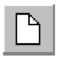
1. BEAM MODEL

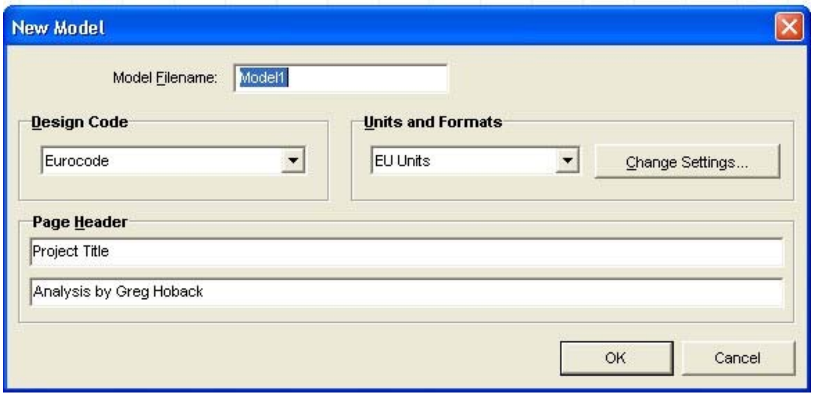
Start

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

New



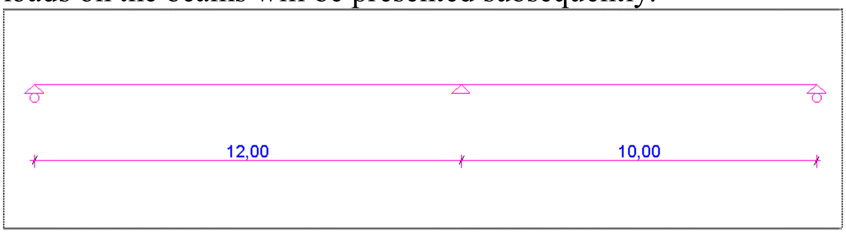
Create a new model with the New Icon. In the dialogue window that pops up, replace the Model Filename with “Beam”.



Select the Design Code. **Click OK** to close the dialog window.

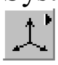
Objective

The objective of the analysis is to determine the internal forces, longitudinal reinforcement and vertical stirrups in the three way supported, reinforced concrete beams illustrated below. The loads on the beams will be presented subsequently.



The analysis will be done according to the Eurocode. The cross-section of the beam is will be a 400mm x 600mm rectangle. The left beam is 12m in length and the right beam is 10m.

Coordinate System



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

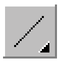
In a new model, the global coordinate ***default location*** of the cursor is the bottom left corner of the graphic area, and is set to X=0, Y=0, Z=0.

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.

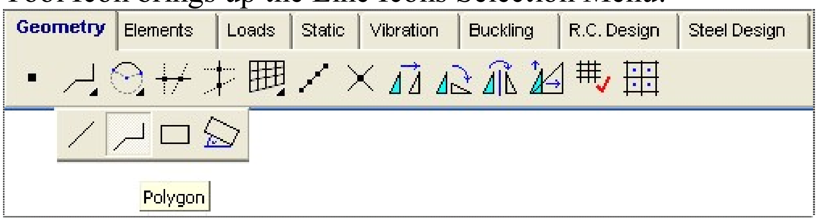
Geometry

The first step is to create the geometry of structure.
Select the Geometry tab to bring up the Geometry Toolbar.

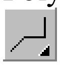
Line



Hold down the left mouse button while the cursor is on Line Tool Icon brings up the Line Icons Selection Menu:



Polygon



Lets **click on the Polygon icon**, which is the second from left to specify the axis of the two beams. When the Polygon is chosen, the Relative coordinate system automatically changes to the local system (‘d’ prefix)

The polygon coordinates can be drawn with the mouse, or by typing in their numerical values. Set the first point (node) of the line by **typing in these entries**:
X=0
Y=0
Z=0
Finish specifying the first line point by **pressing Enter**. The first node of the beam model is now also the global coordinates origin point.

Relative Coordinate System

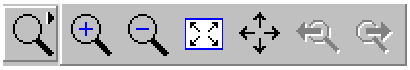
To enter the next two nodes for our beam model **type** in the following sequence:

X=12
Y=0
Z=0, Enter
X=10
Y=0
Z=0, Enter

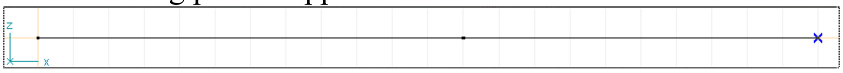
Press ESC twice to exit from polygon drawing function.

Zoom

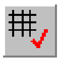
To bring up the Zoom Icon Bar, **move the mouse on the Zoom Icon** in the left side of the desktop window. It contains six icons. Lets choose the third icon (Zoom to fit) from the Zoom Icon Bar, or press Ctrl-W, which has the same effect. An alternative way of zooming is to press the + or – keys on your numerical keypad.



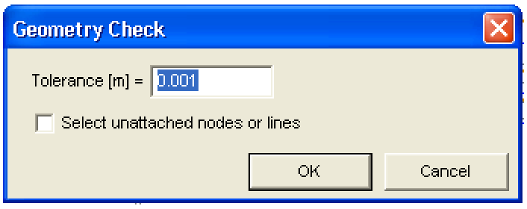
The following picture appears:



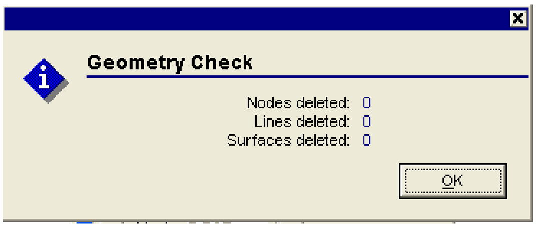
Geometry Check



Click the Geometry Check Icon on the top of the desktop, to check for geometric ambiguities. The program will ask for the maximum tolerance (distance) for merging points.

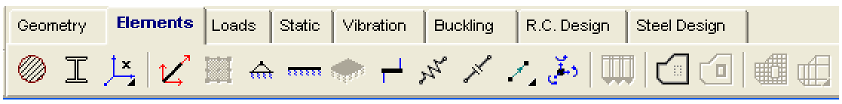


After the geometry check a summary of actions appears.



Elements

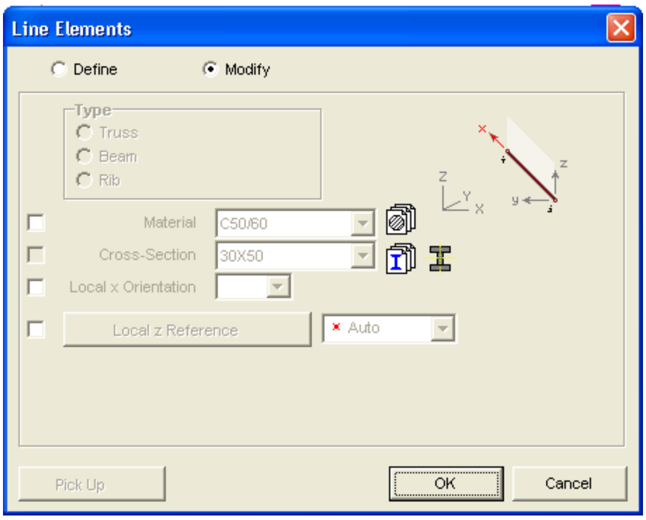
The next step is to specify the finite elements. **Click on the Elements tab** to bring up the Finite Elements Toolbar.



Line Elements

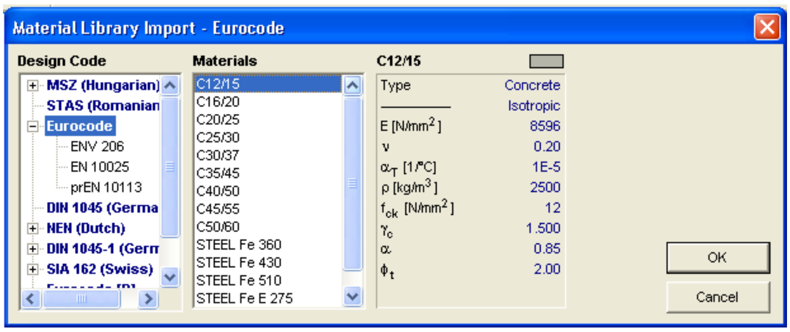
Press the Line Elements Icon,

then on the appearing selection icon bar use the asterisk (All) command, then **click OK**. The Line Elements dialog window appears. Select Define or Modify if you are correcting an earlier parameter.



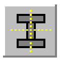
Material Library Import

Press the Material Library Import icon to select the material.

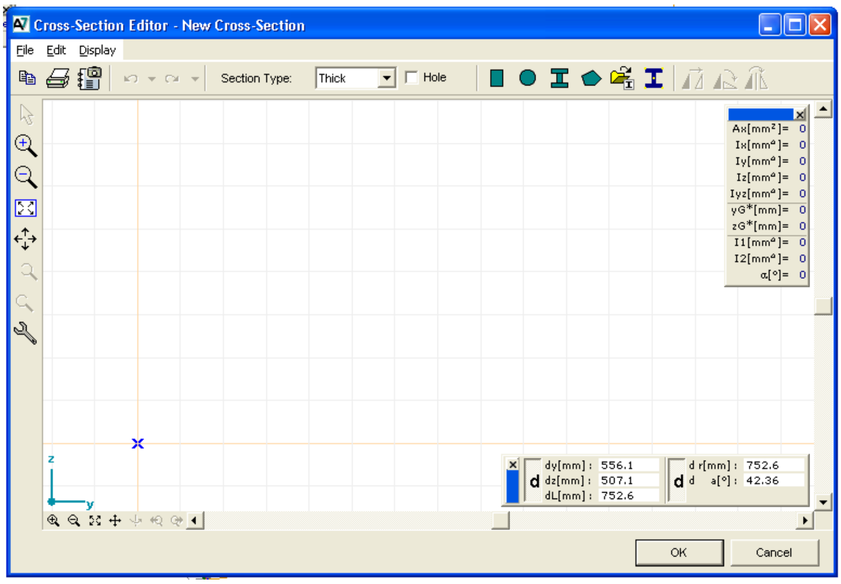


In the dialog window that appears **select Concrete C25/30** in the Materials column, then **click OK**.

New Cross Section

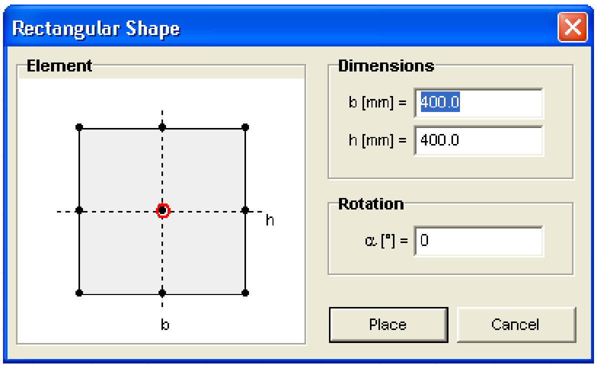


Click on the New Cross Section Icon (the rightmost in the sections line) to create a new cross-section.

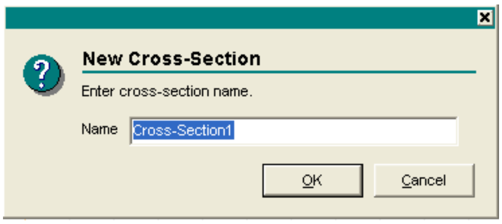


Rectangle

Define a rectangular cross section by **clicking on the Rectangular Icon**.



Finish the cross-section definition by **clicking on the Ok button**.

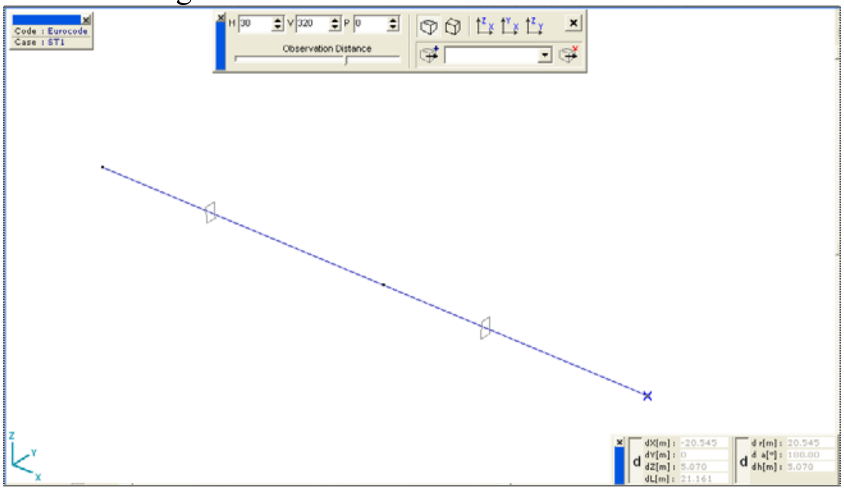


Enter a name for the newly created cross-section. **Type in 400x600, and then press OK.**

Leaving the Local x Reference on Auto, the orientation of the local x axis of the beam will be along the x axis of the element, and the local z axis will be in a vertical plane passing through the x axis.

Perspective

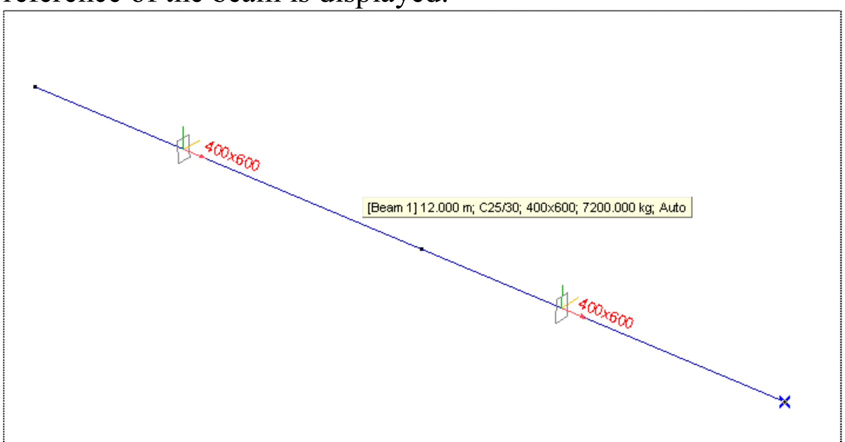
Lets check the structure in space! **Click the Perspective Icon** in the left side of the application. You can pan or rotate the structure using the mouse.



Display Options

The local systems, the node numbering and other useful graphical symbols can be switched on/off by clicking the Display Options Icon in the left side of the application. (Hint: the same dialog window can be displayed by **selecting the Display Options item** after a right click in the Graphics Area). **Check the Beam box** in the Symbols/Local Systems panel, then **select the Labels tab** to **check the Cross-Section Name box.** Exit from the dialog window with **OK**. The local system of the beams and the name of the cross-sections will be displayed. Move the cursor on the axis of the beam to bring up an info label showing relevant information about the beam.

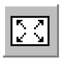
Because the Elements tab is selected, the tag number, length, material name, cross-section name, self-weight and local reference of the beam is displayed:



Finally **switch from perspective view to Z-X plane**.

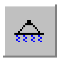
Zoom to Fit

In order to have a good overview, use the **Zoom To Fit**

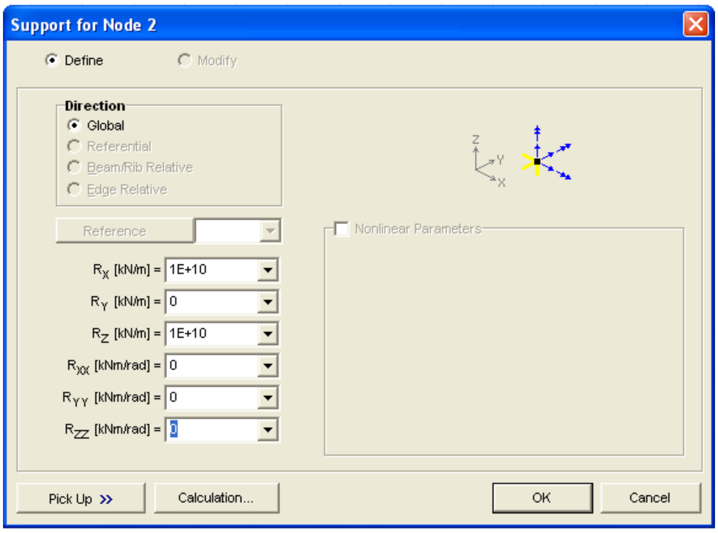


command.

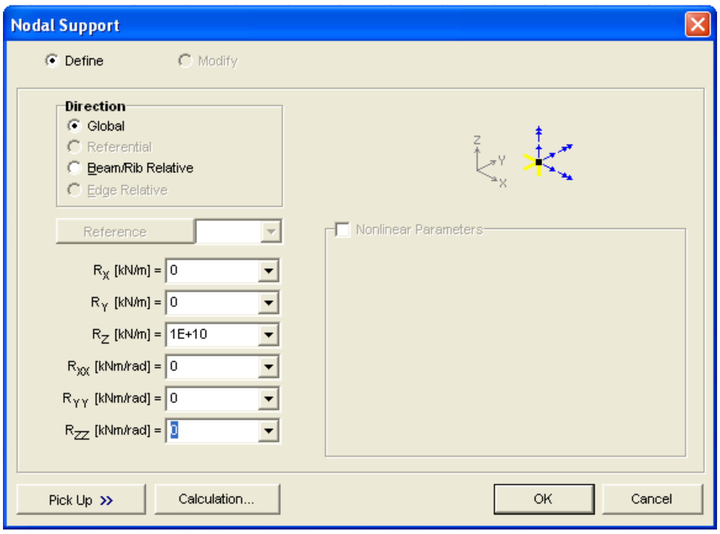
Nodal Support



Click the Nodal Support Icon and select the middle support. A dialog window appears, where you can set the translational and/or rotational stiffness of the node. **Select the global direction**, and specify the stiffness values.
The first three entries are for the translational stiffness, measured in [kN/m]. The default value is 1e+10 [kN/m], meaning a full restriction of the translation, while the value 0 [kN/m] would mean a free translation.
The next three entries are for the rotational stiffness, measured in [kN/rad]. The default value of 1e+10 [kN/rad] means a fully restricted rotation, while the value 0 [kN/rad] means a free rotation. **Set all rotational stiffness to zero**, and **restrict the translation along X and Z direction. Use the settings** in the following box:

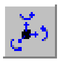


Finish the support definition with **OK**
Select the two exterior supports and make them horizontally free (X, Y axis) supports in a similar way:

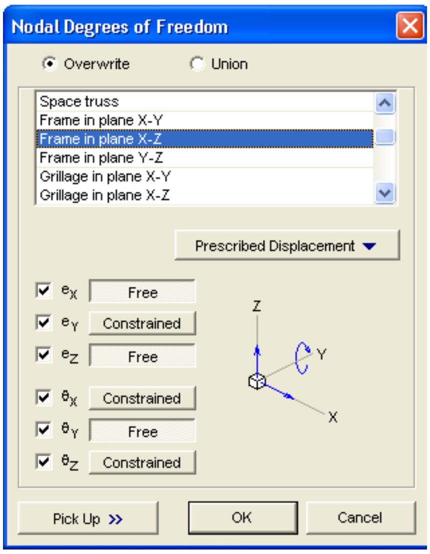


On the screen, restricted translations are shown as yellow stripes, restricted rotations as orange stripes to their rotational axis.

Nodal DOF



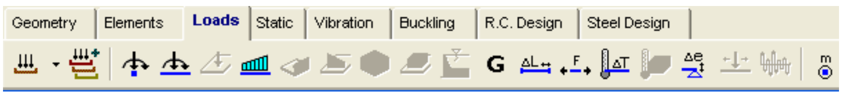
Click the Nodal DOF (Degrees of Freedom) Icon, and **select all nodes** with the All command. In the Nodal Degrees of Freedom dialog window **select 'Frame** in Plane X-Z' from the predefined settings. After closing this window with **OK**, all the nodes will change their color to blue.



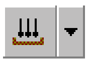
This setting selects the nodes of the beams only in translation in plane X-Z with the rotation around the Y-axis.

Loads

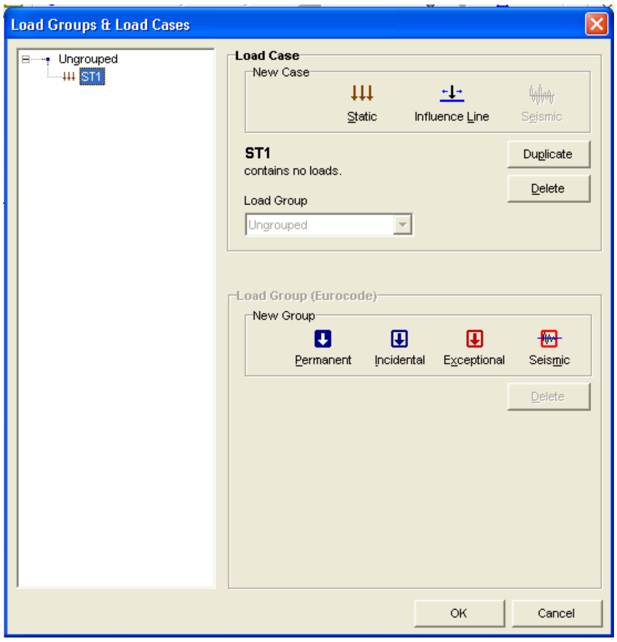
The next step is to apply the loads.
Click the Loads tab.



Load Cases

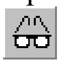


It is useful to separate the loads into load cases. **Click the Load Cases Icon** to create the load cases. The following dialog window appears.



In the left tree view you can see the first load case, created automatically by the program. Its name is ST1. **Click on the ST1** to change the name of the load case, and **overwrite it with SELF-WEIGHT.** Click **OK** to return to the graphics area. The active load case will be SELF-WEIGHT. You can see it on the Info Window.

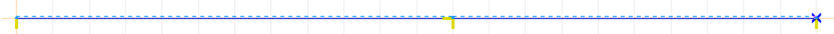
Display Options



In the Display Options window **select the cross-section name** under the Labels tab, and the **cross-section shape and local system** under the Symbols tab, leaving the rest of the default settings.

Self Weight

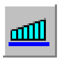
Click the Self-Weight Icon, and **select all elements** with the All command. When the selection is finished by pressing **Ok**, two blue dotted lines will show near the beams axis that their dead load is placed on them (It will act by default along the –Z direction, with the gravitational acceleration taken as g=9.81 m/s2).



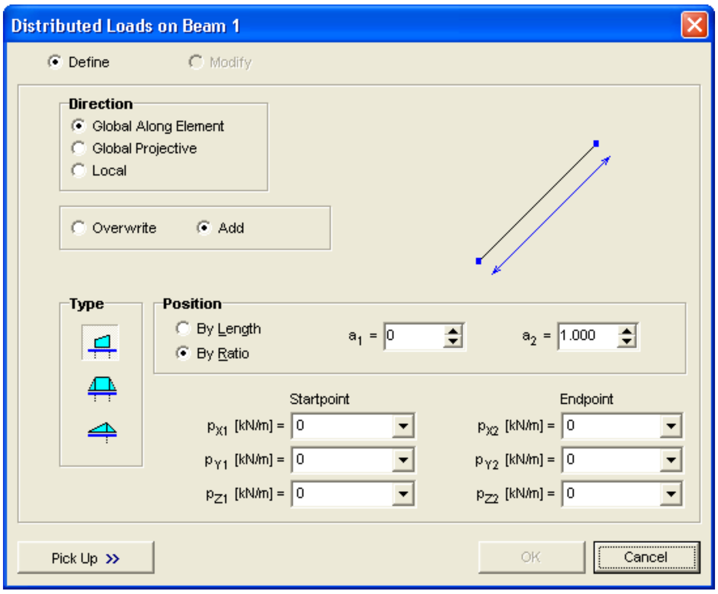
Load Cases/Groups

Click the Load Cases Icon again, and **create three more load cases** by clicking repeatedly the Static Button in the New Case panel. **Name them** VARIABLE1, VARIABLE2 and SUPPORT DISPLACEMENT. **Make VARIABLE1 the active load case by clicking on it**, and press **OK**.

Line Load

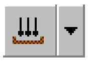


Click the Line Load Icon and **select the left beam.** After finishing the selection with **Ok** the following dialog window appears:

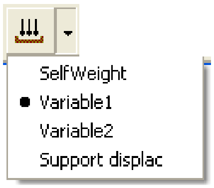


As the load intensity **type -17.5** in the pz1, pz2 edit boxes, then press **Ok**.

Load Cases/Groups

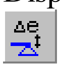


Click on the downward pointing triangle on the right of the Load Cases Icon and the following menu will pop up.

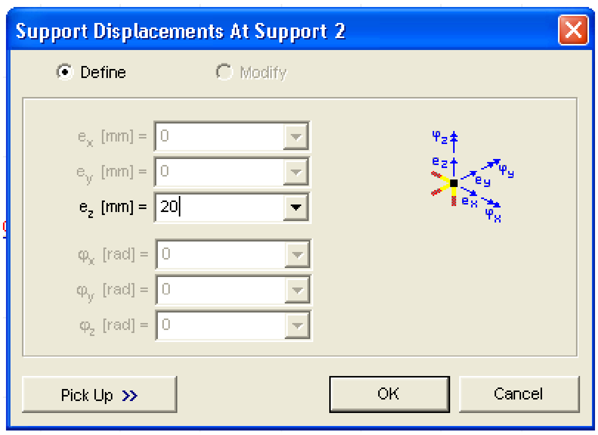


It shows all load cases, a black dot marking the active one. **Click on VARIABLE2** to make it the active load case.
Apply on both beams a -17.5 kN/m uniform linear load acting in Z direction in the same way as before.

Forced Support Displacement

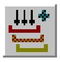


Finally **select the SUPPORT DISPLACEMENT load** case. **Click the Forced Support Displacement Icon**, select the middle support and press **Ok**. This brings up the following dialog window.

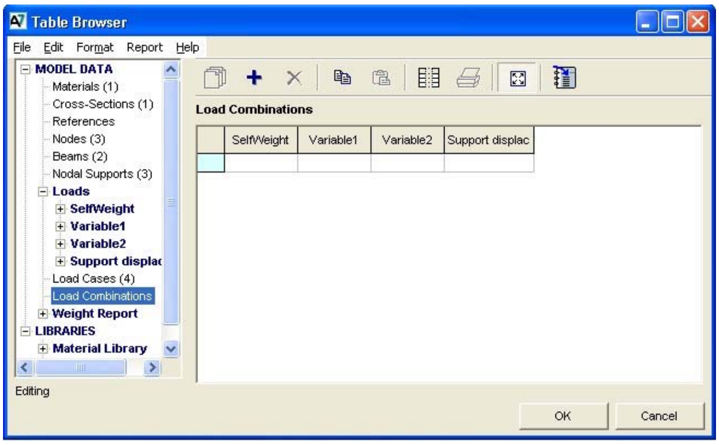


Type in 20 [mm] in the ez edit box.

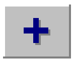
Load Combination



Click the Load Combination Icon, which will open the Table Browser.



New Load Case



Click the New Icon to create an empty load combination. You must specify a combination factor for each load case. For now type in the following factors (press enter after input into each cell)

Selfweight- 1.2
Variable1- 1.4
Variable2- 0
Support displacements- 1.0

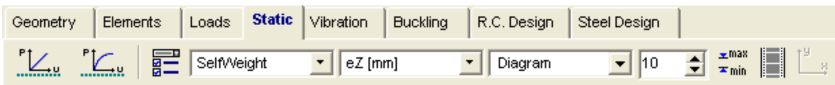
Make another load combination, this time with the following factors:

Selfweight – 1.2
Variable1- 0
Variable2- 1.40
Support Displacement- 1.0

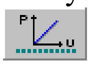
Finish the creation of load combinations by pressing **Ok**.

Static

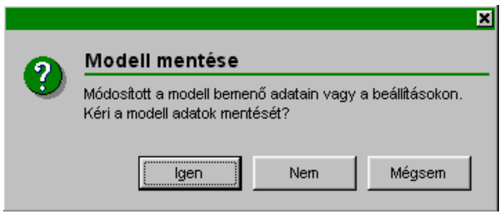
The next step is the analysis and post processing.



Linear Analysis

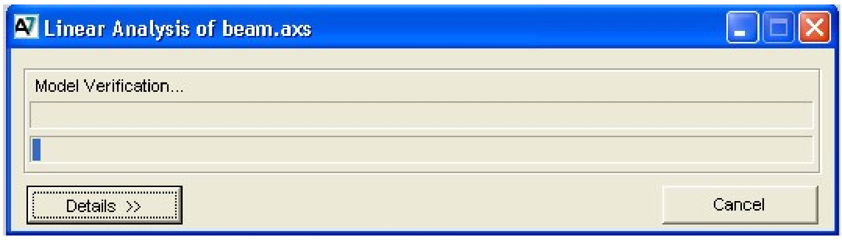


Click the Static tab, then the **Linear Icon** to start the analysis.



If the application prompts for saving, save the model on a local hard disk. After saving, the analysis will start.

Analysis



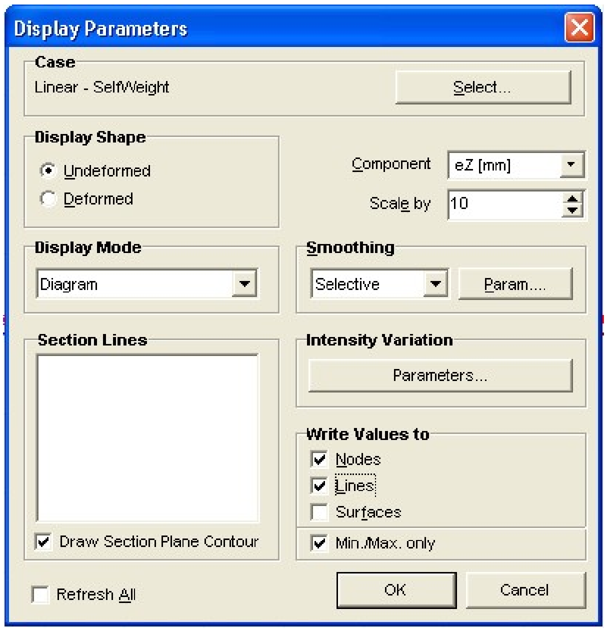
Click the Details button to view the details of calculation.

Static

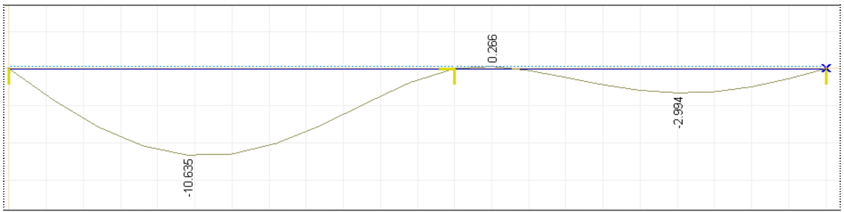
When the analysis has finished, press **Ok**. By default the postprocessor will start with the ez displacement of the first load case, which is now SELF-WEIGHT. The display mode will be iso surfaces. You will see the displacements from the dead load in global Z direction.

Result Display Parameters

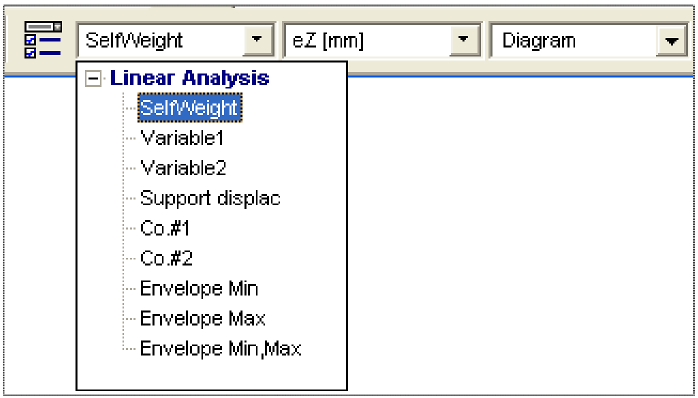
Click the Result Display Parameters Icon and **set the parameters** according to the picture below.



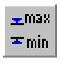
In the Case Selector combo box **select the SELF-WEIGHT** load case. If you leave the Undeformed radio button checked in the Display Shape panel, then the various results will be drawn on the undeformed shape of the structure. In the Component combo box **select ez** from displacements. **Set the Display Mode** to diagram. In the Write Values To Panel **check Nodes and Lines**. Close the dialog window with **Ok**. You should see the following picture:



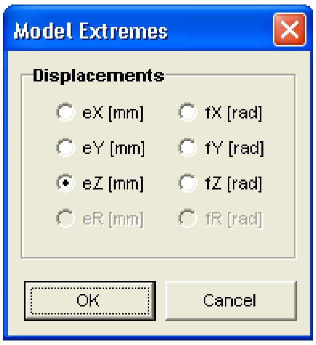
Check the displacement diagrams of various load cases whether they comply with the expected result. To do this, **click on the combo box** next to the Result Display Parameters Icon, and **select desired load** case. This time select the first load combination (Co. #1).



Min, Max Value

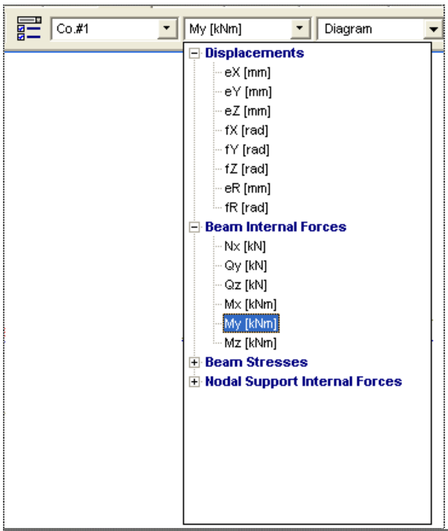


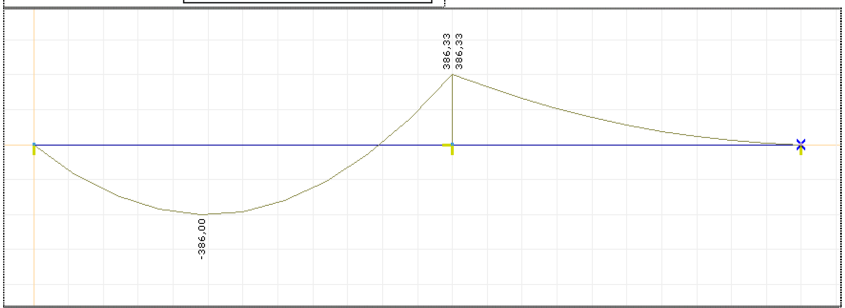
Click on the Min-Max Value Icon to obtain the location and value of the maximum and minimum displacements. The following dialog window appears:

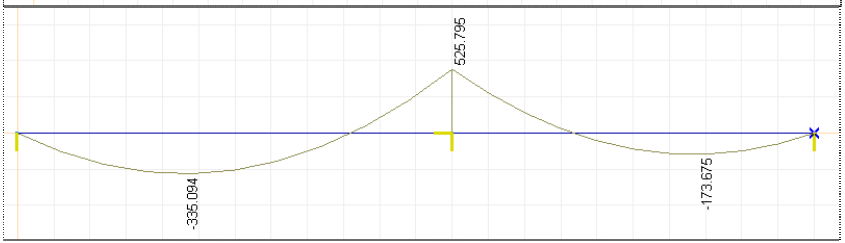


Select the eZ displacement component, and press **OK**. The location and value of the negative maximum displacement pops up in a window. Pressing **OK** closes it, and the positive maximum displacement window pops up. Press **OK** to close it too.

The various internal force and stress results can be selected through the second combo box. First **view the My bending moment** in the first and second load case (Co#1, Co#2), which is accessible by **clicking on the Beam Internal Forces**.

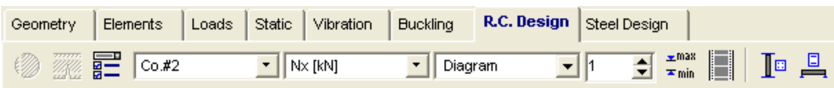






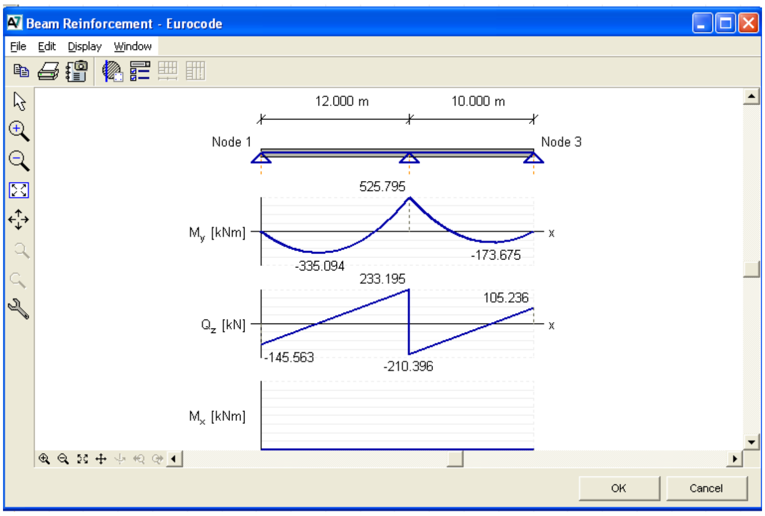
R.C. Design

Click the R.C. Design tab to find out the area of longitudinal reinforcement and vertical stirrups.



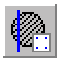
Beam Reinforcement Design

Click the Beam Reinforcement Design Icon, then **select all beams** with the All command (the asterisk), then press **Ok**. The following window appears.

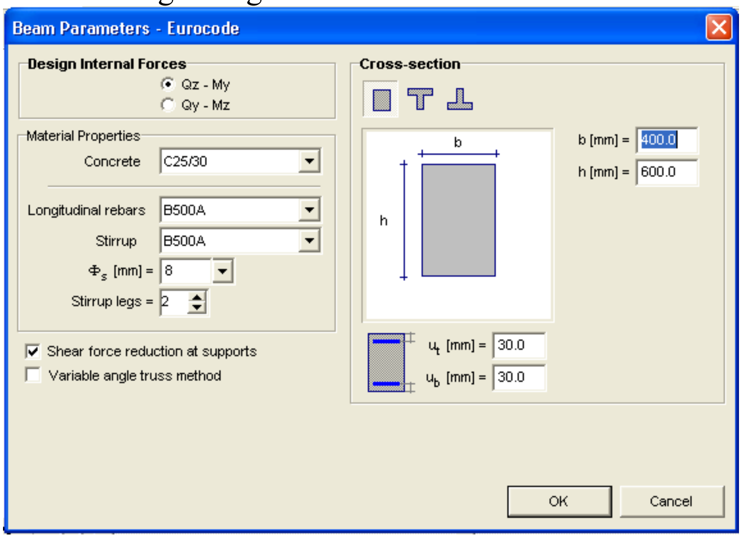


The topmost diagram is the statical layout of the beam, below it is the My moment diagram and the Qz shear force diagram.

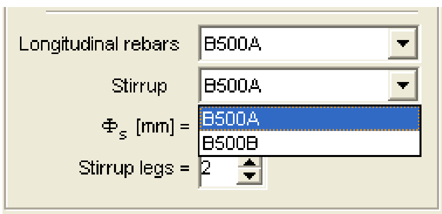
Beam Parameters



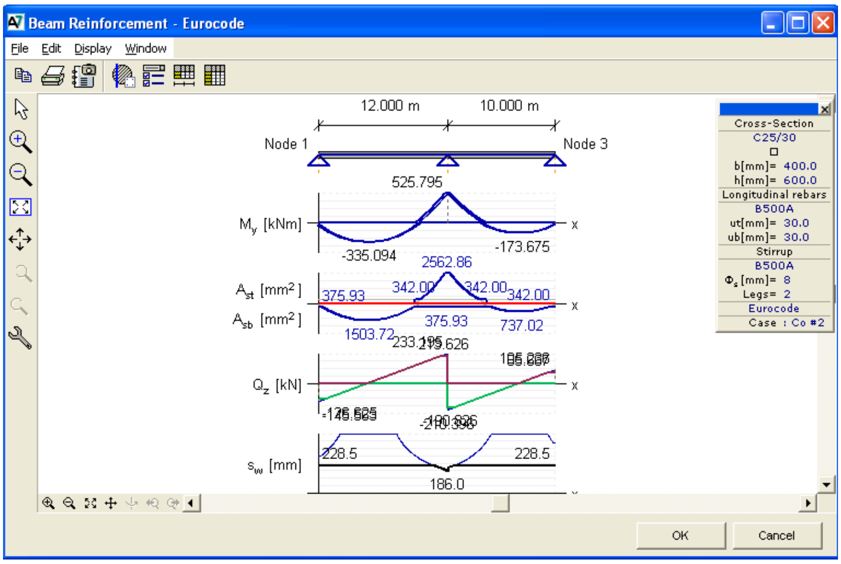
In the Beam Reinforcement Window, **Click the Beam parameters Icon** to set the properties of the beam. It brings up the following dialog window:



Set the longitudinal rebar and stirrup material property to B500A.



Close the dialog window with **Ok** and the following window will appear:



Note that alongside the original My moment diagram (thin line), the diagram shifted according to code (thick line) is also present. Below the My moment diagram is the As diagram, below the Qz diagram is the s diagram.
‘As’ is the area of the necessary longitudinal reinforcement of the beam, while ‘s’ is the required maximal distance of the stirrups. The longitudinal reinforcement in tension is shown in blue, the compressed in red. The area 342 mm2 on the ‘As’ diagram is the minimum area of the tensioned longitudinal reinforcement, while the value 228 mm on the ‘s’ diagram is the maximum stirrup distance.

Click **Ok** to close this reinforcement window.