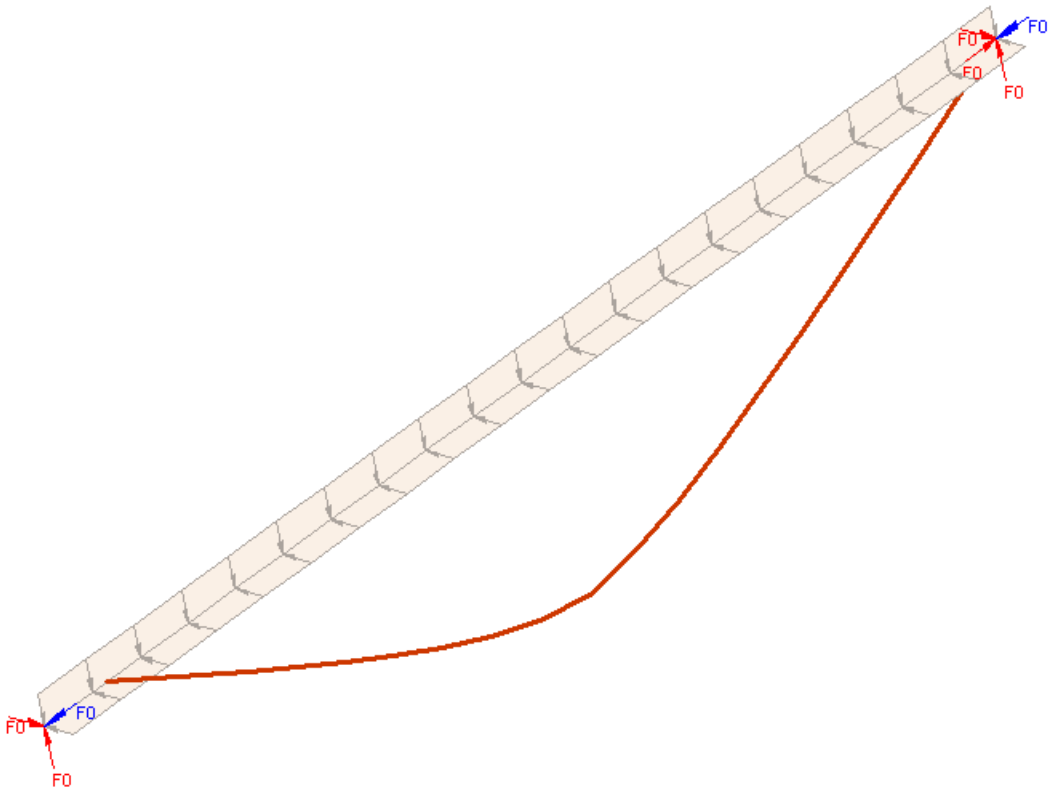
Example for GID-SAFIR 3D Structural Analysis

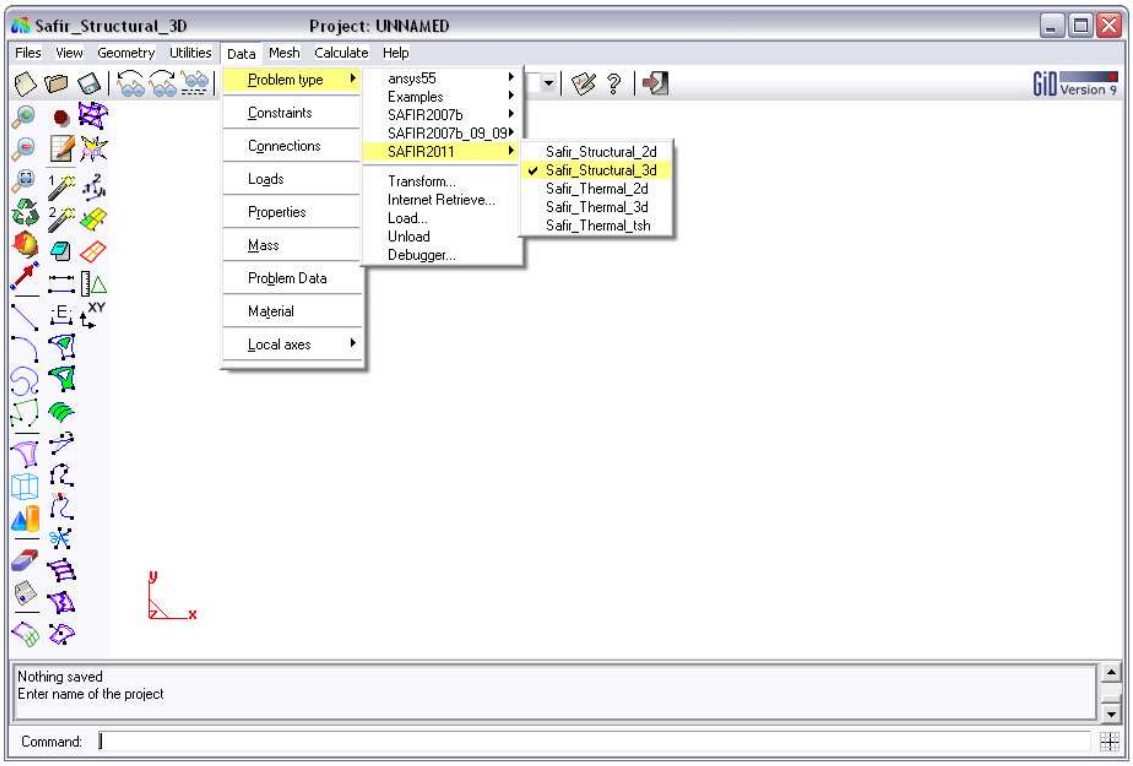
Exercise n°6 - Beam 3D



1. Create a new project of type Safir_Structural_3d

From the pull down menu select:

- ***Data->Problem type->SAFIR20011->Safir_Structura_3d***



To save the project select (or use icon on the left):

- ***Files->Save*** ***or***  ***or [Ctrl + s]***

⚠ *If Caps lock is active on your keyboard, shortcut don't work*

Enter a file name, eg.: ***Beam3D***

GiD creates a directory with the name ***Beam3D.gid***

GiD creates a number of system files in this directory.

When you start the SAFIR calculation the SAFIR ***.IN*** and ***.OUT*** file will be created here. Before starting a calculation all ***.TEM*** files must be placed into this directory.

2. Create the system geometry

To change to the 3d isometric view select from the pull down menu:

- *View->Rotate->isometric*

If you want to define a point of view by your own use:

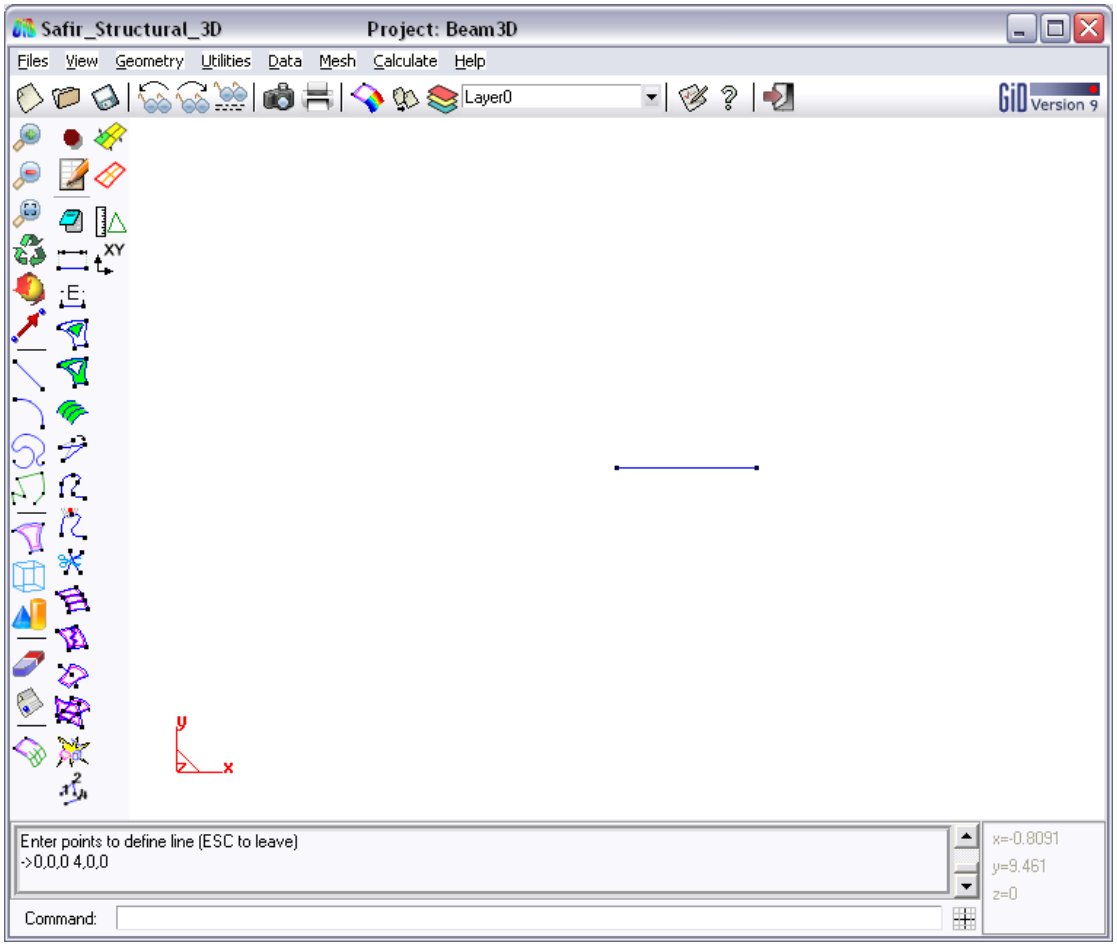
- *View->Rotate->Trackball* *or [F7] or* 

Create the system lines:

- *Geometry->Create->Straight Line* *or* 

Enter in the command line (at the bottom of the widows):

0,0,0 4,0,0, and press *[Enter]*



To change the view, select from the pull down menu:

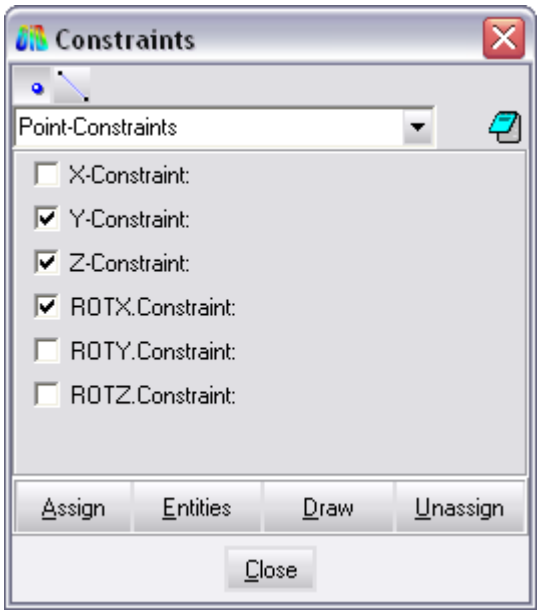
- *View->Zoom->Frame* *or [F11] or* 

3. Define constraints for the supports:

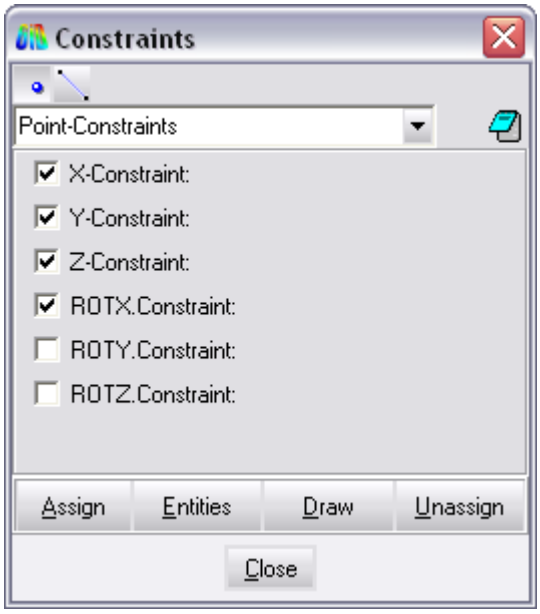
From the pull down menu select

- ***Data->Constraints***

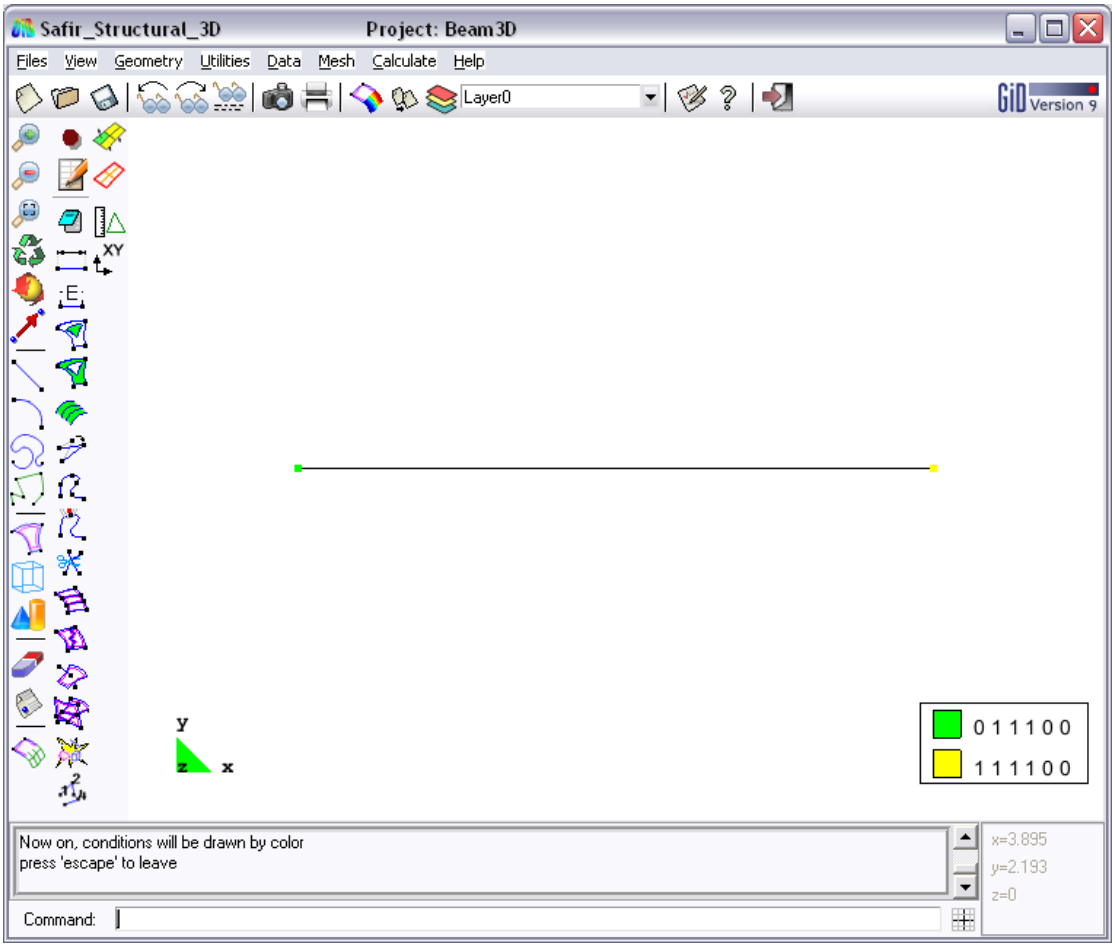
Select y and z constraint and x rotation constraint and assign it to the left points of the beam



Select x,y and z constraint and x rotation constraint and assign it to the right point of the beam.



In the dial box, with ***Draw->Colors*** you can display the constraints.

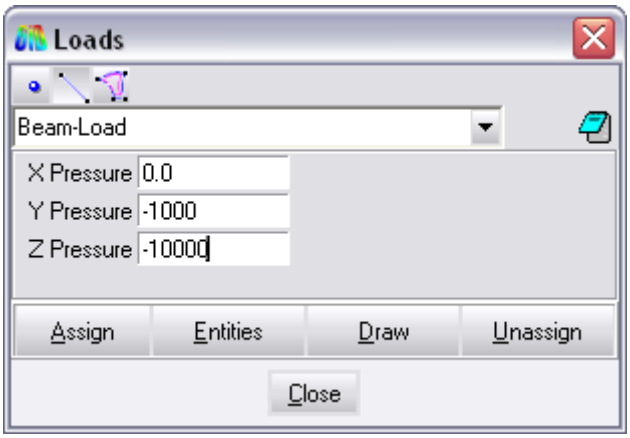


Press ***Finish*** or ***[Esc]*** to leave this view mode

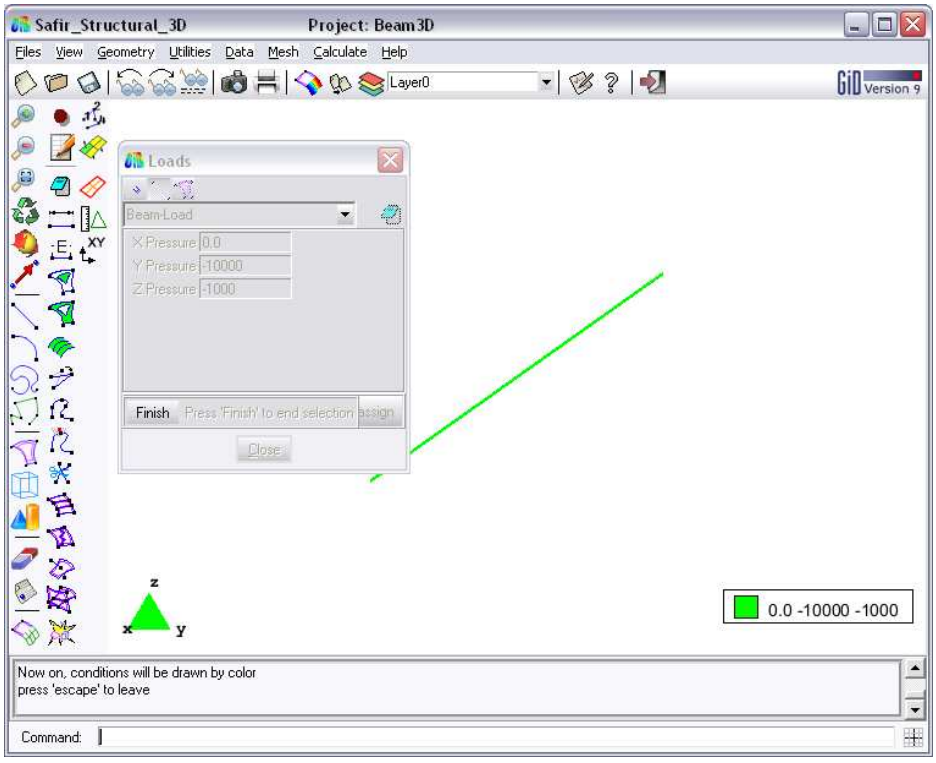
4. Define loads

From the pull down menu select:

- ***Data->Loads***



In the dial box select ***Beam-Load***  and enter a Z-Pressure of ***–10000*** N/m and a Y-Pressure of ***-1000*** N/m and ***Assign*** it to the beam.

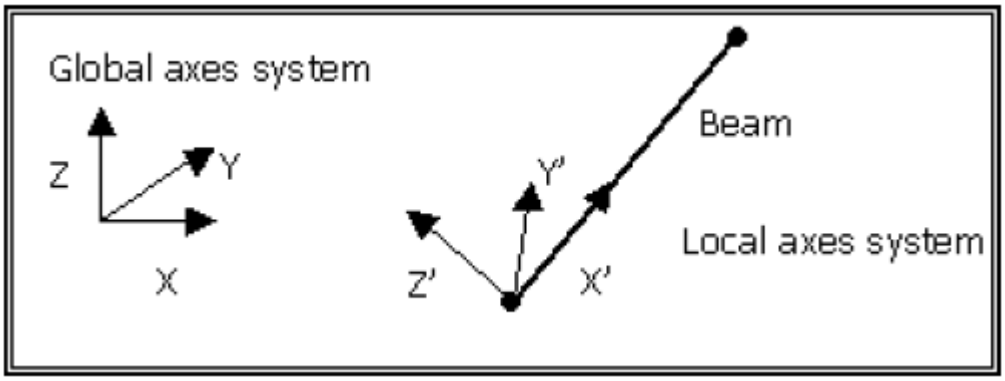


To display the loads select ***Draw->Colors*** in the dial box

Press ***Finish*** or ***[Esc]*** to leave this view mode

5. Create Local axes

Local Axes: The orientation of the cross-section is controlled by defining a local axes X'Y'Z' – system.



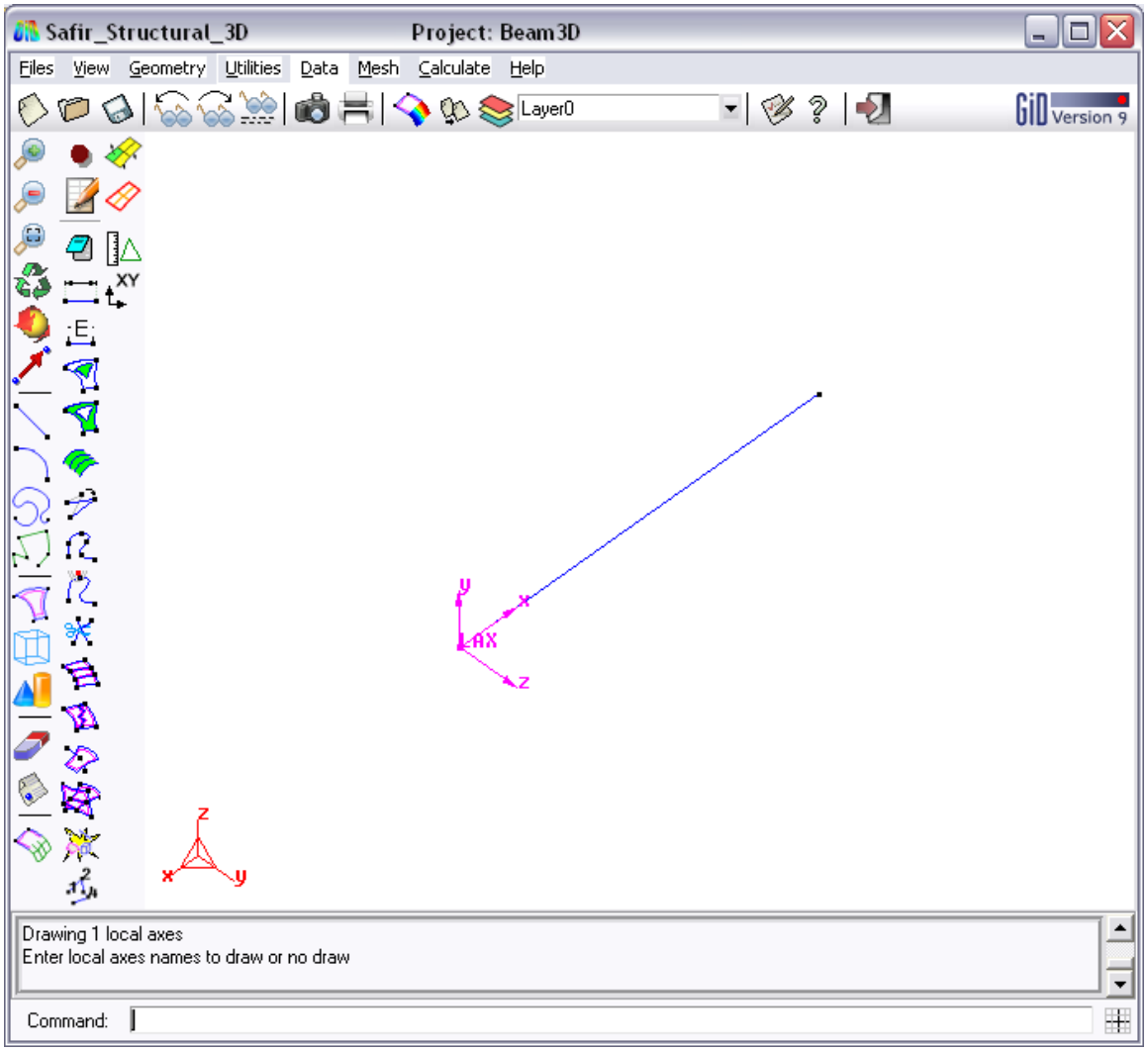
⚠ *Unlike SAFIR which needs a 4th node to describe the orientation of a cross section on a beam, the GiD-SAFIR interface uses a local X'Y'Z' axes system. When you start the SAFIR calculation the GiD-SAFIR Interface creates the 4th node in the X'Y' plane. If the center of the local axes is not located on the system line of the beam, the direction vector of the Y'-axis is used together with the starting point of the beam to define the 4th node. However the GiD-SAFIR interface will issue a warning message in the View-output window of the calculation run*

The objective is to create one named local axe (LAX) with a X' direction and an angle as shown in the figure below.

From the pull down menu select:

- ***Data->Local Axes->Define***

Enter the new local axe name ***LAX***



Select ***XandAngle***

Press ***[Ctrl+a]*** in order to select the local axe center (left point of the beam) then do the same operation to select a point in the positive x' axe of your local axe (right point of the beam).

Set an angle (90°) in order to get your y' direction (4th node)

To draw local axes select:

- ***Data->Local Axes->Draw all***

6. Assign temperature files (.TEM files)

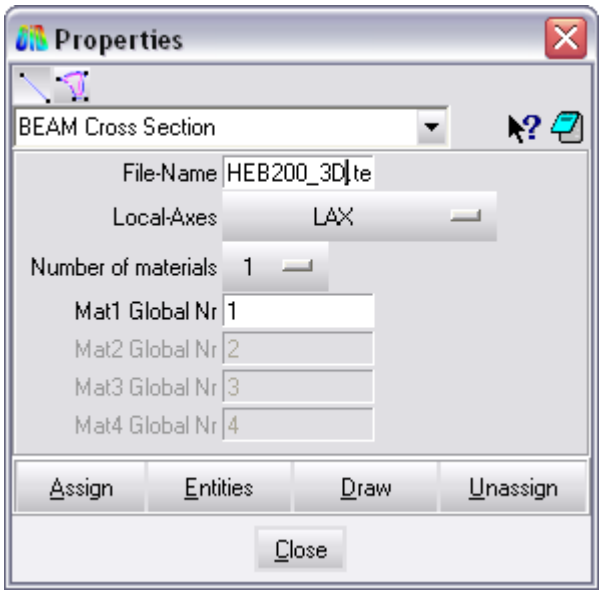
The objective is to assign the .tem file name HEB200_3D to the system line

From the pull down menu select:

- ***Data->Properties***

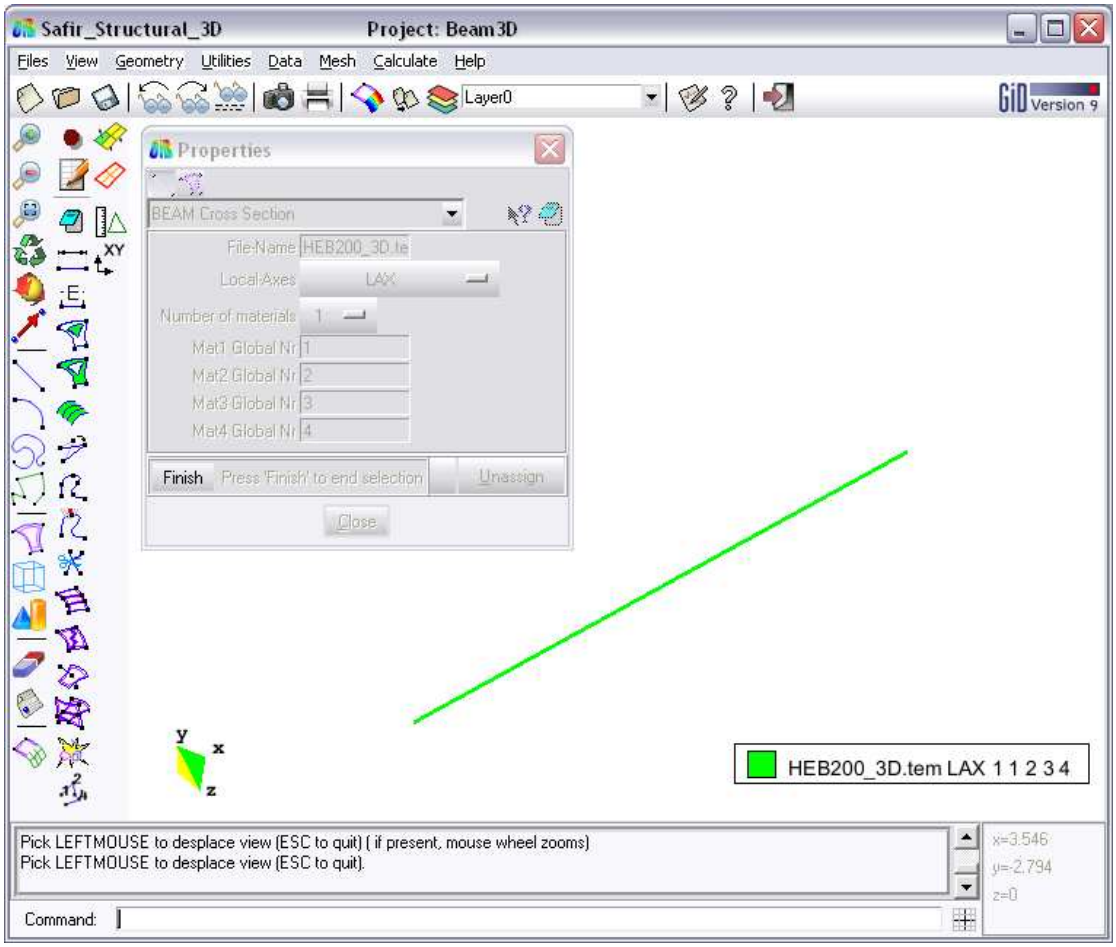
In the dial box change the File-Name: ***Safir.tem*** to the temperature file (.TEM file) of the cross-section, in this case ***HEB200_3D.tem***

Change ***Local-Axes*** from ***-Automatic-*** to ***LAX***



⚠ *In this case all cross-sections have just one material*

Assign the ***HEB200_3D.tem*** section to the beam



To display Property select in the dialog box:

- ***Draw->Colors***

Press ***Finish*** or ***[Esc]*** to leave this view mode

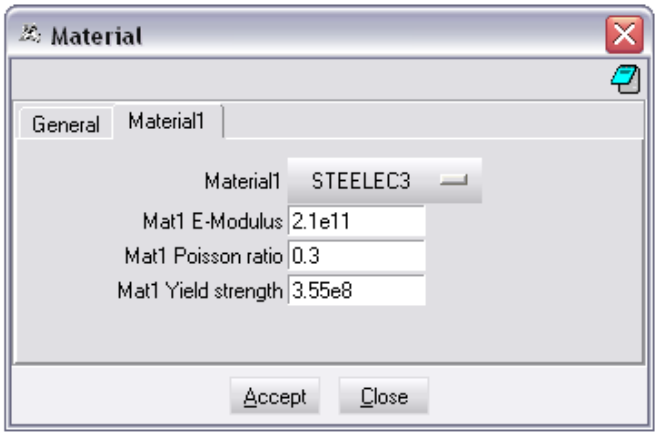
⚠ *Now you have to open the heb200_3D.gid file you already realized (Exercise n°2). Select the .TEM files (heb200_3D.tem) and copy it into your new "Beam3D.gid" directory*

7. Define global materials:

To define material select from the pull down menu:

- *Data->Material*

In the general tab, put one material and in the Material1 tab, fill as shown below



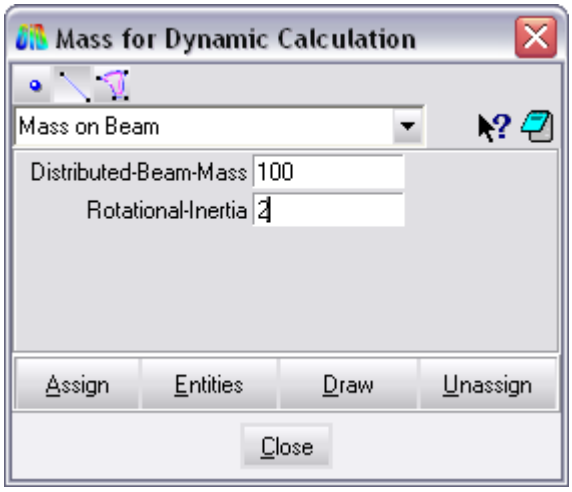
Click on *Accept* to confirm

8. Define the Mass

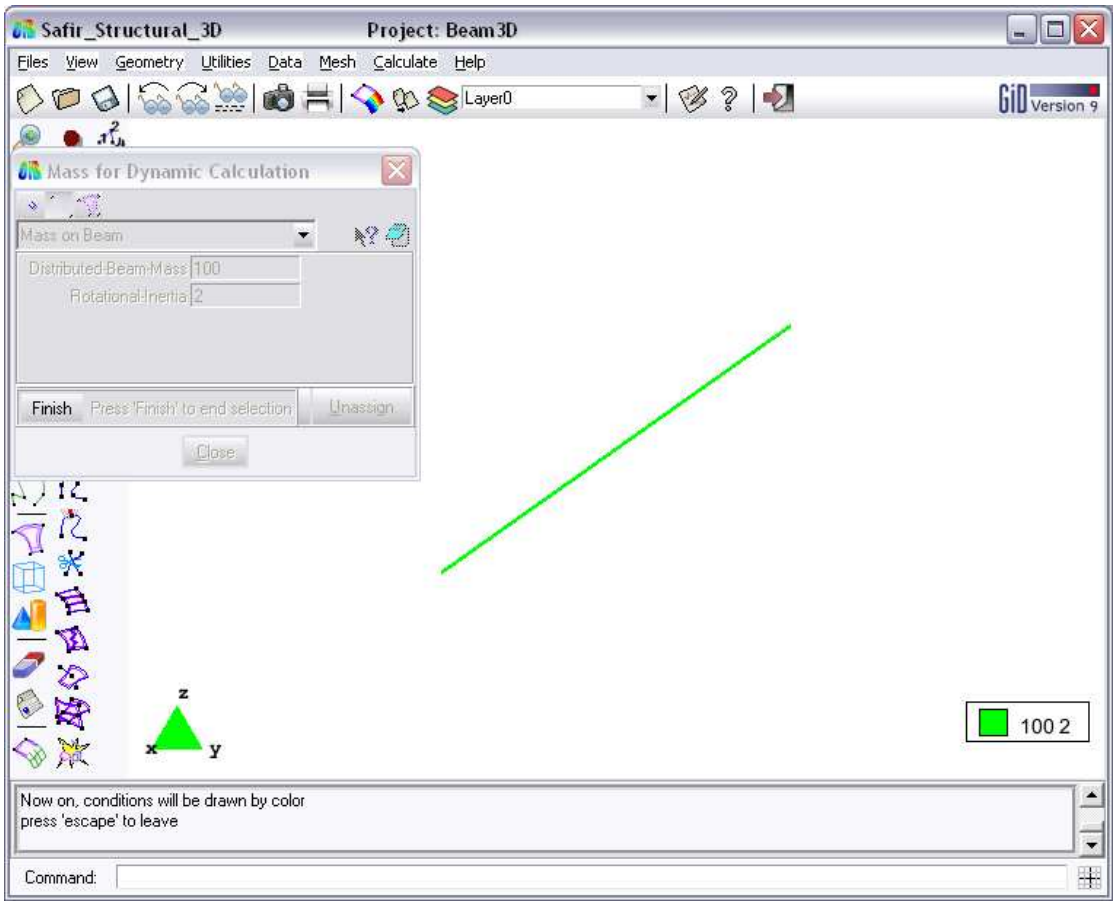
To define to define the mass for dynamic calculation, select from the pull down menu:

- *Data-> Mass*

Select the  button, put 100 kg/m as Distributed-Beam-Mass and 2 as Rotational-Inertia



Assign the mass to the element



To display Property select in the dialog box:

- ***Draw->Colors***

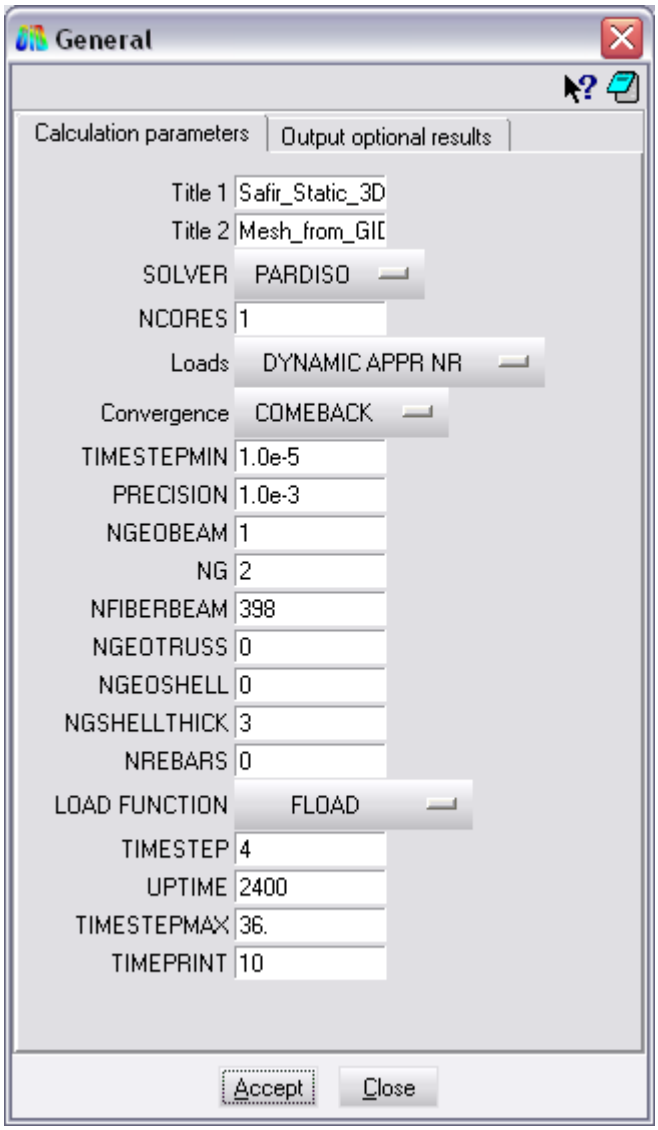
Press ***Finish*** or ***[Esc]*** to leave this view mode

9. Define general data

Select from the pull down menu:

- ***Data->Problem Data***

And fill as shown below



Enter the following

NGEOBEAM = 1 (number of .tem files used)

NFIBERBEAM = 398(max. number of fibers)

TIMSTEP, UPTIME, TIMEPRINT as needed

⚠ *Nfiberbeam is equal to 398 in this case, the only way to know this number is to open the .TEM files you are using with a text editor and read the number of fiber beam on each of them (on the first line). Keep the largest fiber beam number you found and use it as your Nfiberbeam in your problem data.*

⚠ *The Postprocessor Diamond can't open a file bigger than 1.1 Go. It's important to choose your Timestep and other output optional results carefully*

⚠ *You can change TimestepMin, Precision, Timestep, Ttimestepmax and Timeprint as needed but you have to be careful that your UPTIME is less or equal to the UPTIME used for sections 2D calculation*

Click on ***Accept*** to save your modification

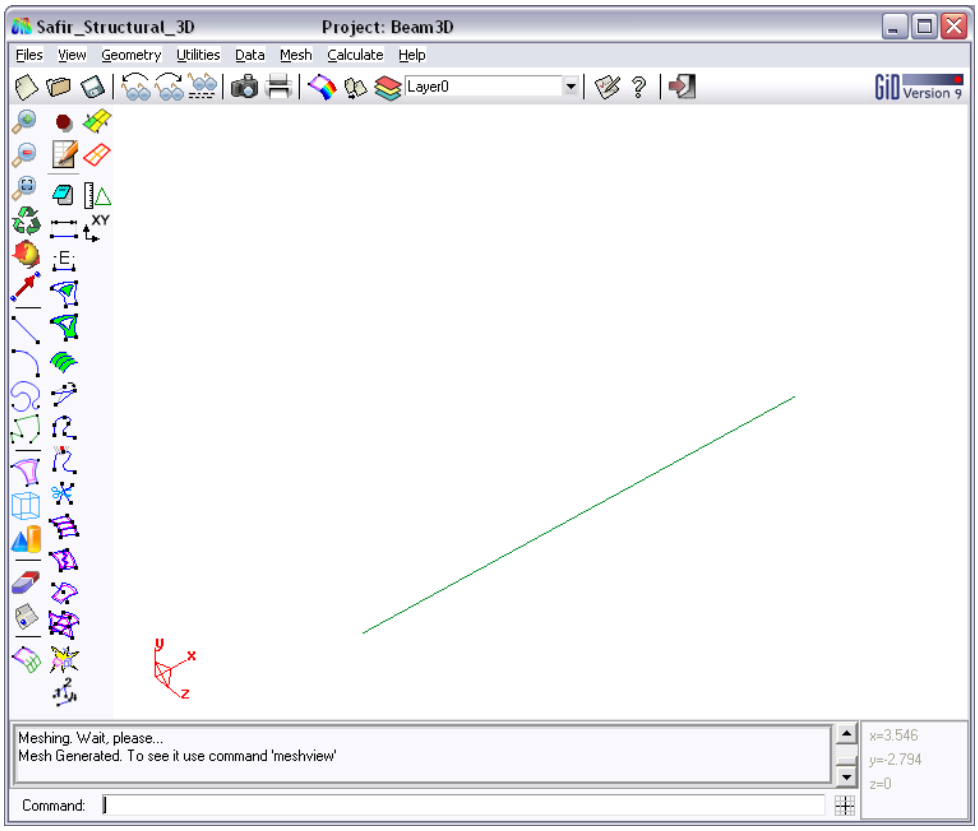
10. Generate the mesh:

Select from the pull down menu:

- *Meshing->Generate* *or [Ctrl + g]*

Enter the element size of 0.2 m

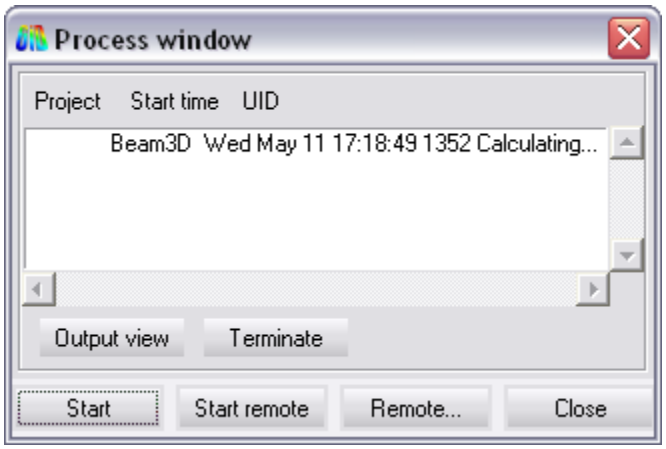
If you are not satisfied with the mesh repeat meshing and change the element size.



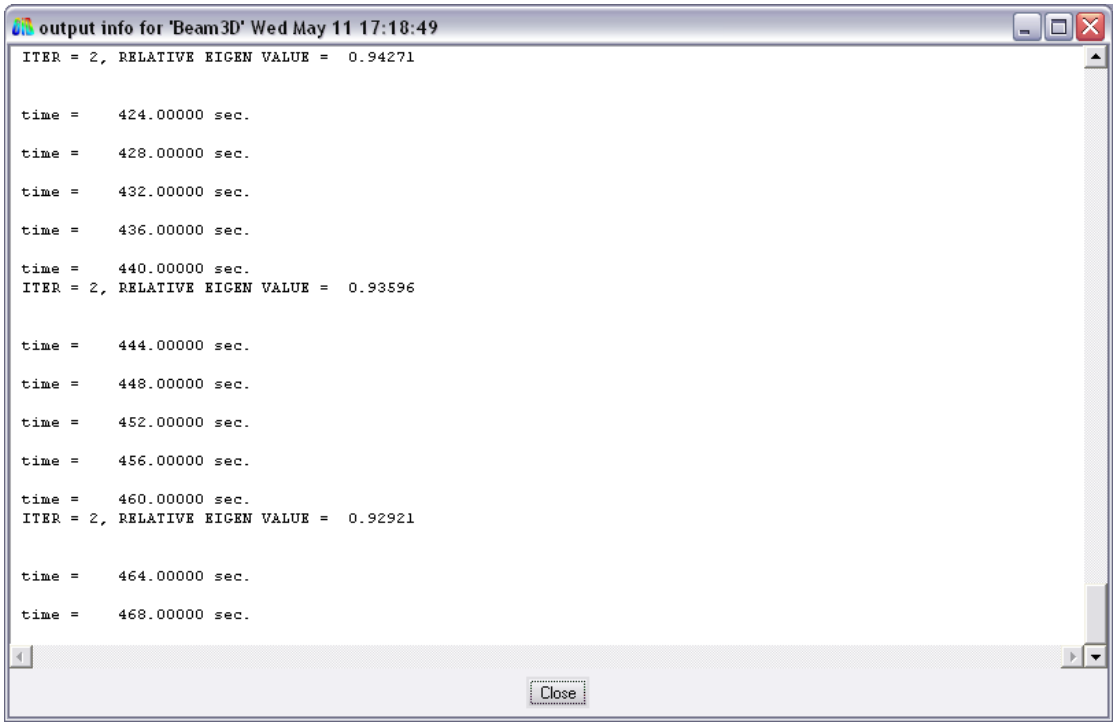
11. Start the calculation:

Select from the pull down menu:

- *Calculate->Calculate window*



Click on the *Start* button then on the Output view button



Click the ***Start*** button and then the ***Output view*** button GiD creates a .IN file in the project directory and starts the calculation. In the output window you can watch the calculation progress from SAFIR and the GiD interface program which generates GiD postprocessor files from the .OUT file.

⚠ *If SAFIR found some errors in the .IN file you will also see the error message in this window. It happens for example when you forgot to copy a .TEM files into the project directory, or if you put a wrong number of NGEOBEM or NFIBERBEAM.*