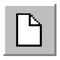
2. FRAME MODEL

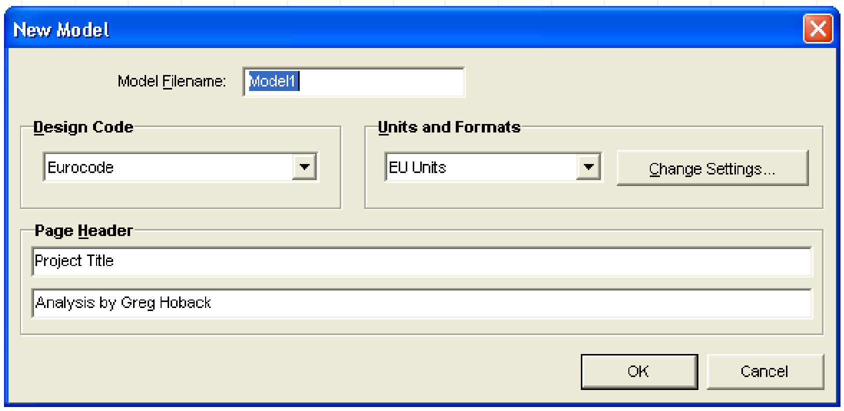
Start

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

New

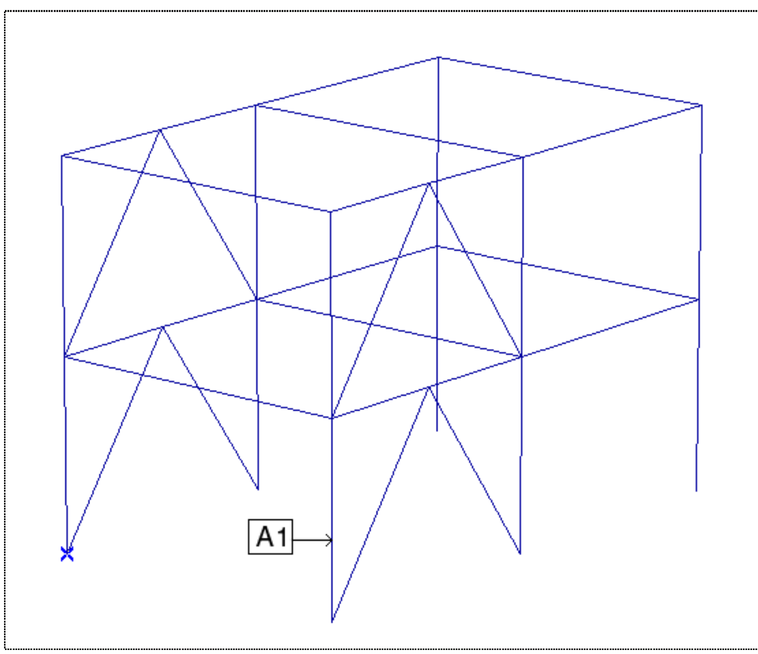


Create a new model with the New Icon. In the dialog window that pops up, **replace the Model Filename** with “Frame”, and in the Design Code panel **select Eurocode**.



Objective

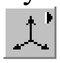
The objective of the analysis is to determine the internal forces of the following frame, and to verify column A1.



Lets use for cross-section of horizontal elements I360, for vertical ones I400, and for inclined ones O 190.0x5.0 SV. The material of the structure is Steel FE 360, and the design verification will be according to Eurocode-3.

By default the Z-axis of the global coordinate system points upward. It has relevance for the direction of gravity, this will be detailed later.

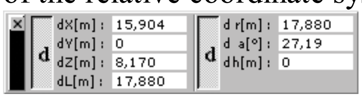
Coordinate System



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.

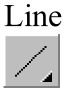


The first step is to create the geometry of the structure.

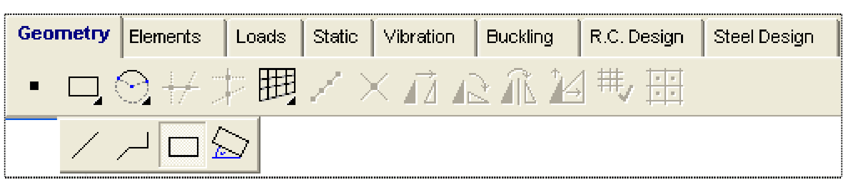
Geometry

Click the Geometry tab, below the menu bar. The Geometry Toolbar appears below the tabs. The geometry of the structure will be created with the Line Tool.

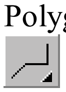
Line



Hold down the left mouse button while the cursor is on the Line Tool Icon will bring up the following Line Type icon bar:



Polygon



Lets **click on the Polygon icon**, which is the second from left. When the Polygon is chosen, the Relative coordinate system automatically changes to the local system ('d' prefix)

The polygon coordinates for the frame model 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 frame model is now also the global coordinates origin point.

To enter the first line (node) of the frame model, **enter the following values**:

X=0
Y=0
Z=3.5, **Enter**

To define the second line of the frame model, **enter the following values**:

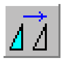
X=6
Y=0
Z=0, **Enter**

To define the third line of the frame model, **enter the following values**:

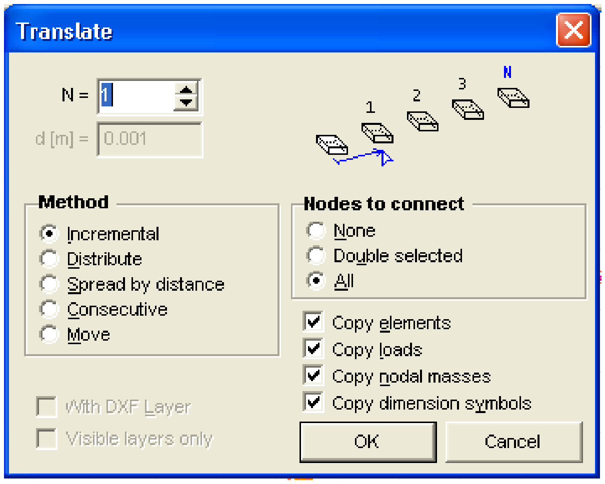
X=0
Y=0
Z=-3.5, **Enter** ***(Note: Negative value)***

Exit from the Polygon command by pressing **Esc** twice.
The following picture is obtained:

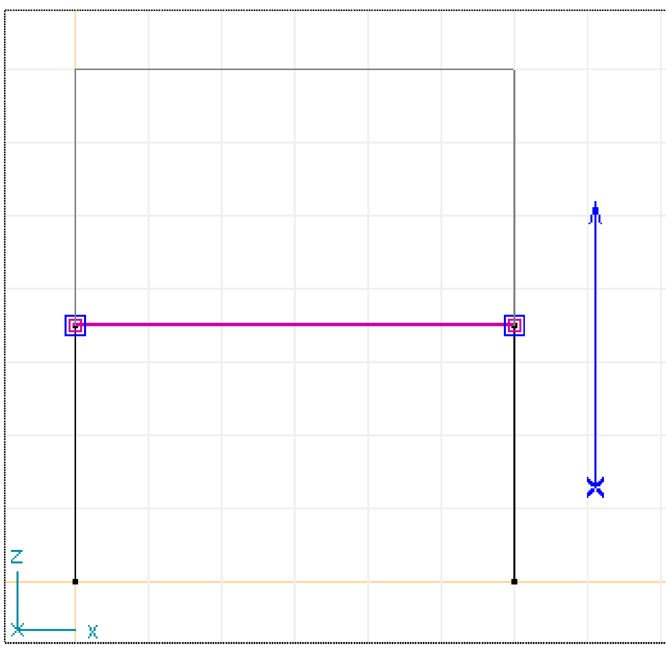
Translate **Copy** the structure vertically upward with the Translate Icon.
For this **click the Translate Icon**, **select** the horizontal line and



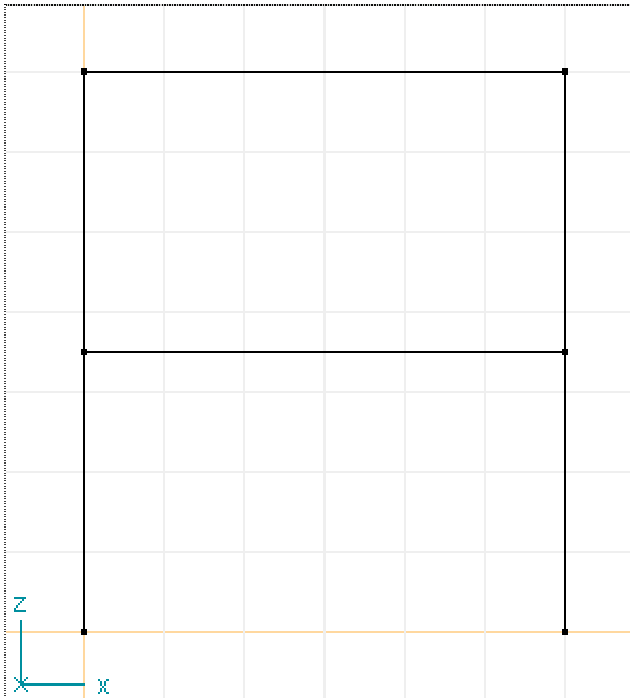
finish the selection with **Enter**. In the Translate dialog window **select Spread by Distance**, in the ‘d [m]=’ edit box **type 3.5**, and in the Nodes To Connect panel **select All**.



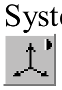
Close the dialog window with **Ok**, then **click** on an arbitrary place in the graphics area and **draw upward** a vertical line, which is longer than 3.5 m.



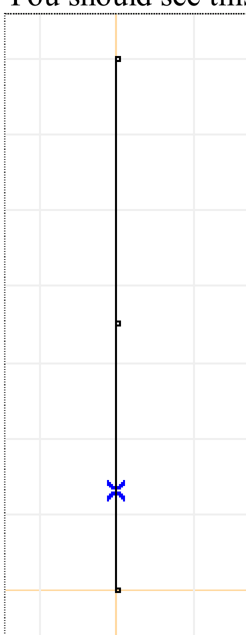
The following picture is obtained:



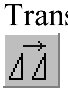
Coordinate System



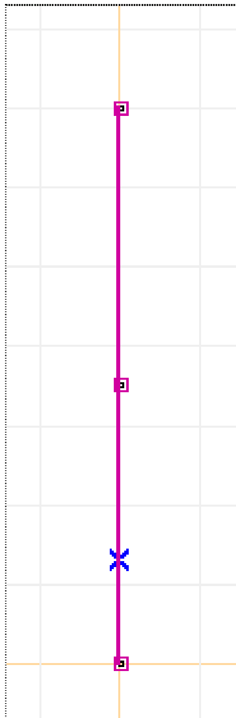
Switch to Z-Y plane.
You should see this picture:



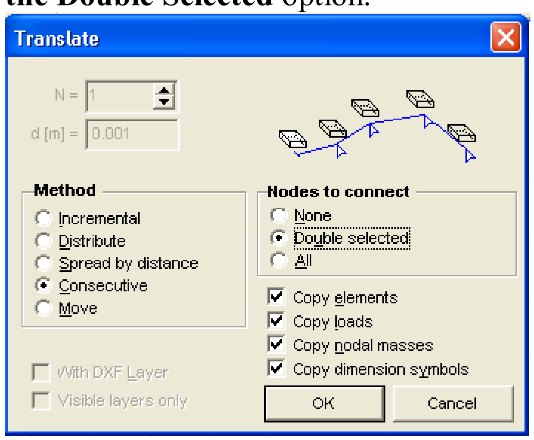
Translate



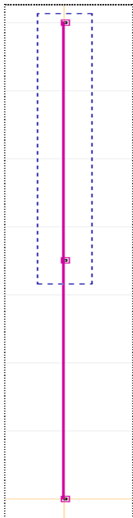
Select the Translate Icon so you can Copy this part of the model geometry structure. In the Selection Icon bar use the **All command** (the asterisk). The selected elements color will change:

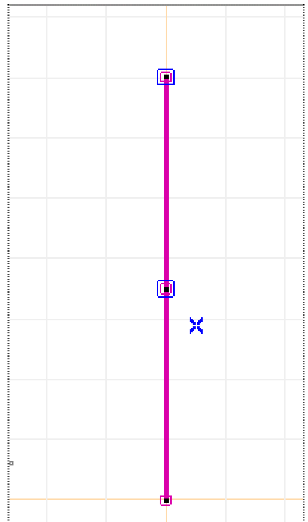


Finishing the selection with **Ok**, in the dialog window **select the Consecutive method**, then in the Nodes to Connect panel **select the Double Selected** option.



Close this dialog window with **Ok**. Now you must select the nodes to connect. **Use a selection window** according to the picture below on the left. The picture on the right shows the result of your selection:





Specify the first displacement vector by **entering the following values**:

X=0
Y=5
Z=0, enter.

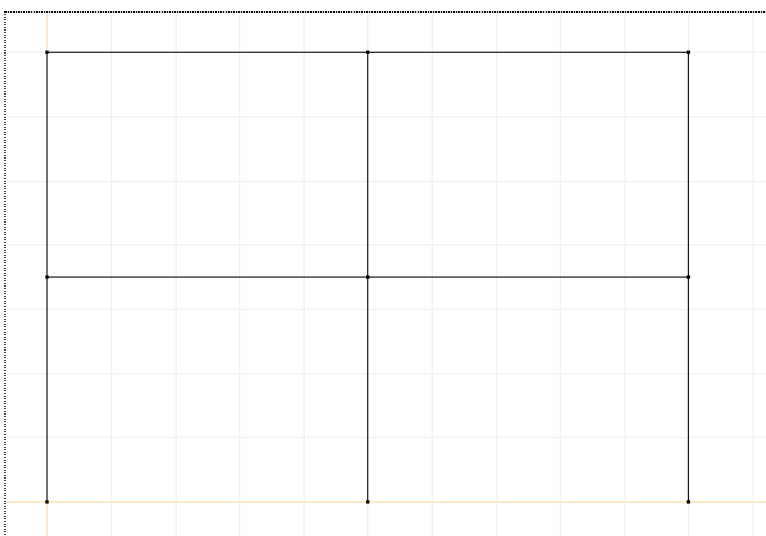
Enter the second vector:

X=0
Y=5
Z=0, enter.

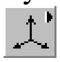
Enter the third vector:

X=0
Y=5
Z=0, enter.

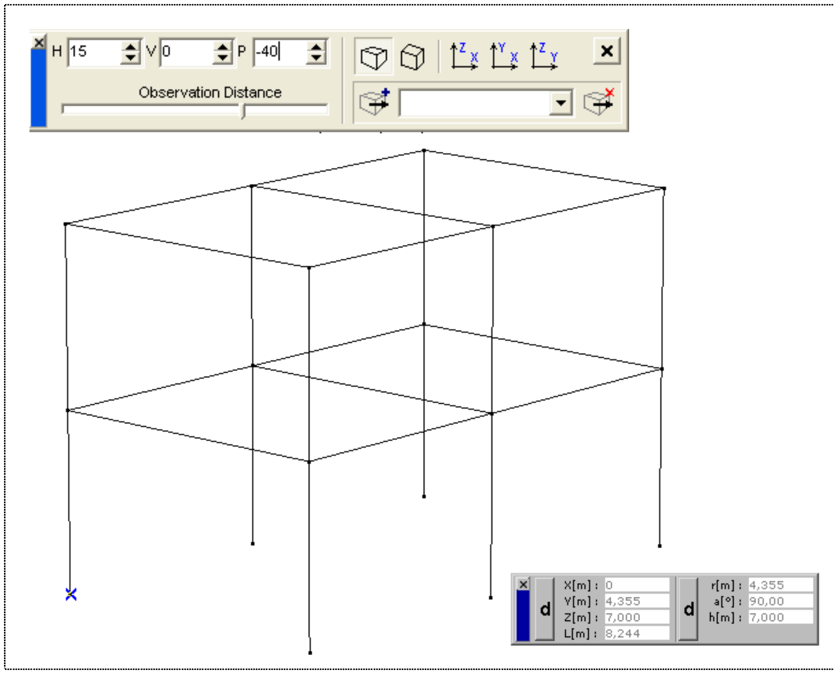
Esc twice to exit from the command. The following picture will be seen:



Coordinate System

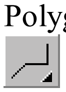


Switch to perspective View. The colums should be on the vertical Z-axis. Use the pan function as needed to bring the model to this perspective.

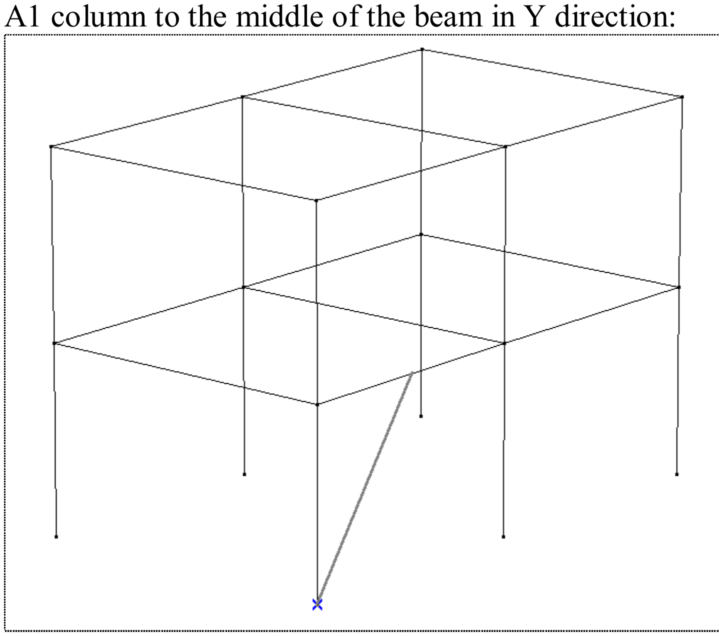


When you **close the dialog bar** this settings will remain active.

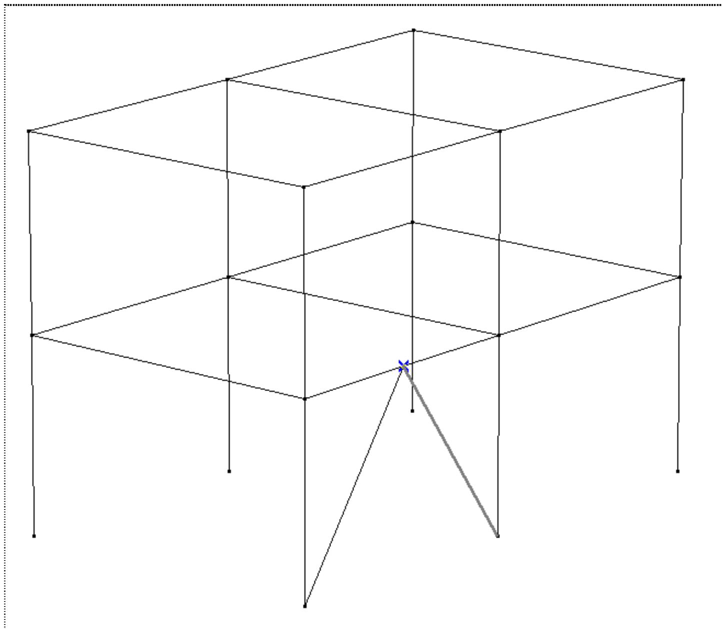
Polygon



Click the Polygon Icon. **Draw a segment** from the bottom of A1 column to the middle of the beam in Y direction:

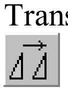


Continue with a segment to the bottom of the middle column:

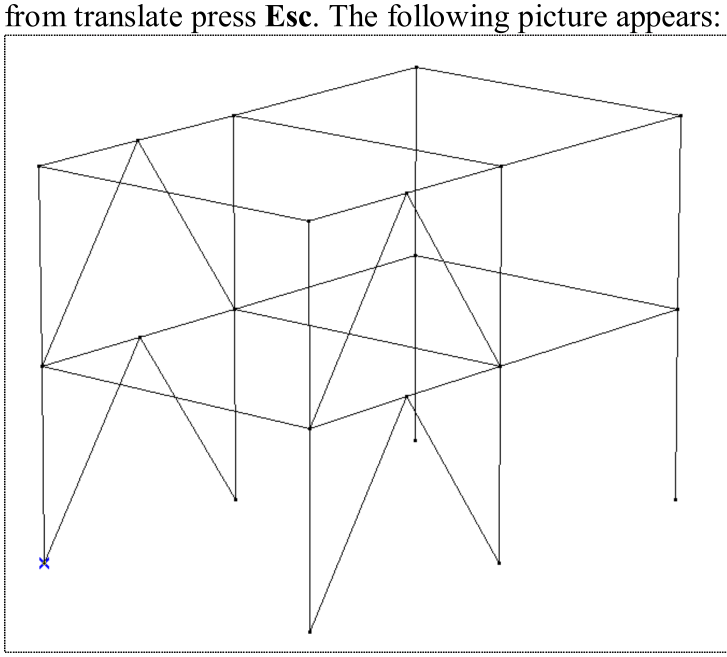


Press **Esc** twice to exit from the command.

Translate

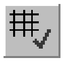


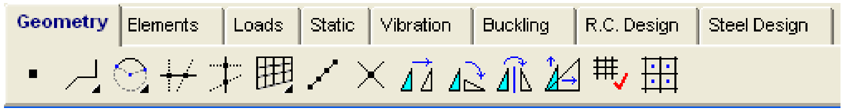
Click the Translate Icon, **select the two inclined bars** then finish the selection with **Ok**. In the dialog window **select the Consecutive method,** and **set the Nodes to Connect** to None. After closing the dialog window with **Ok**, **click on the bottom node** of the A1 column, **then on the middle node** of the A1 column. This will copy the two inclined bars to the upper story. **Copy the bars** on the other side of the structure as well. To exit from translate press **Esc**. The following picture appears:



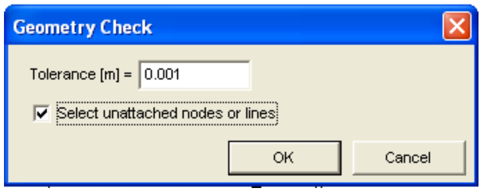
Geometry Check

Check the geometry of the structure with the Geometry Check Icon, which is toward the end of Geometry Toolbar:

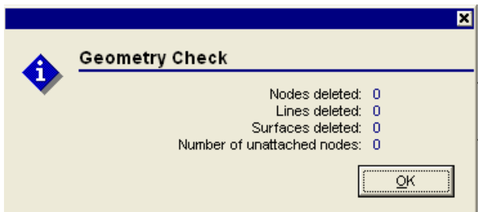




In the dialog window you can set the maximum tolerance for merging nodes, and you can specify whether to search or not for unattached nodes or lines.

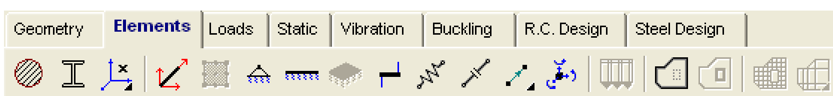


When the check is finished a summary will appear.

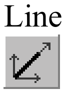


Elements

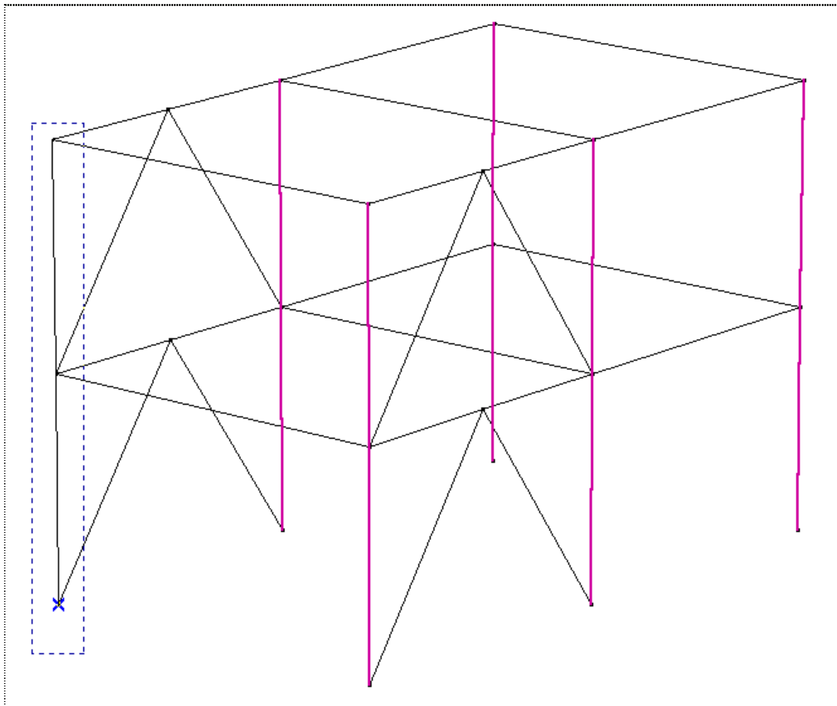
The next step is to create the finite elements. For this **click on the Elements tab.**



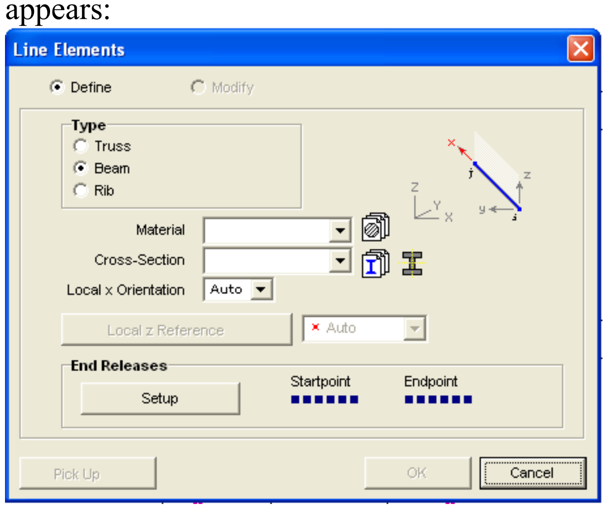
Line Elements



To create the finite elements for beams or columns use the Line Elements Icon. To define the materials for the Columns, **Click the icon**, then **select the the vertical lines** (all columns) by clicking them or by selection windows as in the picture.

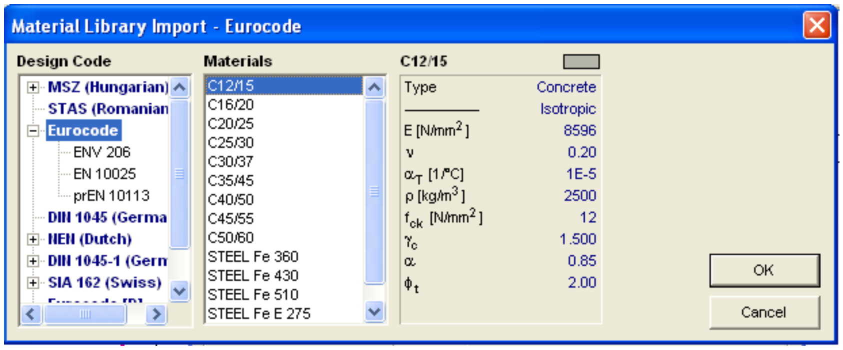


Finish the selection with **Ok**, and the following dialog window appears:



Material Library Import

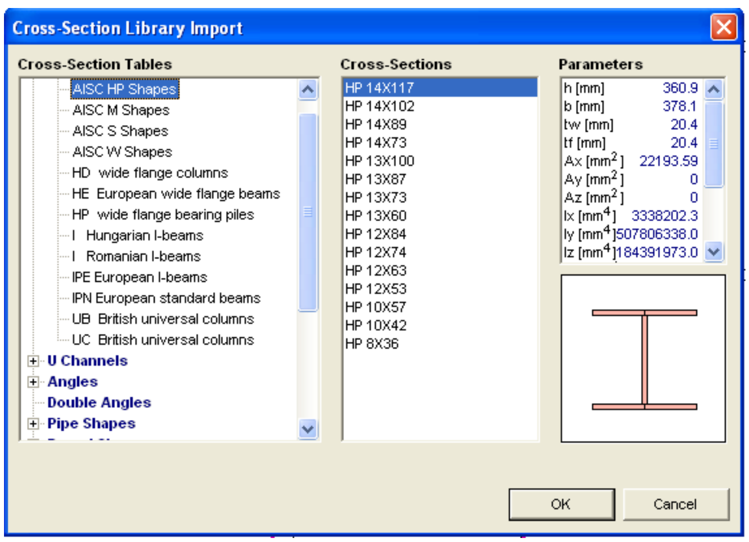
Click the Browse Material Library Icon in the row labeled Material. The following dialog window appears:



Select Steel Fe360 as the active material.

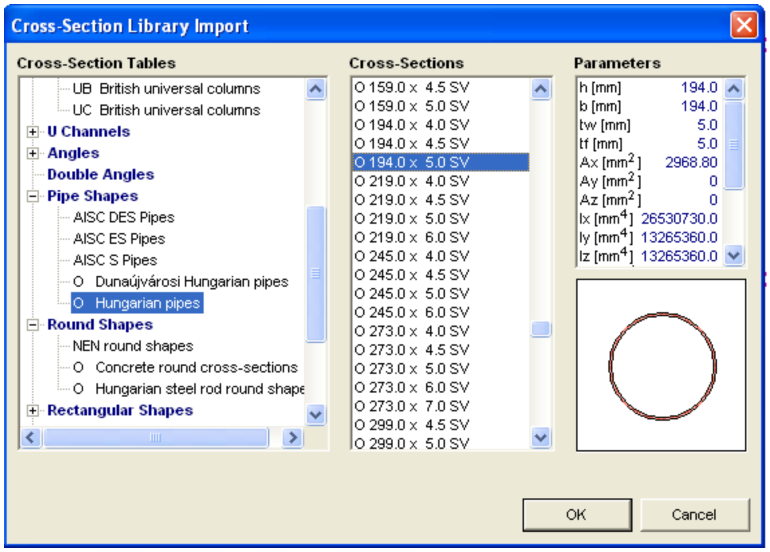
Cross-Section Library Import

Click Cross-Section Library Import Icon. The following dialog window appears :

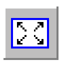


Select from the Cross-Section Tables **I Hungarian Beams**, then from the Cross-Section List **I-400**. Close the dialog window with **Ok**.

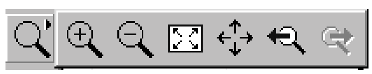
The default value for the Local z Reference is Auto. This means that local x reference of the beam will be along the axis of the element, while local z reference will be parallel with global Z.
Finish the creation of column (beam) elements with **Ok**.
Define the material for the horizontal beams in a similar way, but **use I-360** for their cross-section.
Next, **define** the material for the diagonal braces and **use Hungarian Pipes** O194.0 x 5.0 SV as cross-section.



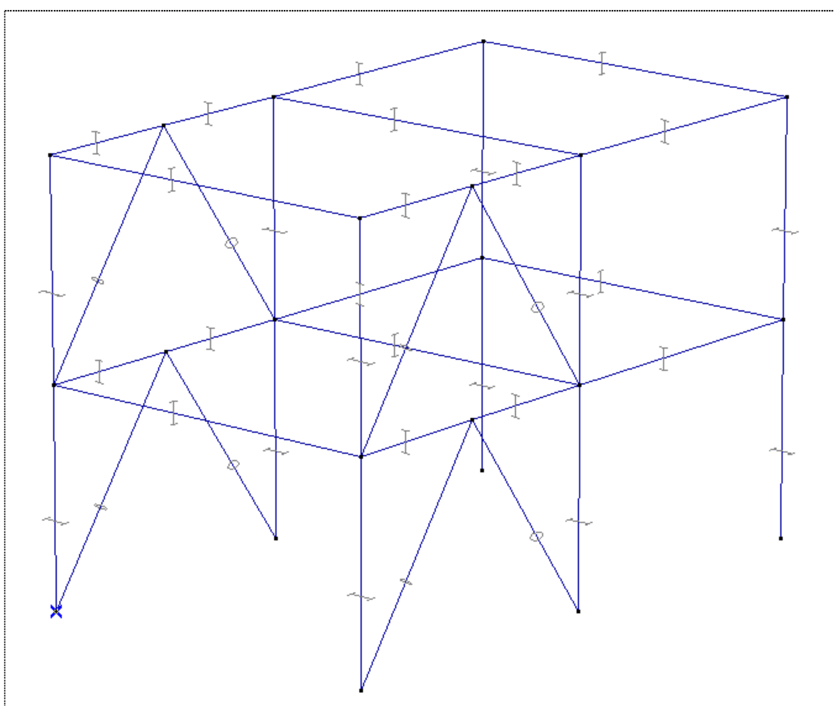
Zoom to Fit



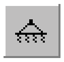
For a better overlook let’s **click** the Zoom to Fit Icon on the Zoom Icon bar.



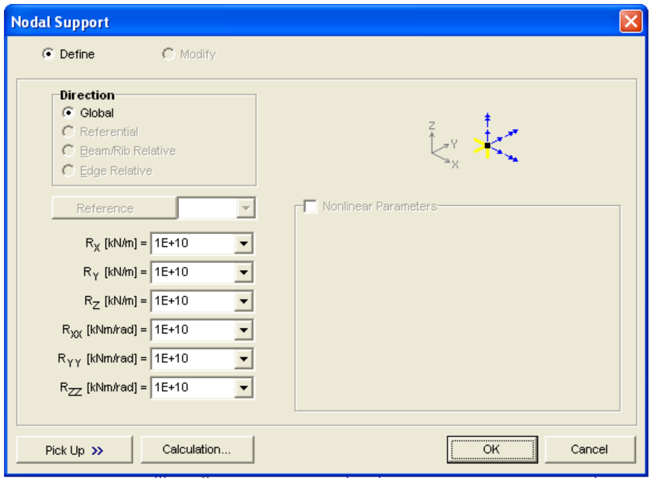
The following picture appears:



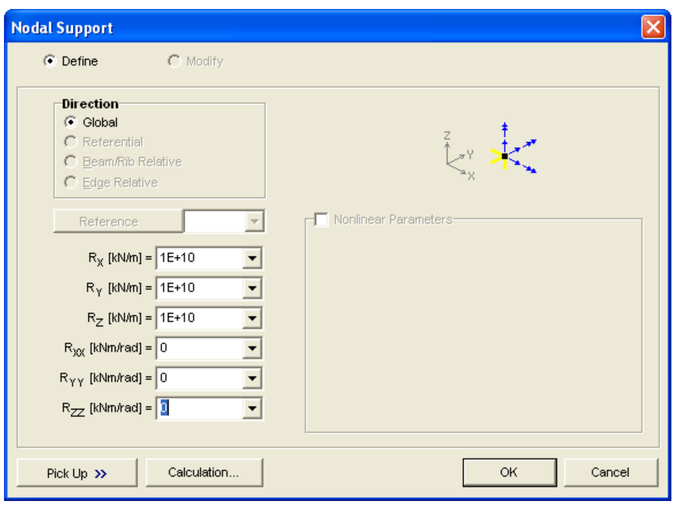
Nodal Support



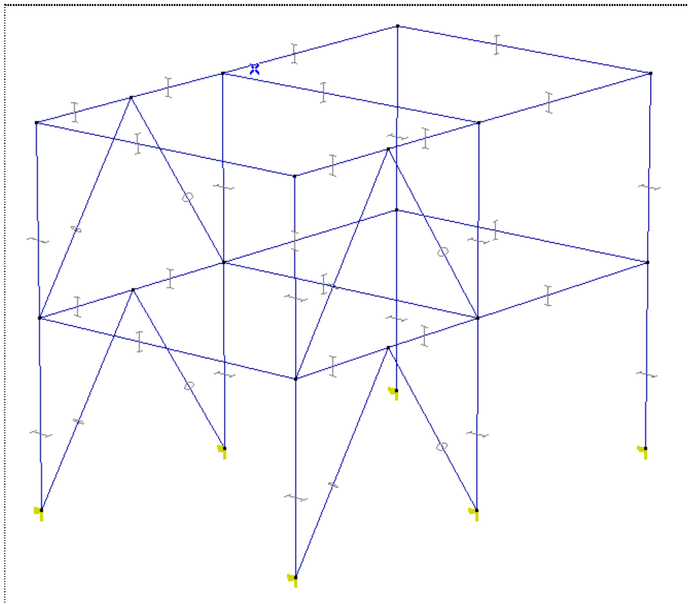
Click the Nodal Support Icon, **select** all 6 column's bottom node and finish the selection with Ok. The following dialog window appears:



In this dialog window you can set the node support conditions. Let's assume pinned supports in all these nodes, so **set the rotational stiffness** Rxx, Ryy, Rzz to 0.



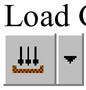
Finish the creation of nodal supports with **Ok**, and the support symbols will appear.



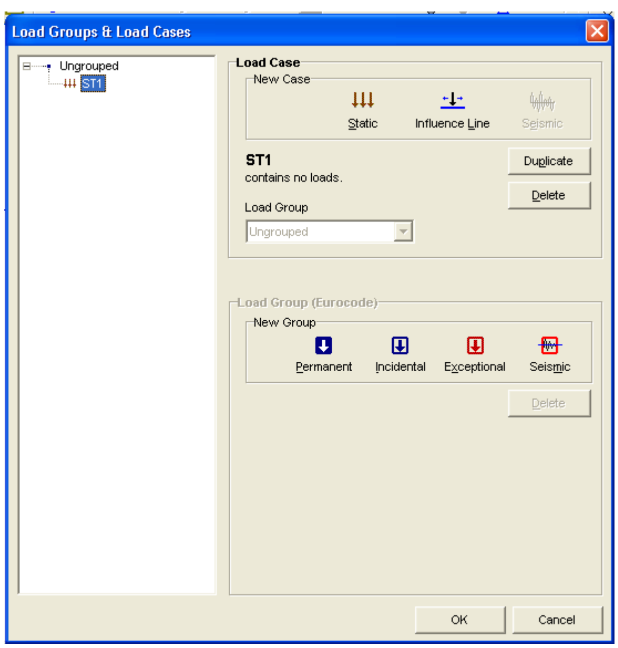
Loads

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

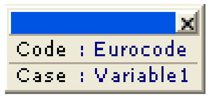
Load Cases & Load Groups



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

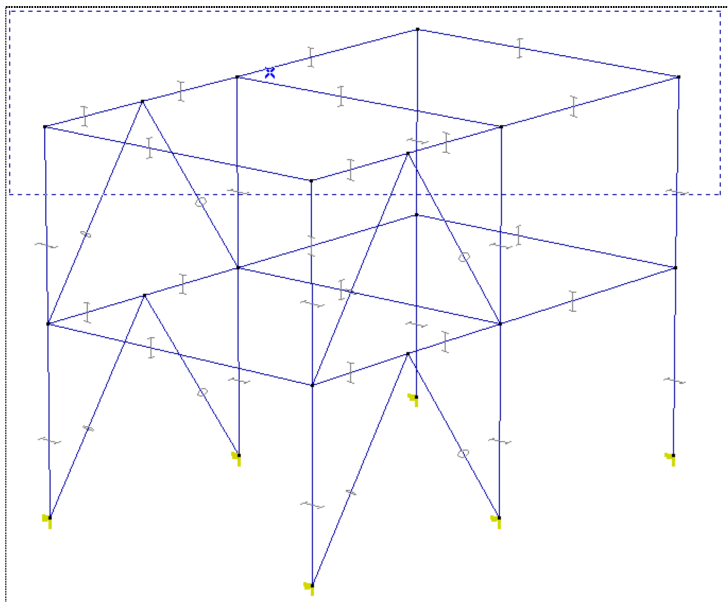


Click on the ST1 (the first static load case) in the upper left corner, and **rename** it to VARIABLE1. Close the dialog window with **Ok**, and VARIABLE1 will be the current load case. You can see in the Info Window the name of the current load case:

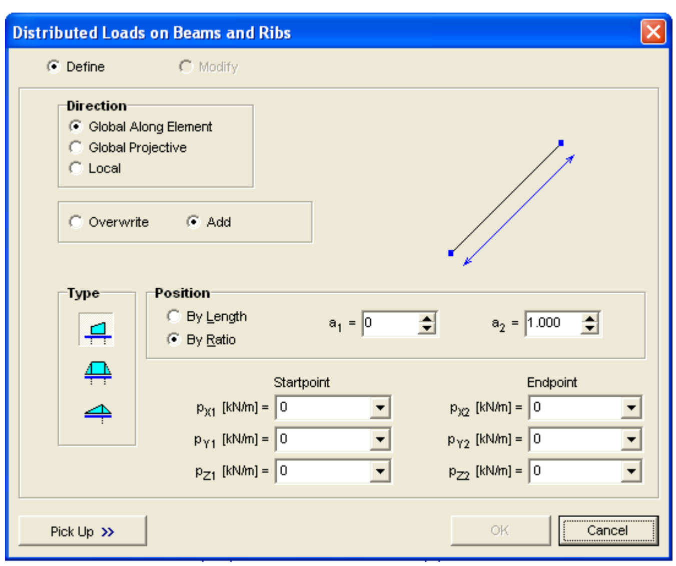


Line Load

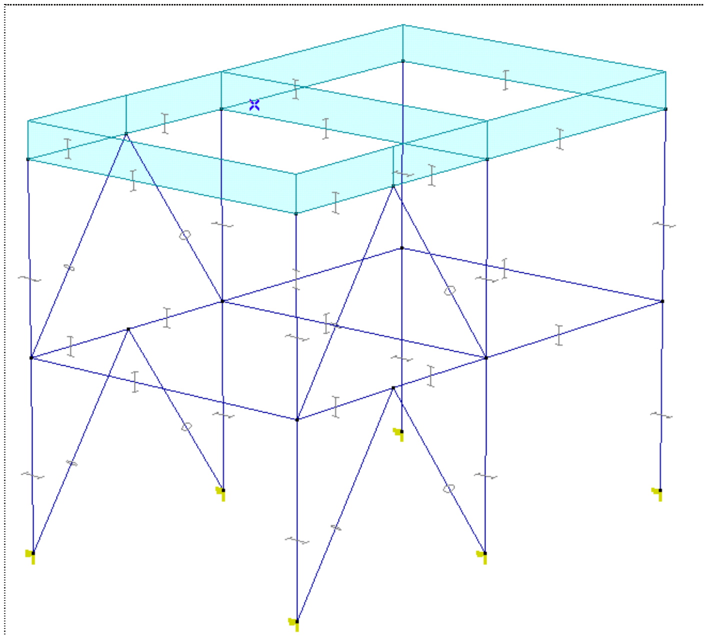
Let’s apply loads on the horizontal beams. **Apply** on the lower beams 50 kN/m, on the upper beams 25 kN/m. For this **click** the Line Load Icon, then **select** the upper beams with a selection window.



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

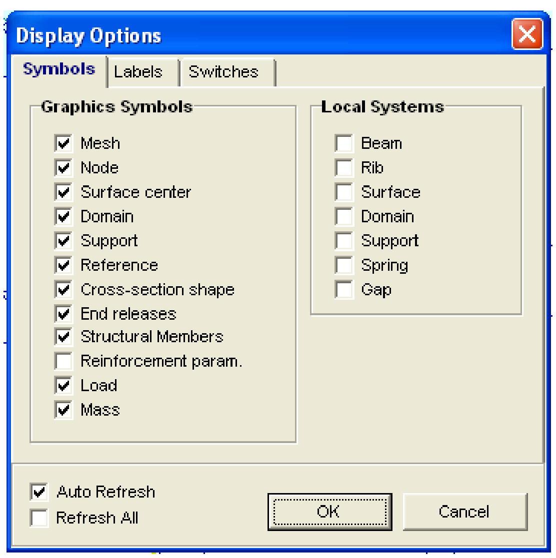


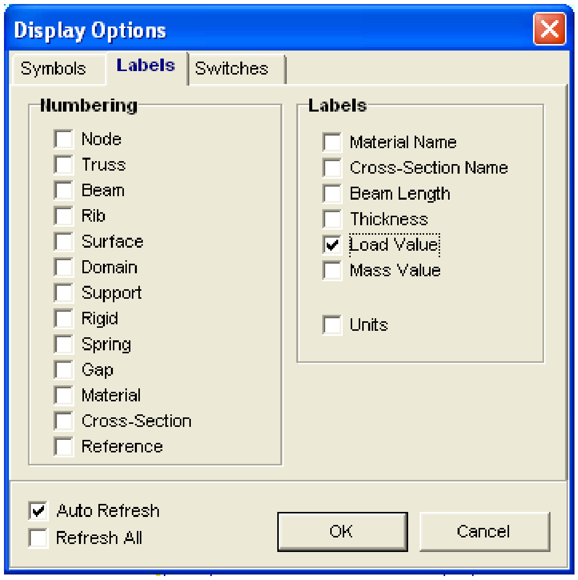
Type -25 in the pz1, pz2 edit boxes, then close the dialog window with **Ok**. The following picture appears:



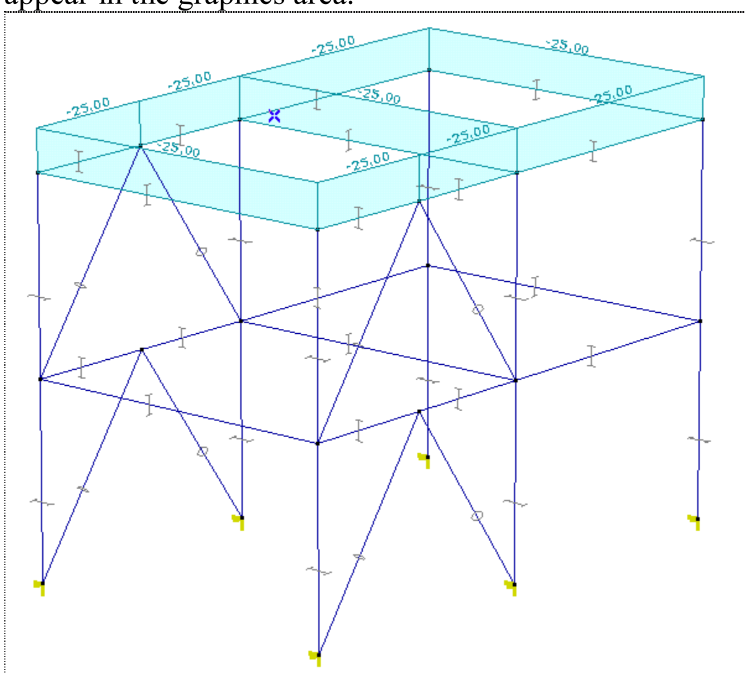
Display Options

Click the Display Options Icon in the Icons Menu. The following dialog window appears:
Select the Labels tab, then **check** the Load Value box:

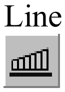




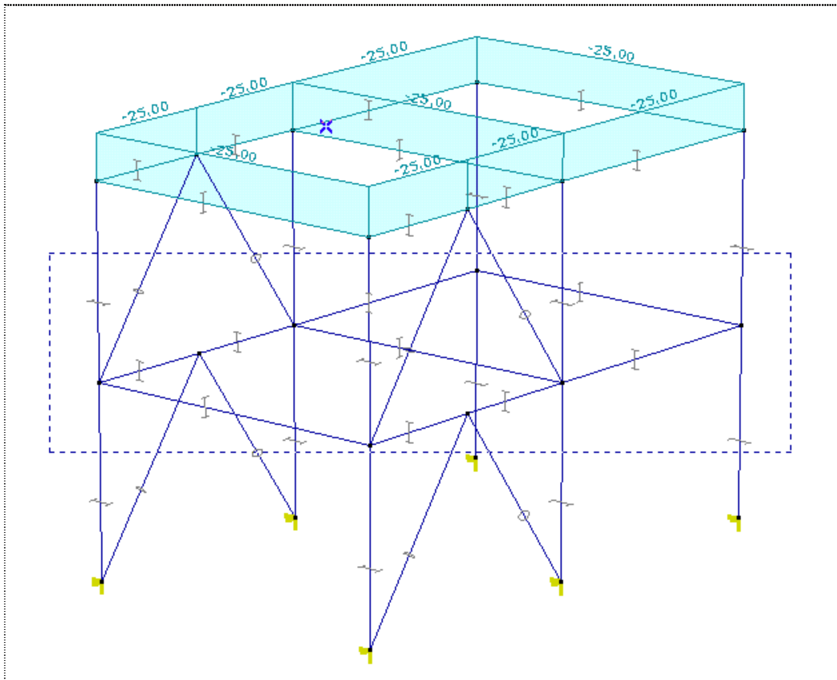
Close the dialog window with **Ok**, and the load values will appear in the graphics area.



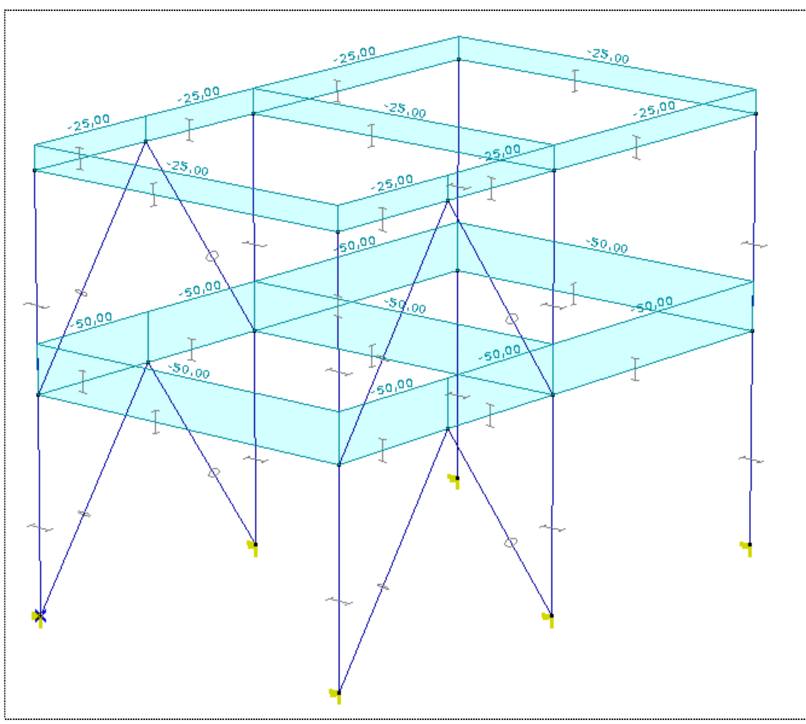
Line Load



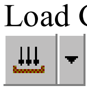
Click the Line Load Icon, and **select** the lower horizontal beams:



Finish the selection with **Ok**, then **type** -50 in the pz1, pz2 edit boxes. Close the dialog window with **Ok** and the following picture results:



Load Cases & Load Groups

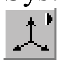


Click the Load Cases & Load Groups Icon.

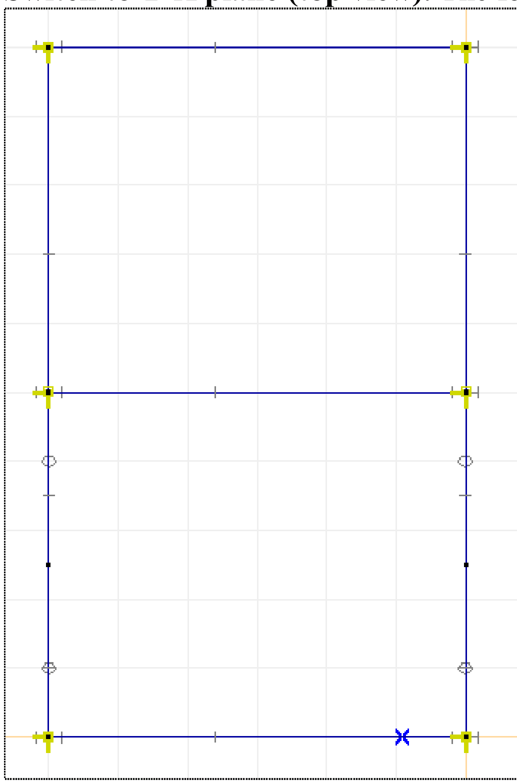
New Load Case Static

In the New Case panel **click the Static Icon** and **name the load case** WIND. Close the dialog window with **OK**. All previous loads ’disappeared’, and the current load case’s name in the Info Window is WIND.

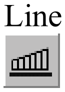
Coordinate System



Switch to Y-X plane (top view). The following picture appears:

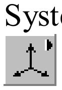


Line Load

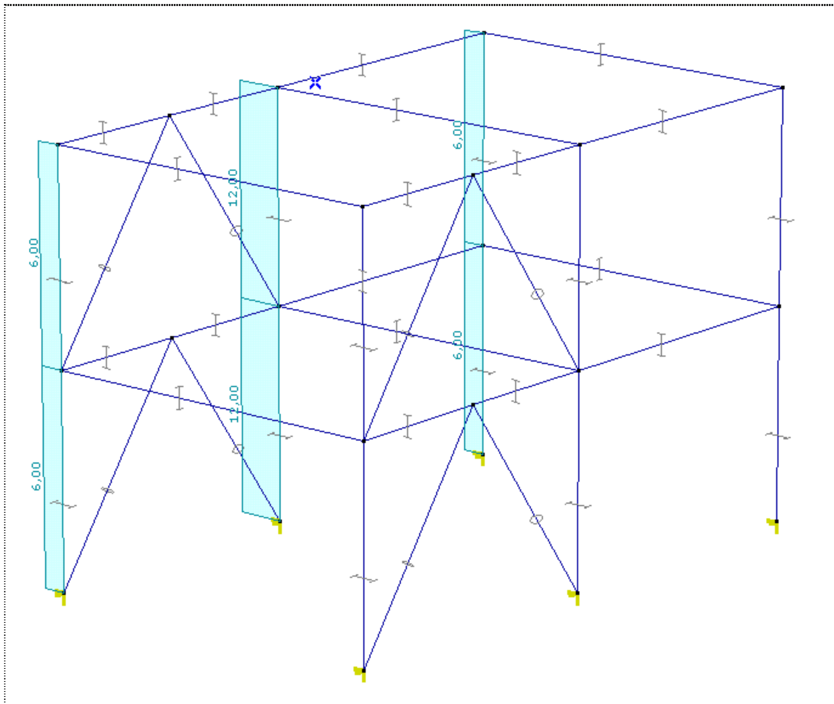


Click the Line Load Icon, and **define** on the upper left columns a load of intensity 6 kN/m in x direction. From the top view select the upper left node with a selection windows (thus selecting everything inside the selection window, including the two columns). Finish the selection with Ok, then **type a load intensity value** of 6 in px1, px2 edit boxes and close the dialog window. **Repeat the above step** for the bottom left node. **Repeat the above step** for the middle left column, except type a load intensity value of 12.

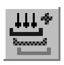
Coordinate System



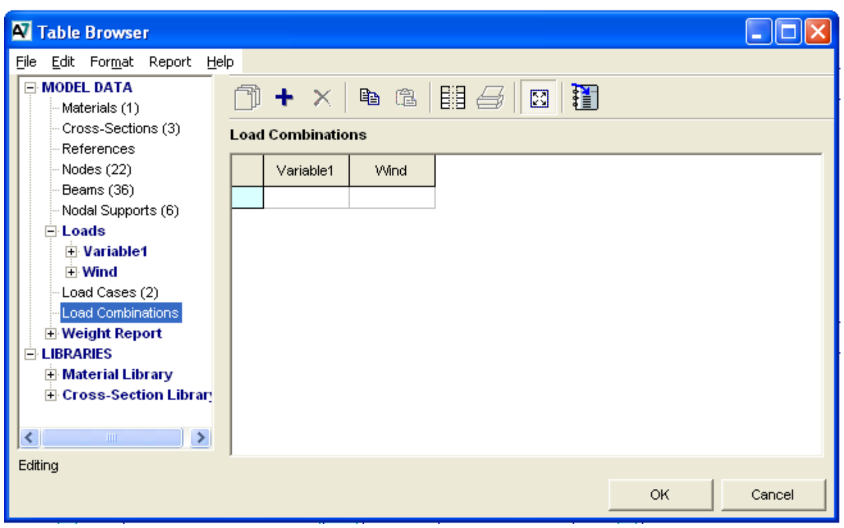
Switch to Perspective View. The following picture appears:



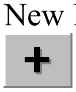
Load Combinations



Let’s create a load combination. **Click** the Load Combinations Icon, and the Table Browser will appear.



New Row



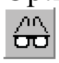
Use the New Row Icon to add a new load combination. You have to specify a factor for each load case in a load combination. Let’s assume the following factors. **Type in these factors** in their columns:

VARIABLE1	1.2, Enter
WIND	1.2, Enter

Accept the new load combination(s) by closing the Table Browser with **Ok**.

Now the preprocessing part of the example is finished.

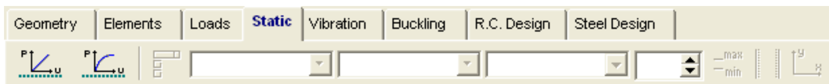
Display Options



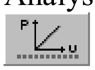
Click the Display Options Icon, and **uncheck** the Node, Cross-Section Shape, Load boxes in the Symbols tab, andthe Load Value box in the Labels tab.

Static

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



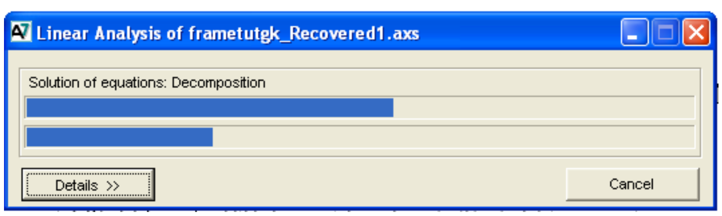
Linear Static Analysis



Click on the Linear Static Analysis Icon.

A Model Save Dialog will appear if you haven't already assigned a name for the model. Accept save and a Save dialog window appears, where you can specify the model filename and path.

During the analysis the following window appears:

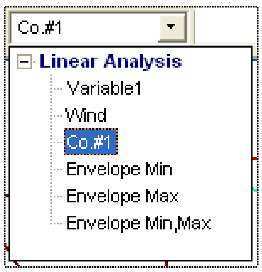


Details

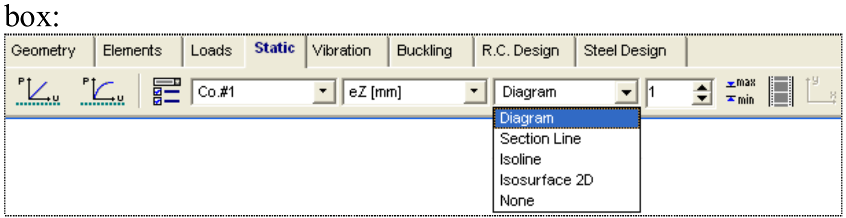
If you click the details button to view details of computation, the topmost label shows the current computation step, the upper bar shows its progress. The lower bar shows the global progress of computation. The estimated memory requirement shows the estimated virtual memory demand. If the virtual memory of the computer is set to a lower value, an error message will appear. When the computation has finished, the two progress bars will disappear. Close the window with **Ok**.

Static

By default the postprocessor will start with the ez displacement of the first load case, which is now VARIABLE1. The display mode will be iso surface. **Change to isoline** display. You will see the displacements from the VARIABLE1 load case in global Z direction. To view the results from the load combination **select Co. #1** in the Case Selector combo box.

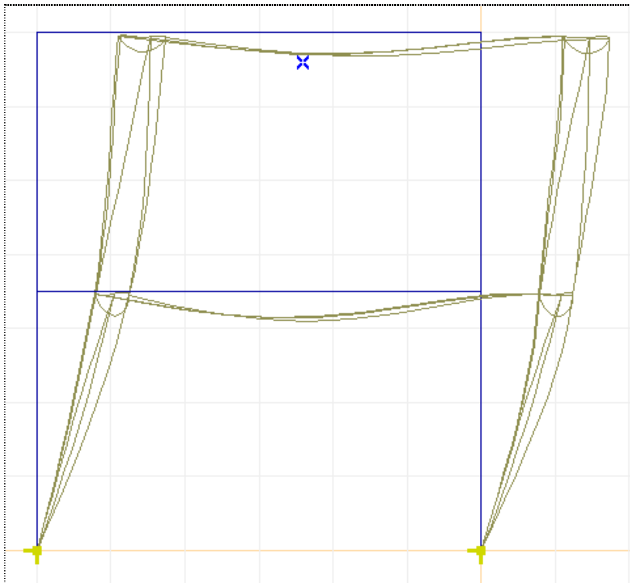


Switch from Isoline to Diagram by **Clicking** the Result Display Parameters Icon and **select** Diagram in the Display Mode menu box:

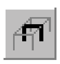


Coordinate System

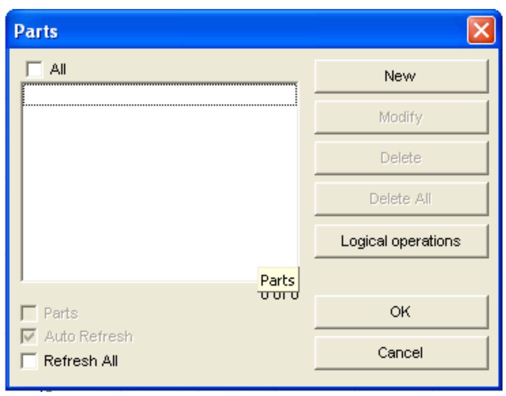
Switch to Z-X plane. The following picture appears.



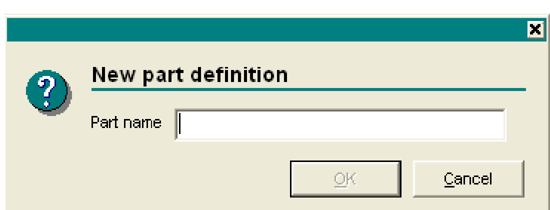
Parts



Click the Parts Icon on the left Icons Menu. The following dialog window appears.

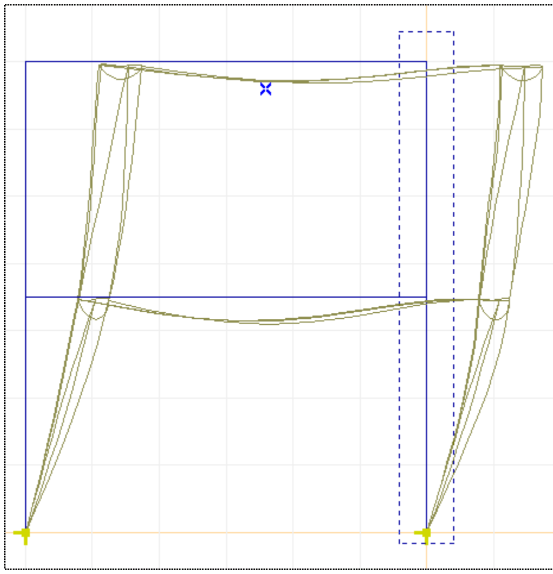


Click the New Button, which brings up a window where you can specify the name of the part.

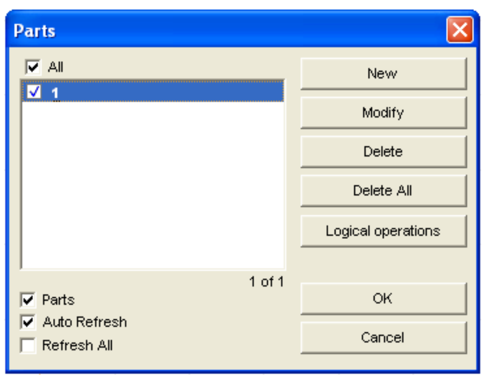


Type in 1 and close this window with **Ok**.

You have to select the entities which will make up the part named 1. **Select** the right columns with a selection window according to the following picture.

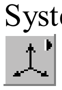


Finish the selection with **Ok**. The dialog window will reappear as in the picture below.



Close the dialog window with **Ok**, and part 1 will be accepted.

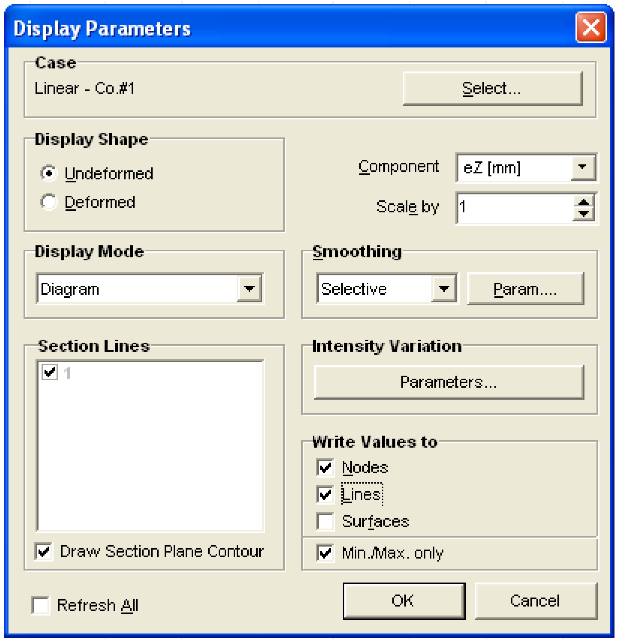
Coordinate System



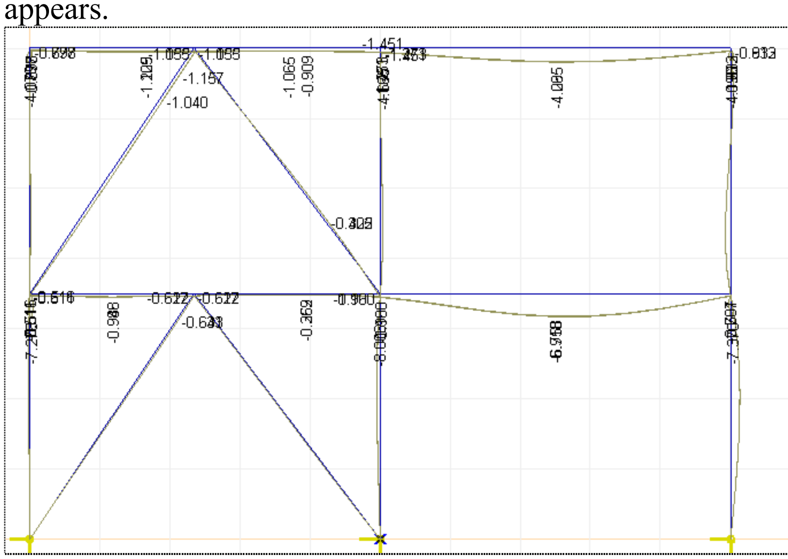
Switch to Z-Y plane.

Result Display Parameters

Click the Result Display Parameters Icon, and **check Nodes and Lines** in the Write Values to box.

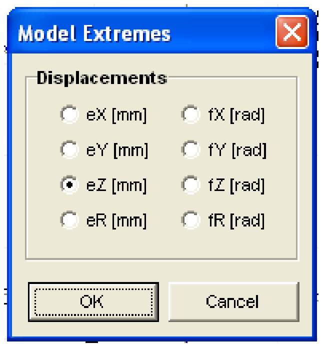


Click **OK** to close the dialog window, and the following picture appears.

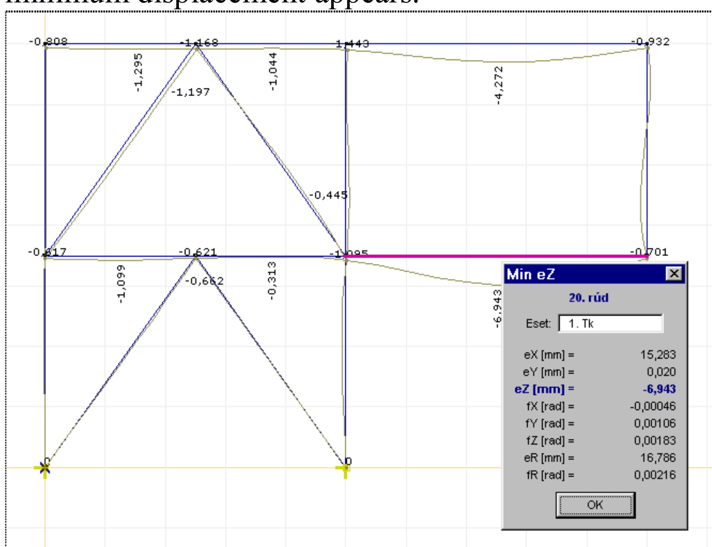


Min/Max Values

Click the Minimum and Maximum Values Icon to find out the location of maximum displacement. The following dialog box will appear:

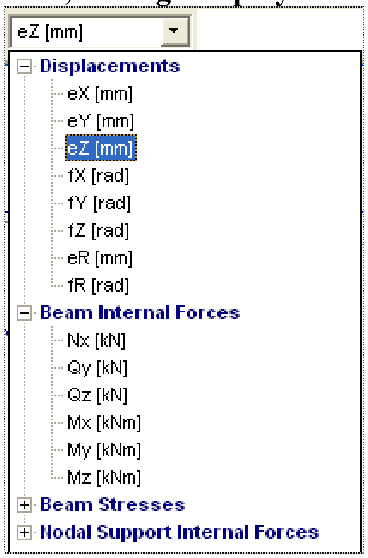


Here you can select one displacement component. **Leave it on ez** and click **Ok**. First the location and value of the negative minimum displacement appears.

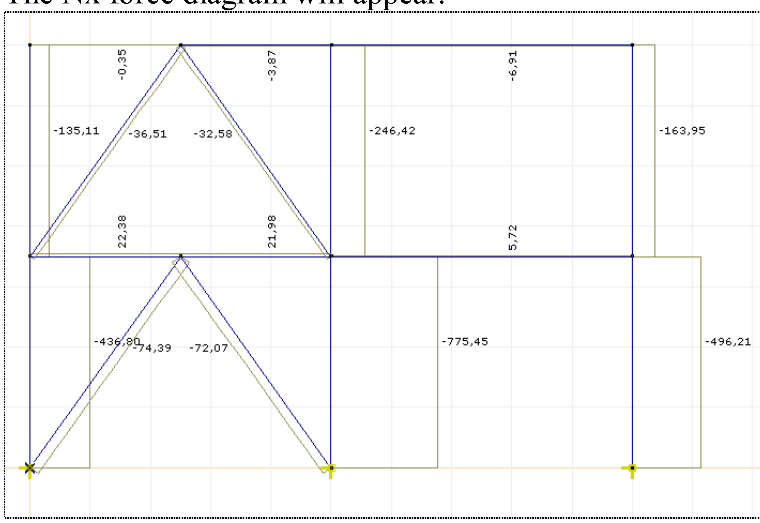


Click **Ok**, and the location and value of positive maximum displacement will appear.

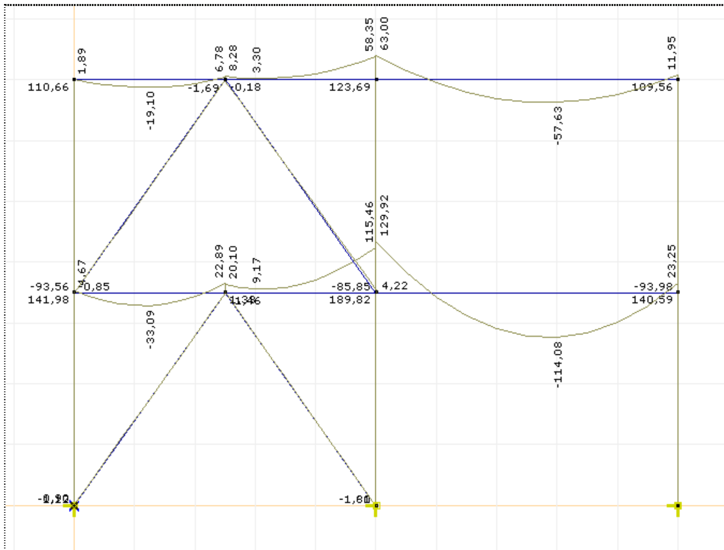
Select from the Result Component combo box Nx from the Beam Internal Forces. **Click the Result Display Parameters Icon, Change display to section line.**



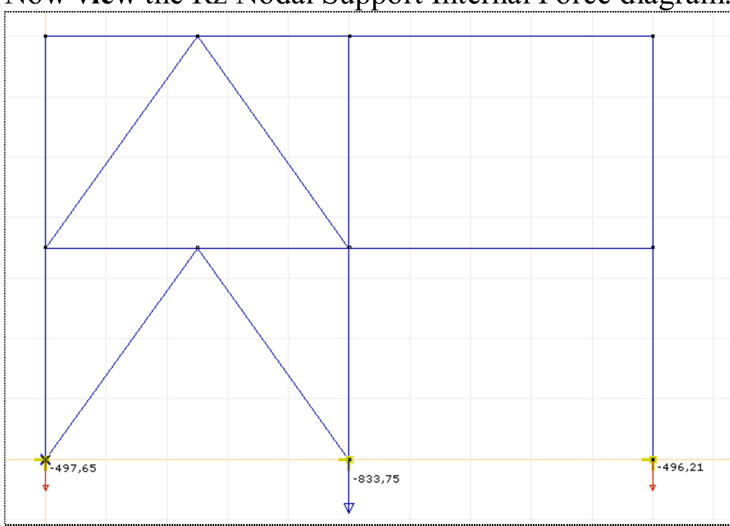
The Nx force diagram will appear.



View the My moment diagram in a similar way.



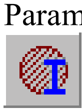
Now **view** the Rz Nodal Support Internal Force diagram.



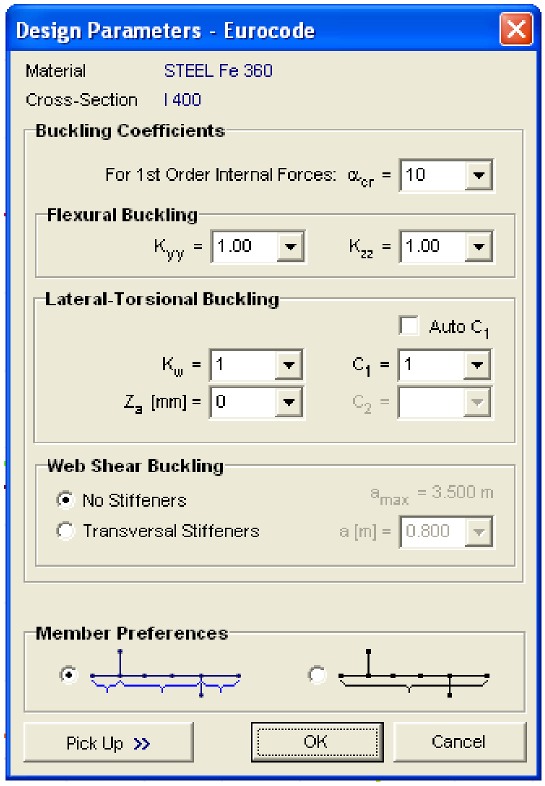
Steel Design

Click the Steel Design tab to start the checking of column A1.

Design Parameters



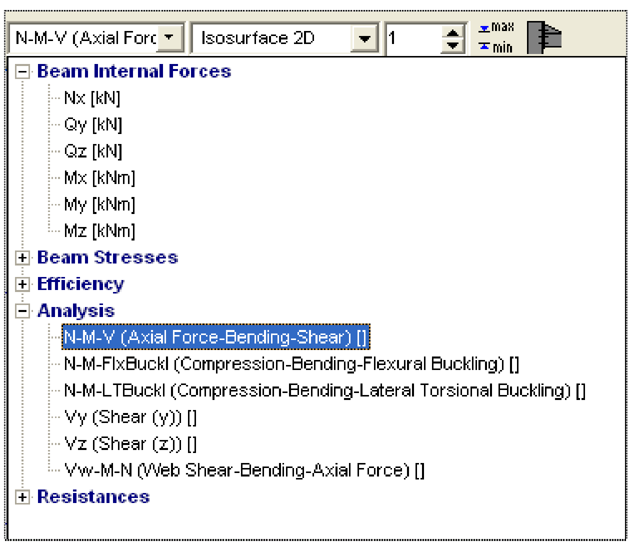
Click the Design Parameters Icon, then **select** column A1 and finish the selection with **Ok**. The following dialog window appears:



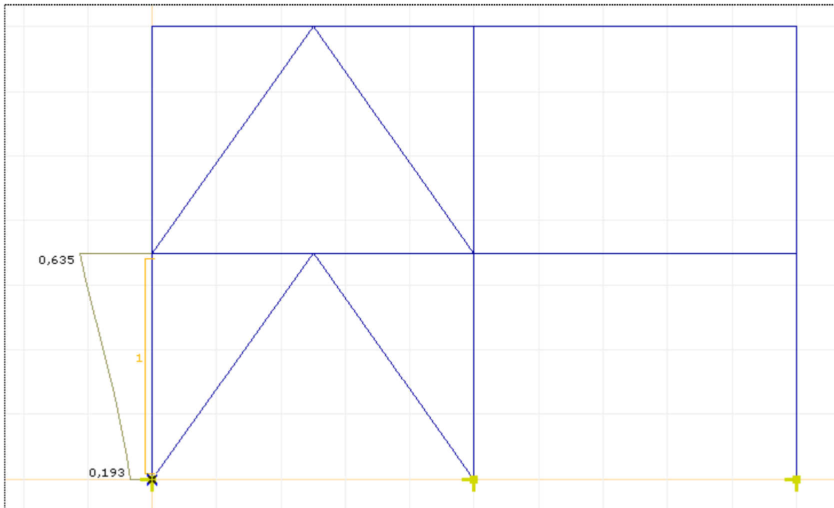
Overwrite Kyy with 1.25, and then close the dialog window with **Ok**.

Axial Force-Bending-Shear

Let's **view** the N-M-V diagram.

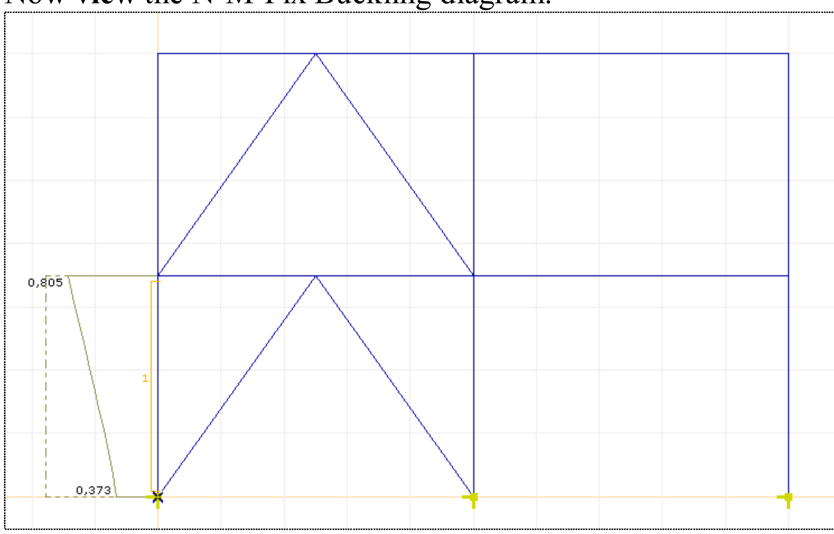


The following picture appears:

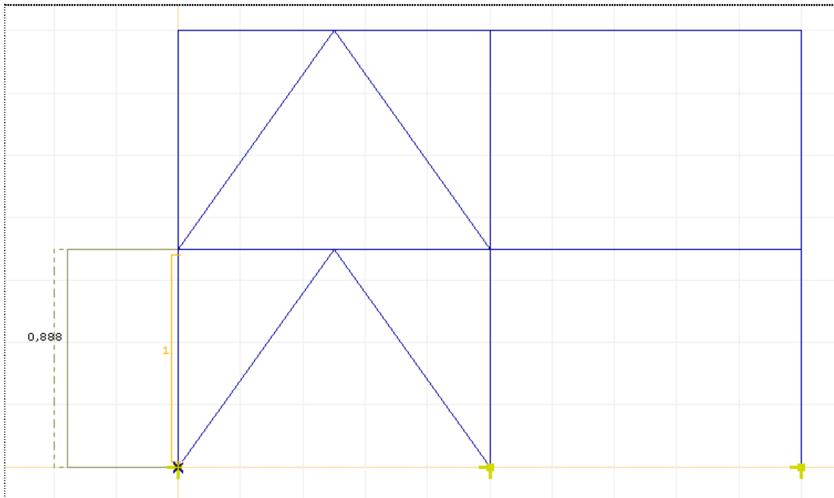


Buckling

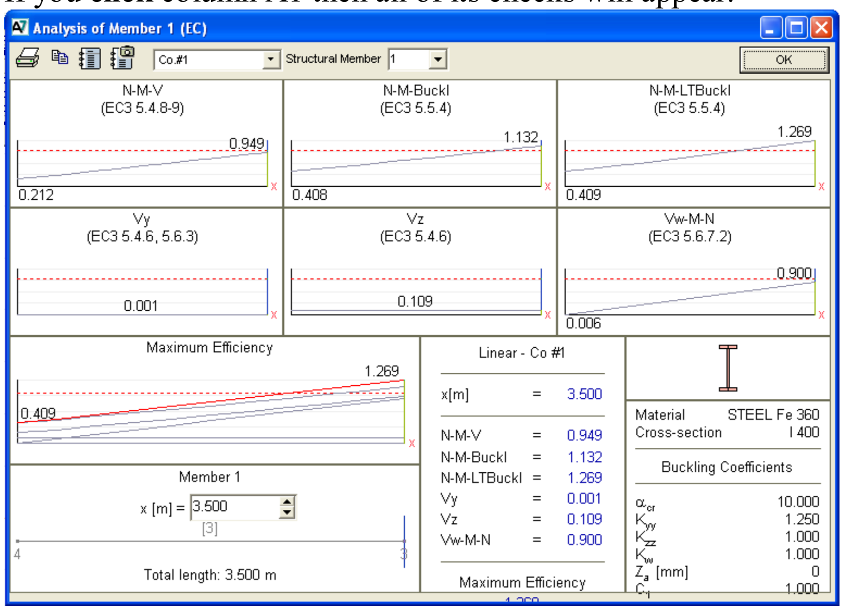
Now **view** the N-M-Flx Buckling diagram:



Choose the efficiency diagram. The following picture appears:



If you **click** column A1 then all of its checks will appear.



Click **Ok** to close this window.