let's view the displacement in isoline display mode too. **Click the Display Mode combo box** (the one which is displaying

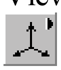
Isosurface 2D), and select Isoline from the list.



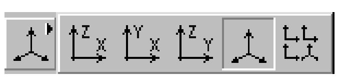
The following picture appears:



Perspective View

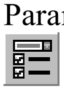


Let's view the results in perspective. **Click the Perspective View Icon** from the View Icon Bar.

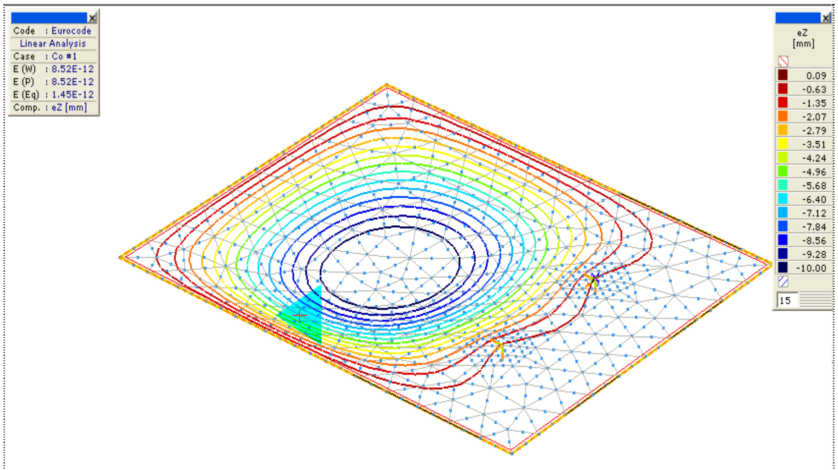


Accept the perspective display values in the dialog window by closing it with Close Icon.

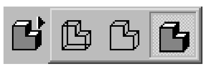
Result Display Parameters



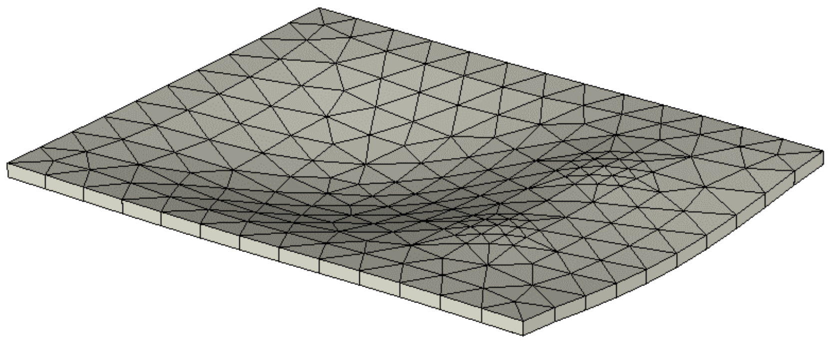
Click the Result Display Parameters Icon to view the deformed shape. In the Display Shape Panel **select Deformed**. When the dialog window is closed the deformed shape of the structure is shown.



Rendered

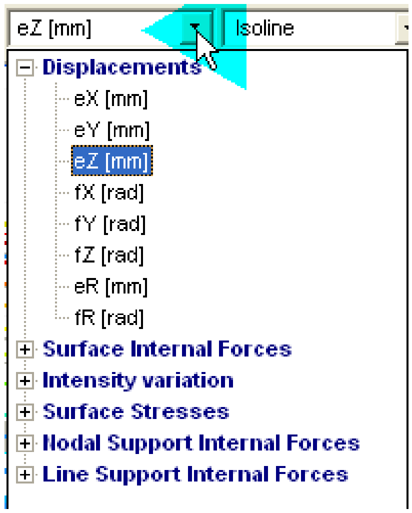


Click the Rendered Icon in the Display Mode Icon Bar, and the deformed shape of the structure will be rendered.



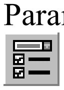
Click the Wireframe Icon and **return to the Isoline** display mode.
Let's switch to X-Y Plane.

After studying the deformed shape let's look at the internal forces. **Click the Result Component combo box** (the one which displays ez), and the following list appears:

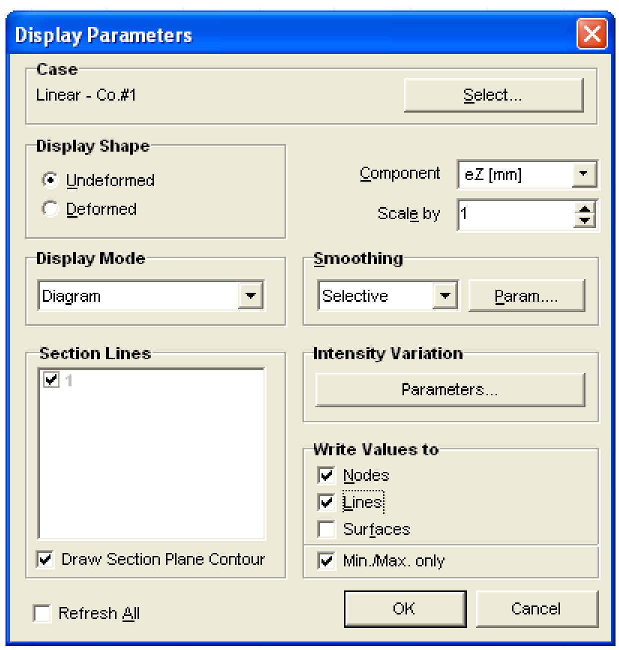


Open the Surface Internal Forces by clicking on it, then **select mx**. The isoline display of the mx internal moments appears on the screen. This is the moment that is taken by the reinforcement in the -x direction. The my, mxy internal moments and the qxz, qyz shear forces can be viewed in a similar way.
Open in the Result Component combo box the **Nodal Support Internal Forces**, and **select Rz**. This way you will be able to see the compressive force acting on the columns.

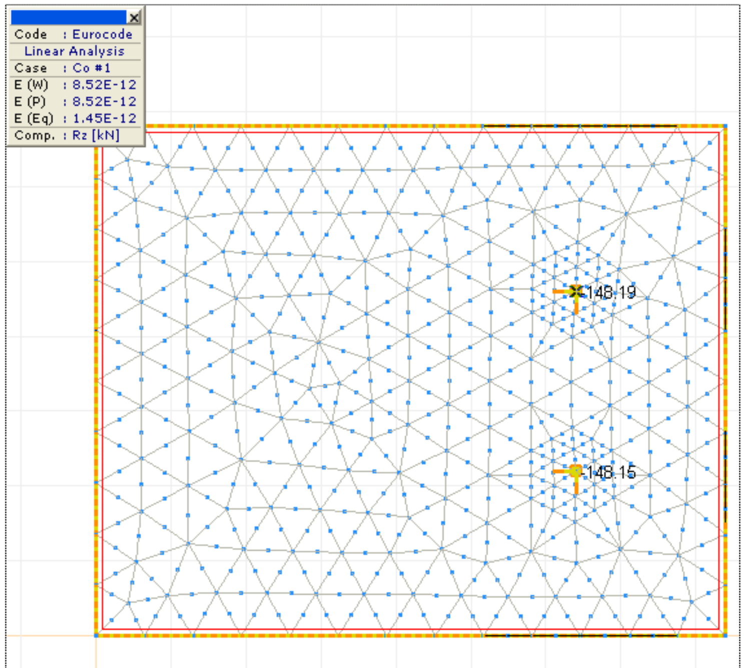
Result Display Parameters



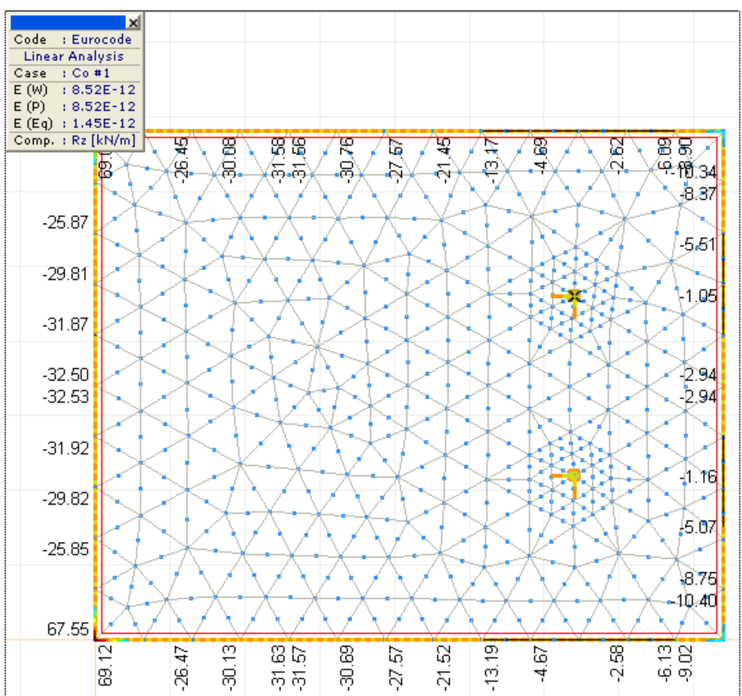
For this **click the Result Display Parameters Icon**.
The following dialog window appears:



In the Write Values To Panel **check** the Nodes box, and **uncheck** the Min, Max. only. Close the dialog window with **Ok** and the value of the axial forces in the columns appears near the nodes.

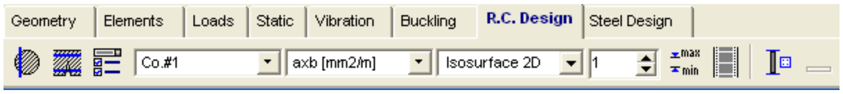


The reactions from the line supports can be viewed in a similar way. In Result Display Parameters **check only Lines** in the Write Values to Panel. **Select Line Support Internal Forces and value Rz.**



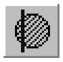
R.C. Design

Let's **switch** to R.C. Design tab.

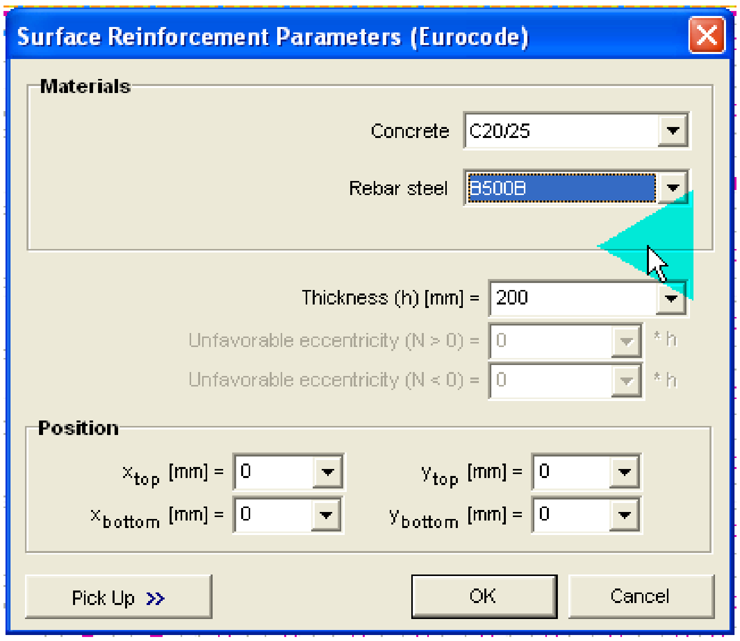


Here the reinforcement areas from the internal forces can be obtained.

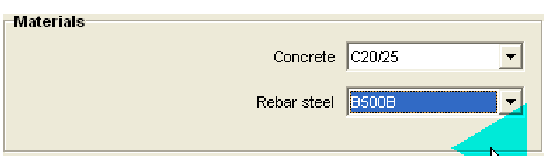
Reinforcement Parameters



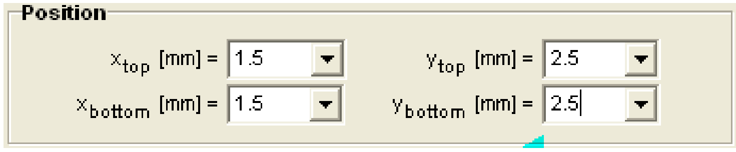
Click the Reinforcement Parameters Icon, and **select all surface elements** with the All command. Finish the selection with **Ok**, and the following dialog windows appear:



The characteristics of the concrete are already known from the creation of domain. Select B500B for the type of the reinforcement:



Type in 1.5 for the depth of concrete cover in -x direction, **and 2.5** for the -y direction.

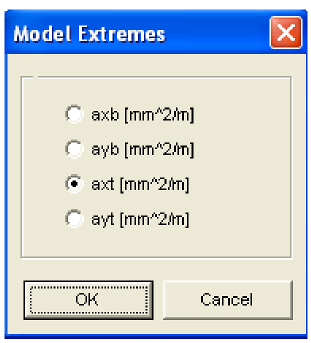


When the dialog window is closed, the axb diagram appears, which is the isosurface diagram of the bottom steel area in -x direction. In the Result Component combo box you can select the top or bottom -x or -y direction of the steel reinforcement.

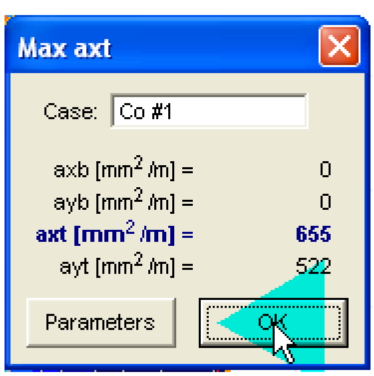
By changing the number of levels and the top and bottom values in the Color Legend Window, it is easy to see variations in the required reinforcement needed.
In this case let's study the reinforcement at the top in -x direction. **Switch to –axt** in the Result Component combo box.

Min/Max Values

Find the maximal amount of steel reinforcement using the Min, Max. Values command. **Clicking** on its icon the following dialog window appears:

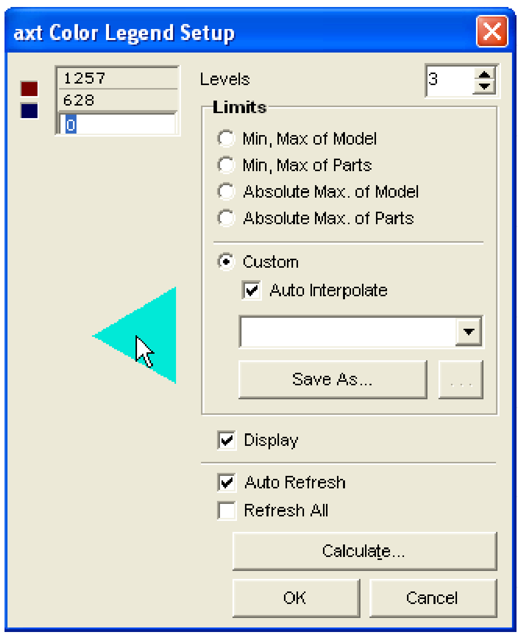


Continue with **Ok**, and a dialog window appears with the location and area of maximum reinforcement.

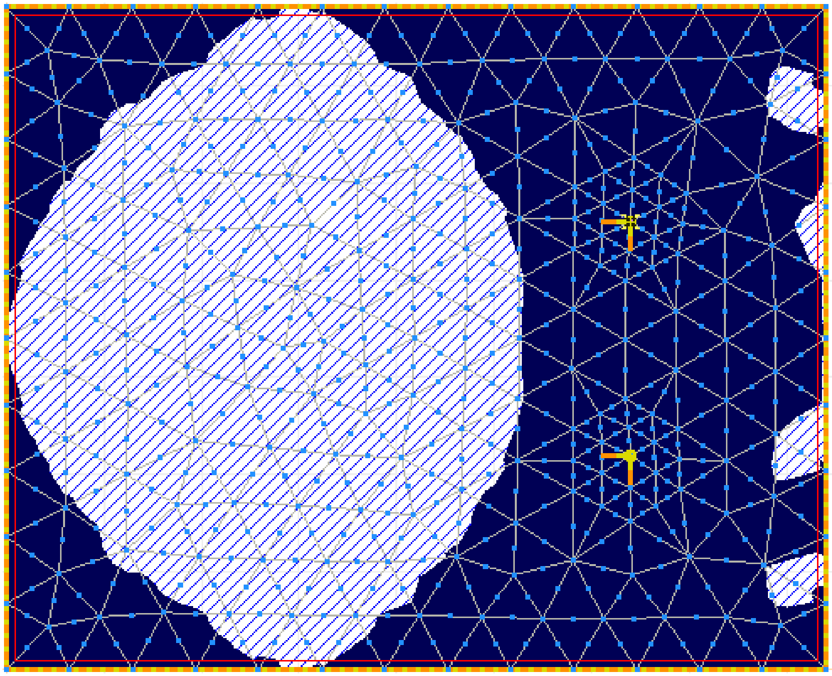


Let's use as minimal reinforcement (0.3%) fi12/18, whose area is 628 mm2/m, and for actual reinforcement fi12/9, whose area is 1257 mm2/m.
It can be seen that the area for actual reinforcement is greater than the maximum area of calculated reinforcement, so it can be applied over the whole plate.
To separate reinforcement regions **set the number of levels to 3** in the Color Legend Window.

Activate the Color Legend Setup by **clicking** on a value, then **type 1257 in** the top row, **628 under it** and **0 in the last row**.



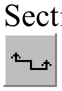
The regions that require the minimum or maximum reinforcement are displayed.



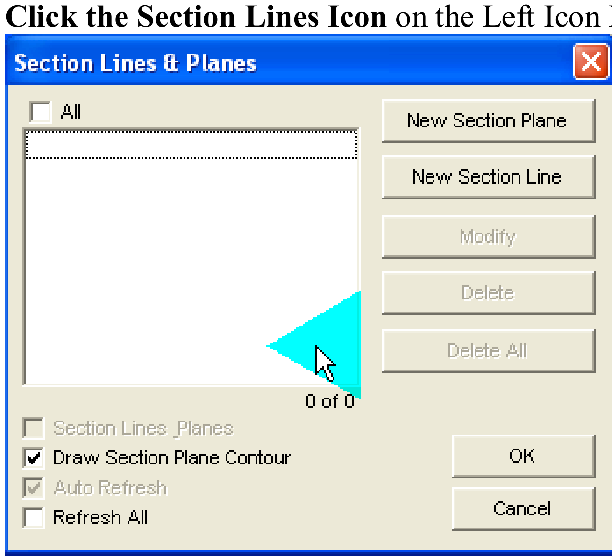
It can be seen that in the middle region of the plate no top reinforcement in -x direction is required from calculation, near the edges the minimal reinforcement is enough and in the area around the columns, the maximum reinforcement is required.

To view the reinforcement needed in the area around the columns **Click the Static tab.** In Result Component combo box **select Surface Internal Forces** and **click on -mxy. Set the display to Isosurface. .**

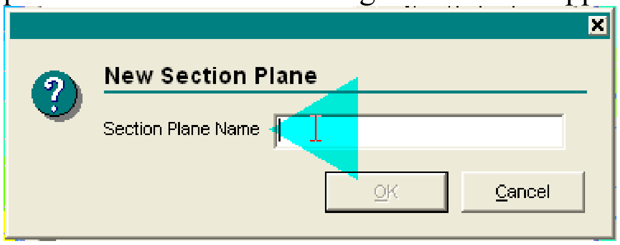
Section Lines



Click the Section Lines Icon on the Left Icon Bar.



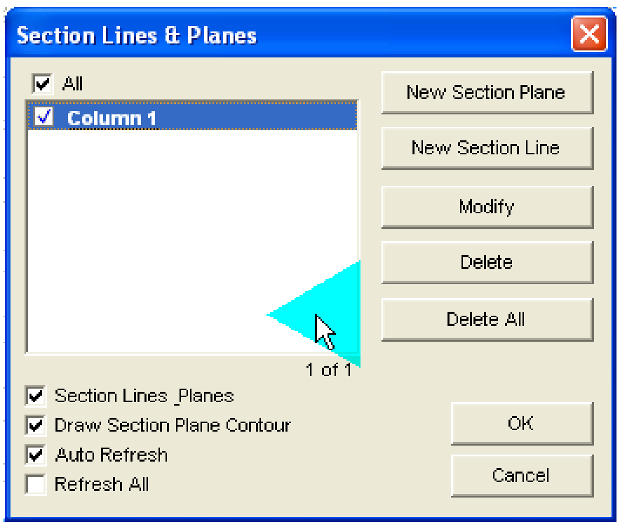
Click the New Section Plane button, and **name** the section plane Column1 in the dialog window that appears:



Accept the name and specify the section plane on the drawing.

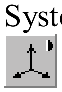
Select one of the **column support nodes**, then the other **column support node**.

The following dialog window returns:

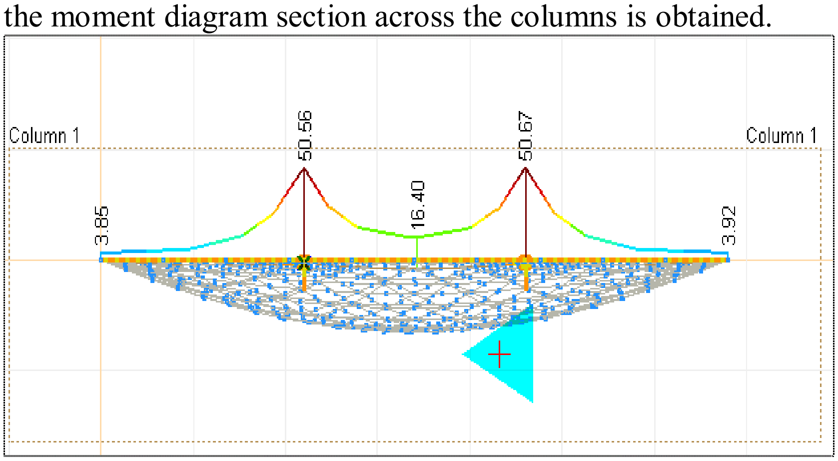


Accept it with **Ok.** The display should be set to the **Section Line.**

Coordinate System



Switch to Z-Y plane, **Select Surface Internal Forces –m1** and the moment diagram section across the columns is obtained.



Let's **switch off** the display of section. **Click the Section Lines Icon uncheck** the box before Column1 and close the dialog window with Ok.

Result Display Parameters

Click the Result Display Parameter Icon, and **uncheck** the boxes in the Write Values To panel.

Switch to perspective view, then **set the display mode to Isosurface 3d.** The **Color Legend window** should be **set to –10 max value.**

The following picture appears, which shows the internal moments in the -x direction.