1. PLATE MODEL



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

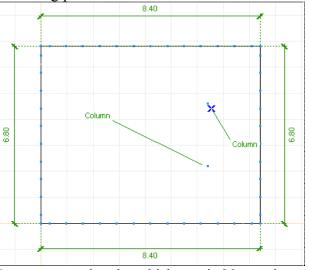
New

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**.

ew Model Model <u>F</u> ilename:	Model1			
Design Code	•	Units and Formats	•	Change Settings
Page <u>H</u> eader Project Title				
Analysis by Greg Hoback				

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 <u>global</u>



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.

🗶 🚺 dX[m] :	15,904	d r[m] :	17,880
, dY[m]:		d a[9]:	27,19
d dZ[m]:		a dh[m]:	0
dL[m] :	17,880		

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

Geometry	Elements	Loads	Static	Vibration	Buckling	R.C. Design	Steel Design
• 🖵 (3+/ >	囲	1>			₩,⊞	



Line

Click the Y-X view from the View Icon Bar

☆ ☆ ☆ ☆ た 比

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

Rectangle

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 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.)

X dX[m]:	8,4	d r[m] :	8,102
, dY[m]:	6,8	d d a[°]:	24,03
d dV[m]:	0	a dh[m]:	0
dL[m] :	8,102		

The following picture appears:

					-*
Y					
x	.1	1	<u> </u>	.1 1	1.0

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**.

Node Icon

Exit from rectangle line command by pressing **Esc**. **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.ElementsThe next step is to define the finite elements. Click the
Elements tab.

Geometry	Elements	Loads	Static	Vibration	Buckling	R.C. Design	Steel Design
• لِـر •	⊇+⁄ ⊅	ド囲	$\langle \rangle$			1 🎫 🖽	



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.

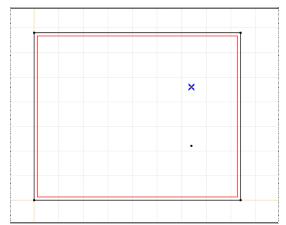
Doma	ain 1	X
(C Define 🕞 Modify	
•	Type Membrane (plane stress) Membrane (plane strain) Plate Shell	
Г	Material C20/25	ð
	Thickness [mm] = 200	×
	Local x Reference × Auto	
	Local z Reference Auto	
	Pick Up >>	OK Cancel

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

esign Code	Materials		C12/15		
🕂 MSZ (Hungarian) ٨	C12/15	~	Туре	Concrete	
STAS (Romanian	C16/20			Isotropic	
Eurocode	C20/25		E [N/mm ²]	8596	
ENV 206	C25/30		v	0.20	
EN 10025	C30/37 C35/45		α _τ [1/°C]	1E-5	
prEN 10113	C40/50		ρ[kg/m ³]	2500	
DIN 1045 (Germa	C45/55		f _{ck} [N/mm ²]	12	
+ NEN (Dutch)	C50/60		Yc Y	1.500	
+ DIN 1045-1 (Gern	STEEL Fe 360		^г с α.	0.85	
E SIA 162 (Swiss)	STEEL Fe 430		φ,	2.00	OK
C	STEEL Fe 510		Ψt	2.00	
<	STEEL Fe E 275	\sim			Cancel

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.

[Do	main 1 (p	plate)] C	20/25; 2	200 mm; :	57.12 m	^2; 2856	50.00 kg	; Auto; A	Auto
7									

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

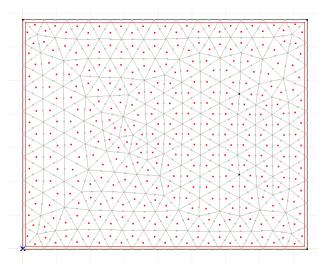
Click the Domain Meshing Icon. Use the select All command (the asterisk) and finish the selection with **OK**. The following dialog window appears:

Meshing Parameters	×
Average Mesh Element Size [m] = 0.66	_

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.

<mark>[Pla</mark> te 264]	C20/25; 200 mm; 28560.00 kg; Auto; Auto	

Refinement

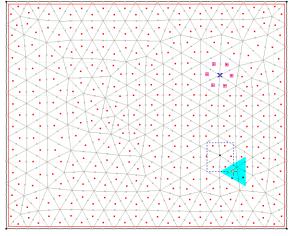
Let's refine the mesh around the two nodal supports. **Depress the left mouse button** over the Refinement Icon, and **click the Refinement by BiSection Icon** that appears.

MAT2	

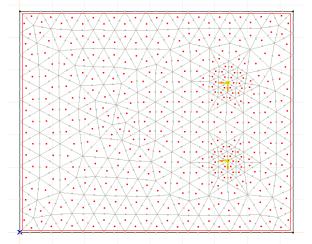
Uniform Refinement



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:



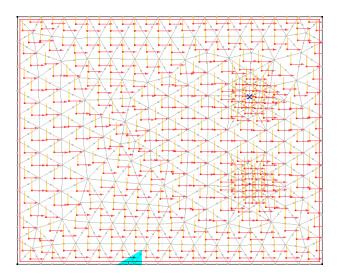
Display Options Let's view the local coordinate system of the surface elements. **Click the Display Options Icon** in the Icons Menu (left side).

Display Options	
Symbols Labels Switches	
Graphics Symbols ✓ Mesh ✓ Node ✓ Surface center ✓ Domain ✓ Support ✓ Reference ✓ Cross-section shape ✓ End releases ✓ Structural Members ■ Reinforcement param. ✓ Load ✓ Mass	Local Systems
I Auto Refresh ☐ Refresh All	OK Cancel

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

Close the dialog window with **Ok**.

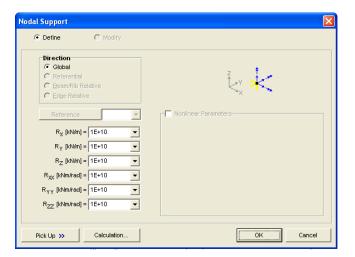
A red line shows the local -x direction, a yellow line the local -y direction and a green line the local -z direction:



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

Display Options

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

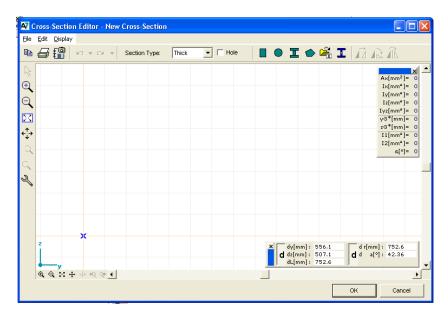


Calculations **Click** the Calculations button. The following dialog window appears:

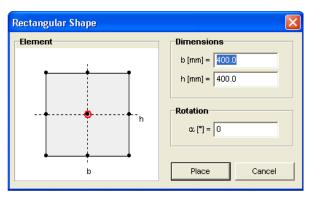
Global Node	Support Calculat	ion			×
Z L	Material C20/25 Cross-Section L [m] = 3.00	•	0 1) <mark>1</mark>	End Rel X	eases Y
	R _X [kN/m] =	0	R _{XX} [kNm/ra	ad] =	0
	R _γ [kN/m] =	0	R _{YY} [kNm/	rad] =	0
	R _Z [kN/m] =	0	R _{ZZ} [kNm/r	ad] =	0
					ancel

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

New CrossClick the New Cross Section Icon. The following dialogSectionwindow appears:

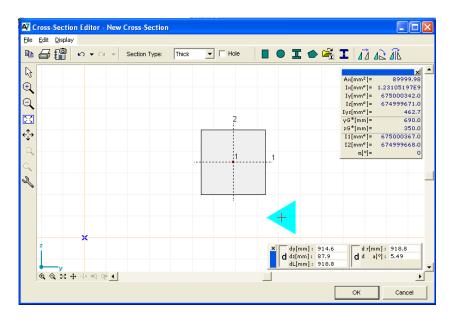


Rectangle Shape **Click the Rectangular Shape Icon**. The following dialog window appears:

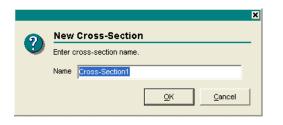


Type 300 [mm] in the upper two edit boxes, as the dimensions of cross section, and click Place. **Click** in the Cross Section Editor Drawing Area to place the rectangle. The location where the rectangle is placed is unimportant.

A following picture appears:

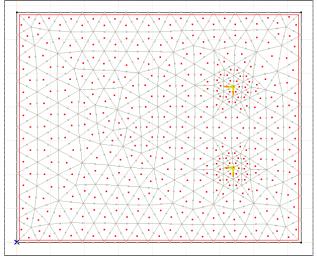


Close the Cross Section Editor with **Ok**. A dialog window asks for the name of the new cross-section.



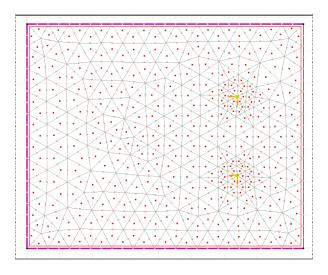
Type in 300x300, then close the dialog window with **Ok**. The Global Node Support Calculation dialog window's stiffness values will take into account this cross-section's properties. Accepting the remaining settings click **Ok**. The stiffness values displayed in the Global Node Support Calculation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

Line Support	
Define C Modify	
Direction Global C BeamRib Relative C Edge Relative	Nonlinear Parameters
$ \begin{array}{l} R_{\chi} \left[(kl Mn/m) = \right] & 1E+5 \Psi \\ R_{\chi} \left[(kl Mn/m) = \right] & 1E+5 \Psi \\ R_{\chi} \left[(kl Mn/m) = \right] & 1E+5 \Psi \\ R_{\chi\chi} \left[(kl Mn/mad/m) = \right] & 1E+5 \Psi \\ R_{\chi\chi} \left[(kl Mn/mad/m) = \right] & 1E+5 \Psi \\ R_{\chi\chi} \left[(kl Mn/mad/m) = \right] & 1E+5 \Psi \\ \end{array} $	
Pick Up >> Calculation	

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 shown in this dialog box.

Local Line Su	pport Calculat	tion		X
y - Z	Materi L C20/, L (rr d (mr	25	• Ø	End Releases
	R _x [kN/m/m] =	7.12E+4	R _{xx} [kNm/ra	d/m] = 8.55E+3
	R _y [kN/m/m] =	2.85E+3	R _{yy} [kNm/r	ad/m] = 1E+0
	$R_z [kN/m/m] =$	6.41E+5	R _{zz} (kNm/ra	d/m] = 1E+0

Depress both the upper and lower End Release Icons. Close

with **Ok** the dialog windows.

Local Line Support Calculation				
y z	Material C20/25 L [m] = 3.0 d [mm] = 300			
	R _x [kN/m/m] = 1.078	'E+5 R _{xx} [kNm/rad/m] = 0		
	R _y [kN/m/m] =	0 R _{yy} [kNm/rad/m] = 1E+0		
	R ₂ [kN/m/m] = 9.628	E+5 R _{zz} [kNm/rad/m] = 1E+0		
Cancel				

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.

Nodal Degrees of Freedom		
Free node Fixed node Truss girder in Plane X-Y Truss girder in Plane X-Z Truss girder in Plane Y-Z Space truss		
Prescribed Displacement 👻		
I▼ e _X Free Z		
IV θ _Y Free		
₩ ^θ Z Free		
Pick Up >> Cancel		

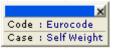
Accepting this will constrain the degree of freedom to vertical displacements and rotations about axes in the plane of the plate.

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:

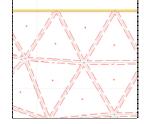
Load Groups & Load Cases		
B Ungrouped	Load Case New Case ↓↓↓ <u>↓↓</u> Static Influence Line	MM S <u>e</u> ismic
	ST1 contains no loads.	Duplicate
	Load Group	Delete
	Ungrouped	
	-Load Group (Eurocode) New Group Permanent Incidental Egocptional	- 🔛 Seis <u>m</u> ic
		Delete
,	ок	Cancel

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:





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 L	oad
Case	
111	
<u>S</u> tatikus	

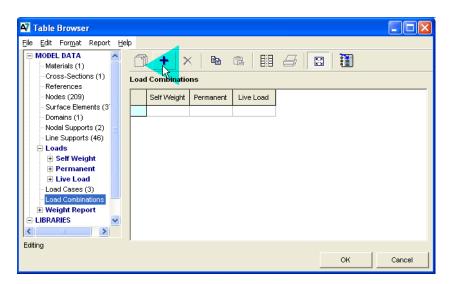
Click the Load Cases & Load Groups Icon again, and **create** a new load case with the Static Icon. **Name it Permanent Load**. This load case contains the dead loads on the plate. Let's assume it is 2.5 kN/m2 distributed load.

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.

Distributed Load on Plates	
Define C Modify	
Direction C Global on Surface C Global Projective C Local	
C Overwrite Add 	
$p_{x} [kN/m^{2}] = 0$ $p_{y} [kN/m^{2}] = 0$	
p _z [kN/m ²] =	
Pick Up >>	OK Cancel

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. Geometry Elements Loads Static Vibration Buckling R.C. Design Steel Design Law Comparison of the static of the
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.

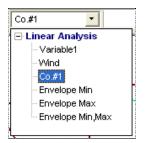
During the analysis the following window appears:

🛛 Linear Analysis of PlateMoo	lel.axs	
PlateT6 element stiffness matrix eva	aluation	
Details >>	AS .	Cancel

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.

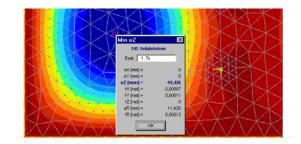
Display Options Click the Display Options Icon on the Icons Menu in the left side. Under the Symbols tab, in the Graphics Symbols Panel switch off the Load and Surface Center options.

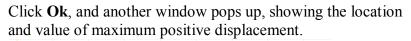
Min/Max Values

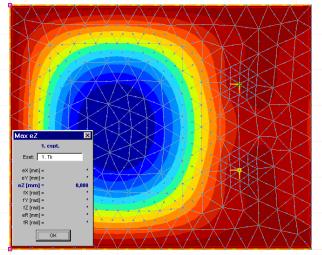
Let's find the maximal displacements. Click the Min, Max Values Icon. The following dialog window appears:

Model Extremes	s 🛛 🔀
Displacements	
CeX[mm]	C fX [rad]
CeY[mm]	C fY [rad]
💿 eZ [mm]	C fZ [rad]
CeR[mm]	C fR [rad]
ОК	Cancel

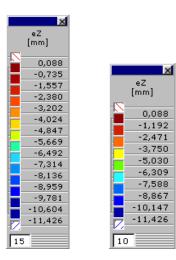
Here you can select the displacement component extremities. Accept ez, and a window pops up, showing the location and value of maximum negative displacement







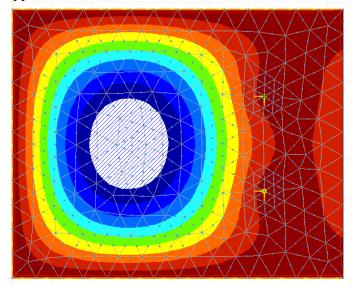
Color Legend The Color Legend Window shows the color ranges. You can change the number of colors by dragging the handle beside the level number edit box or entering a new value.



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.

eZ Color Legend Setup 🛛 🔀				
0.09 -0.75 -1.59 -2.43 -3.27 -4.11 -4.95 -5.79 -6.62 -7.46 -8.30 -9.14 -9.98 -10.82 -11.66	Levels 15 1 Limits Min, Max of Model Min, Max of Parts Absolute Max. of Parts Custom Auto Interpolate Save As			
	Cancel			

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



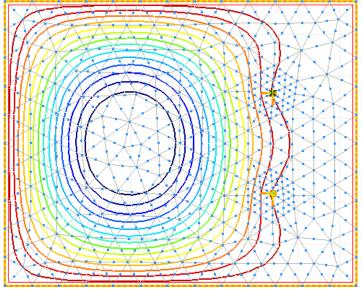
The ranges with a displacement larger than 10 mm are shown by the inclined hatching.

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

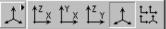
Isosurface 2D), and select Isoline from the list.



The following picture appears:

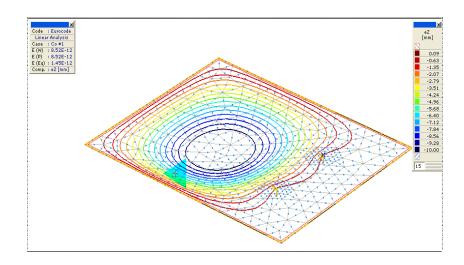


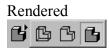
PerspectiveLet's view the results in perspective. Click the PerspectiveViewView Icon from the View Icon Bar.Image: Image: Image:



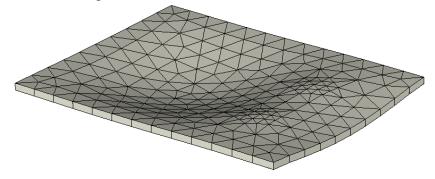
Accept the perspective display values in the dialog window by closing it with Close Icon.

Result DisplayClick 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.





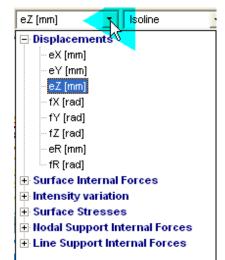
Click the Rendered Icon in the Display Mode Icon Bar, and the deformed shape of the structure will be rendered.



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:



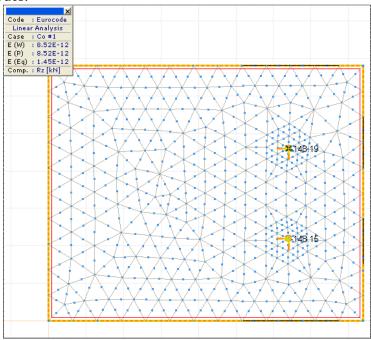
Open the Surface Internal Forces by clicking on it, then **select mx**. The isoline display of the mx internal moments appears on the screen. This is the moment that is taken by the reinforcement in the -x direction. The my, mxy internal moments and the qxz, qyz shear forces can be viewed in a similar 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 DisplayFor this click the Result Display Parameters Icon.ParametersThe following dialog window appears:

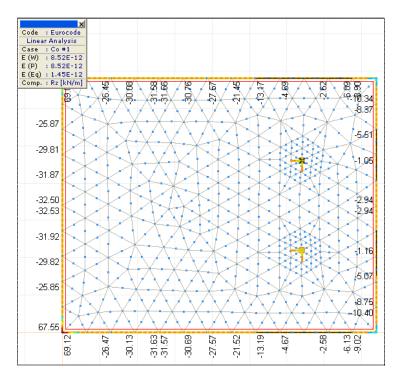
Display Parameters	
Case Linear - Co.#1	Select
Display Shape © Undeformed © Deformed	Component eZ [mm] ▼ Scale by 1
Display Mode Diagram	Selective Param
Section Lines	Intensity Variation Parameters
	Write Values to ✓ Nodes ✓ Lines ✓ Sur <u>f</u> aces
☑ Draw Section Plane Contour	Min./Max. only
∏ Refresh <u>A</u> ll	OK Cancel

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.

Geometry	Elements	Loads S	Static Vibration	Buckling	R.C. Design	Steel Design	
Ø 🌌	Co.#1		▼ axb [mm2/m]	▼ Isos	urface 2D 💌		" II I

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

ReinforcementClick the Reinforcement Parameters Icon, and select all
surface elements with the All command. Finish the selection

with **Ok**, and the following dialog windows appear:

Surface Reinforcement Parameters (Eurocode)	X
Materials	
Concrete C20/25	-
Rebar steel 85008	-
Thickness (h) [mm] = 200	7
Unfavorable eccentricity (N > 0) = 0 + f	
Unfavorable eccentricity (N < 0) = 0 × H	n
Position	
x_{top} [mm] = 0 \checkmark y_{top} [mm] = 0 \checkmark	
× _{bottom} [mm] = 0 v _{bottom} [mm] = 0 v	
Pick Up >> OK Cancel	

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

Materials	
Concrete	C20/25 👻
	85008

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

Position		
\times_{top} [mm] = 1.5 \times_{bottom} [mm] = 1.5	•	y _{top} [mm] = 2.5 💌 y _{bottom} [mm] = 2.5 💌

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

Model Extremes			
C axb [mm^2/m] C ayb [mm^2/m] C axt [mm^2/m] C ayt [mm^2/m]			
OK Cancel			

±max ≭min

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

Max axt	
Case: Co #1	
axb [mm ² /m] =	0
ayb [mm ² /m] = axt [mm ² /m] =	0 655
ayt [mm ² /m] =	522
Parameters 🧧	ok 1

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.

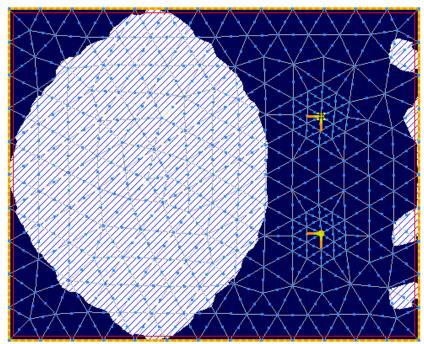
It can be seen that the area for actual reinforcement is greater than the maximum area of calculated reinforcement, so it can be applied over the whole plate.

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.

axt Color Legend Se	etup 🔀
■ 1257 628 0	Levels 3 1 Limits Min, Max of Model Min, Max of Parts Absolute Max. of Model Absolute Max. of Parts Custom Auto Interpolate Save As
	✓ Display ✓ Auto Refresh ✓ Refresh All
	Calcula <u>t</u> e
	OK Cancel

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

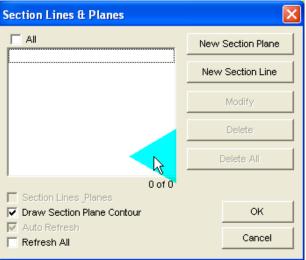


It can be seen that in the middle region of the plate no top reinforcement in -x direction is required from calculation, near the edges the minimal reinforcement is enough and in the area around the columns, the maximum reinforcement is required.

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.

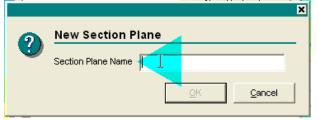


Click the Section Lines Icon on the Left Icon Bar.



Click the New Section Plane button, and name the section

plane Column1 in the dialog window that appears:



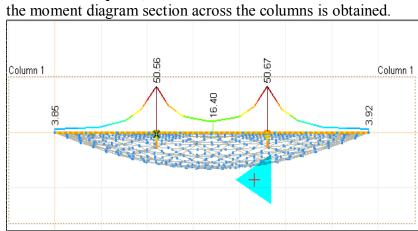
Accept the name and specify the section plane on the drawing.

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

The following dialog window returns:

Section Lines & Planes	
I AII I Column 1	New Section Plane
	New Section Line
	Modify
	Delete
	Delete All
1 of 1	
Section Lines Planes Draw Section Plane Contour	ок
✓ Auto Refresh Refresh All	Cancel

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





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.

