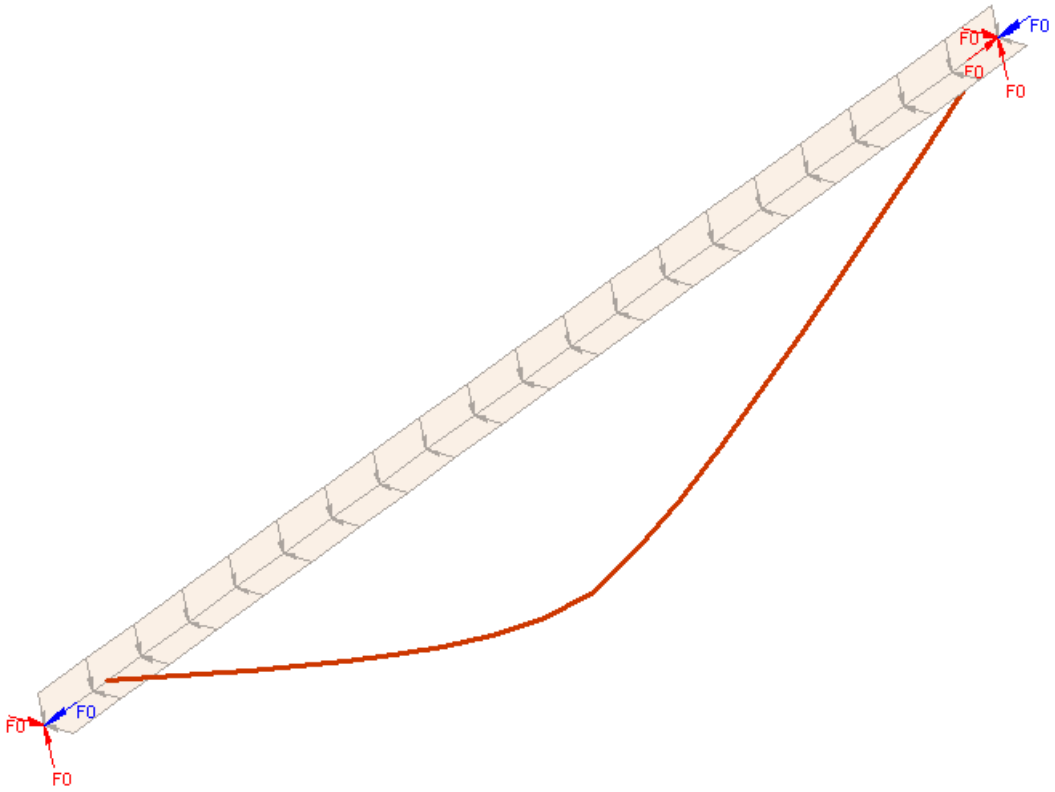
Example of GID-SAFIR

3D Structural Analysis

Exercise n°6 – Beam 3D

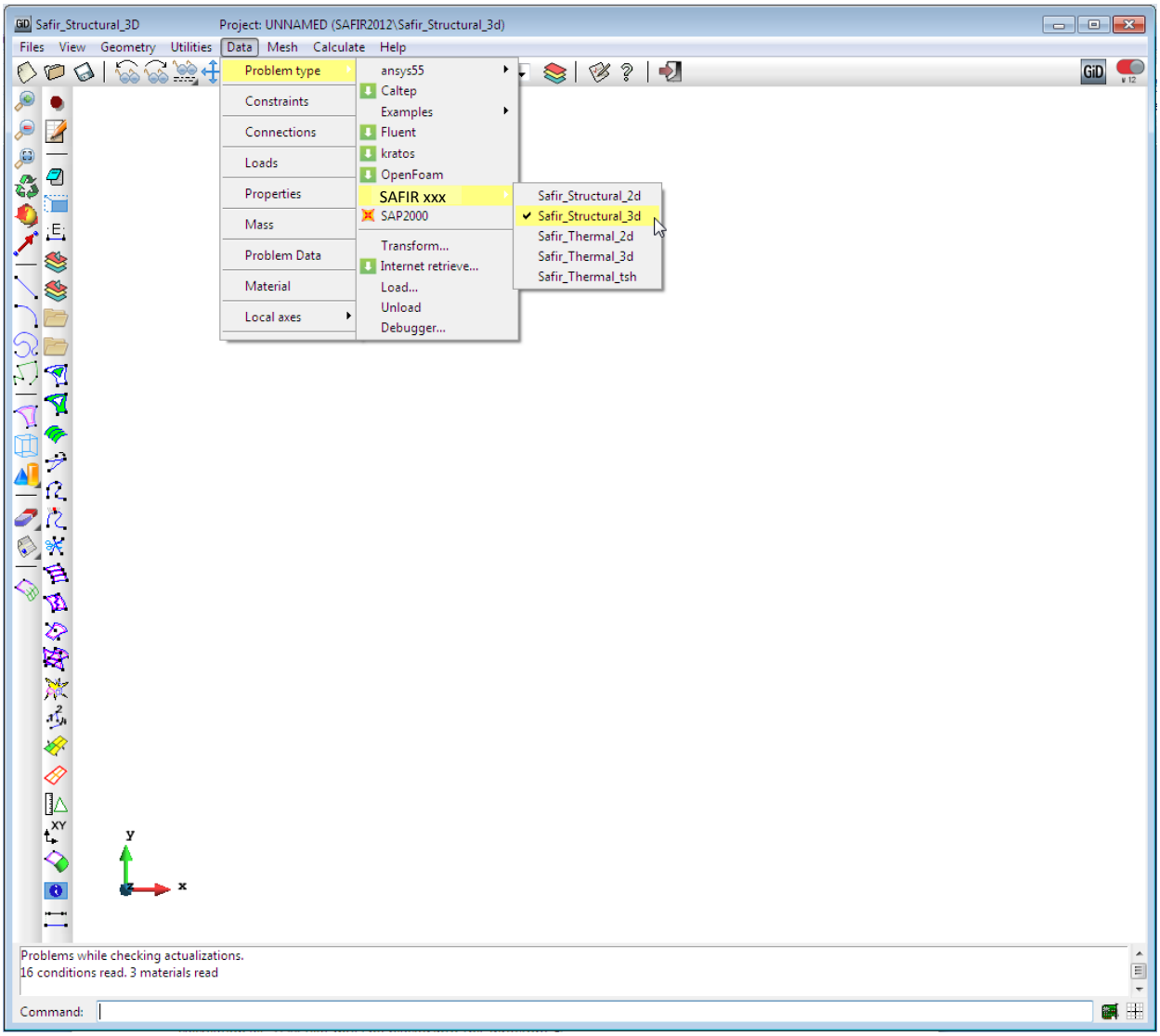


This example explains how to calculate the structural behavior of a 4m length simply supported HEB220 beam in a 3D plane, and subjected to ISO fire. The thermal analysis has been done in the Exercise 2-Part 2.

1. Create a project for a 3D Structural Analysis

From the pull down menu select:

- *Data -> Problem type -> SAFIRxxx -> Safir_Structural_3d*



To save the project select :

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

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

2. Create the geometry of the structure in the xyz-plane

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

- *View->Rotate->isometric*

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

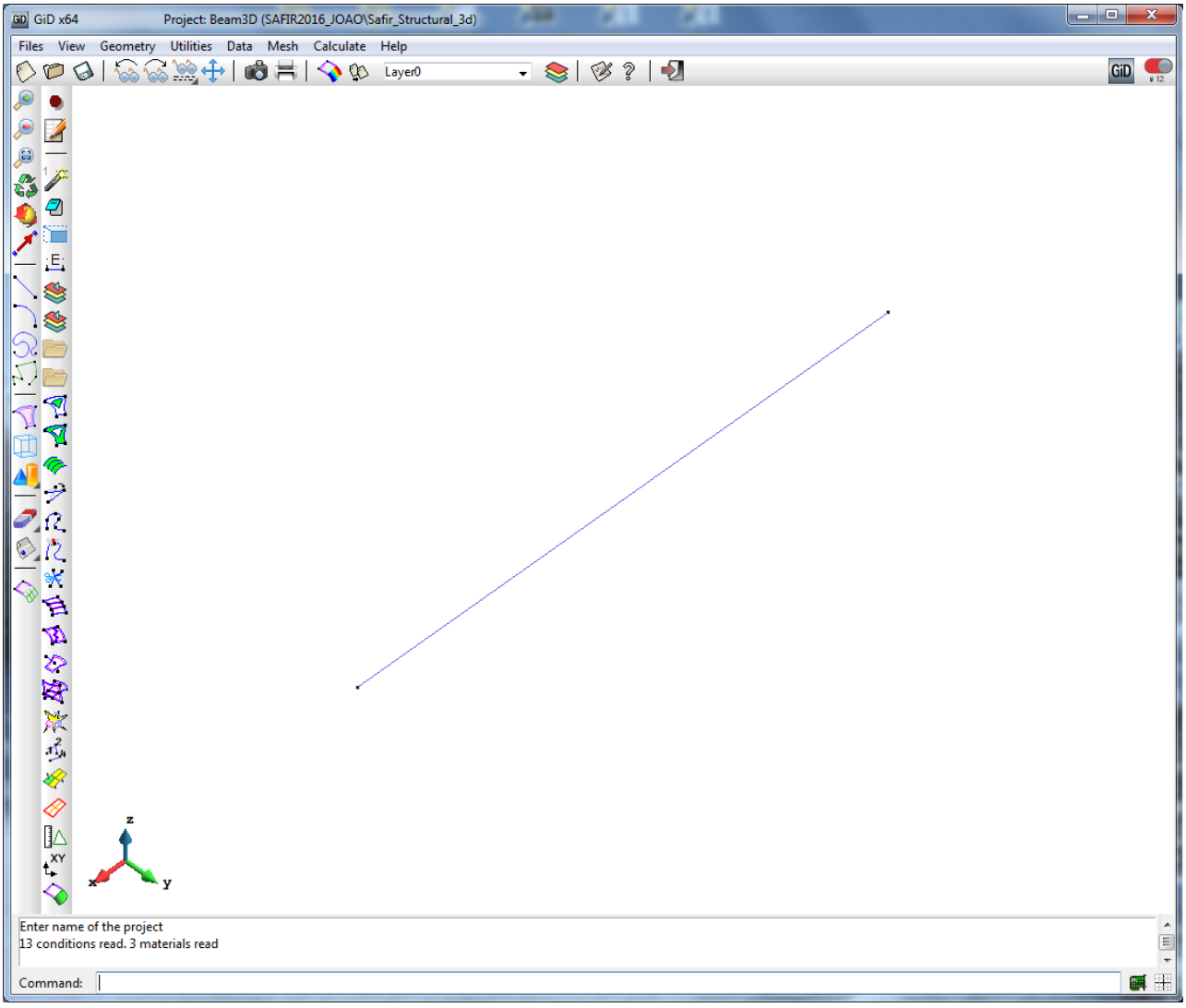
- *View->Rotate->Trackball* *or [F7] or*

Create the system lines:

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

In the command line (at the bottom of the window) enter the coordinates in [m] of the line points (with a whitespace between each coordinates):

0,0,0 4,0,0 and press *[Enter],* then twice *Esc* to quite this line mode.



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

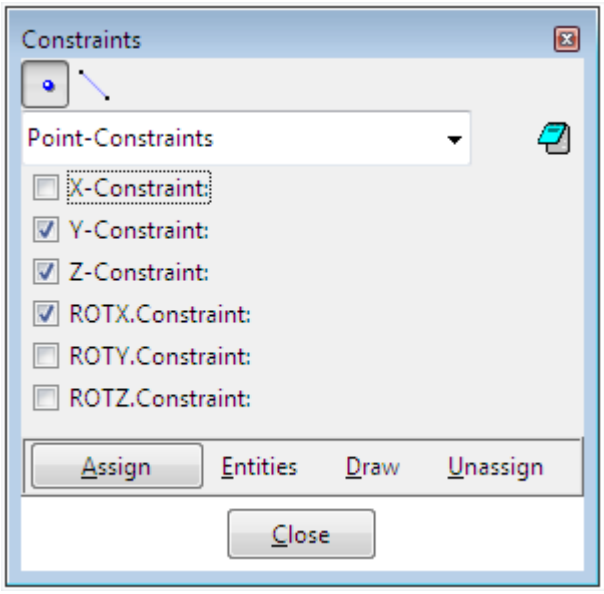
- *View->Zoom->Frame* *or [F11] or*

3. Define constraints for the supports:

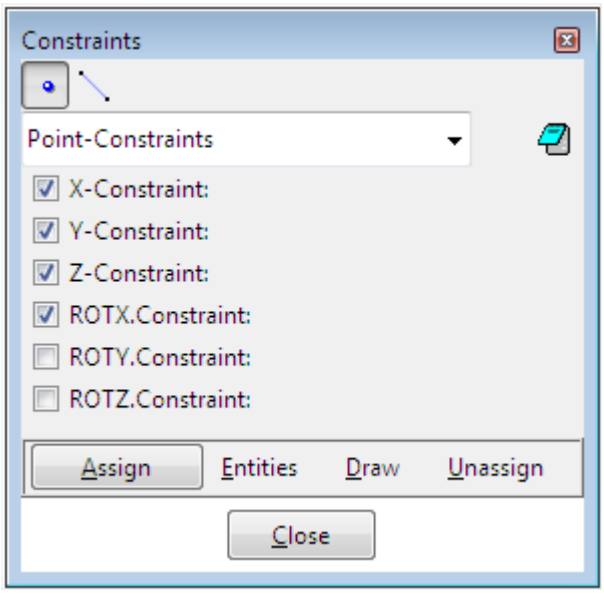
From the pull down menu select

- ***Data->Constraints***

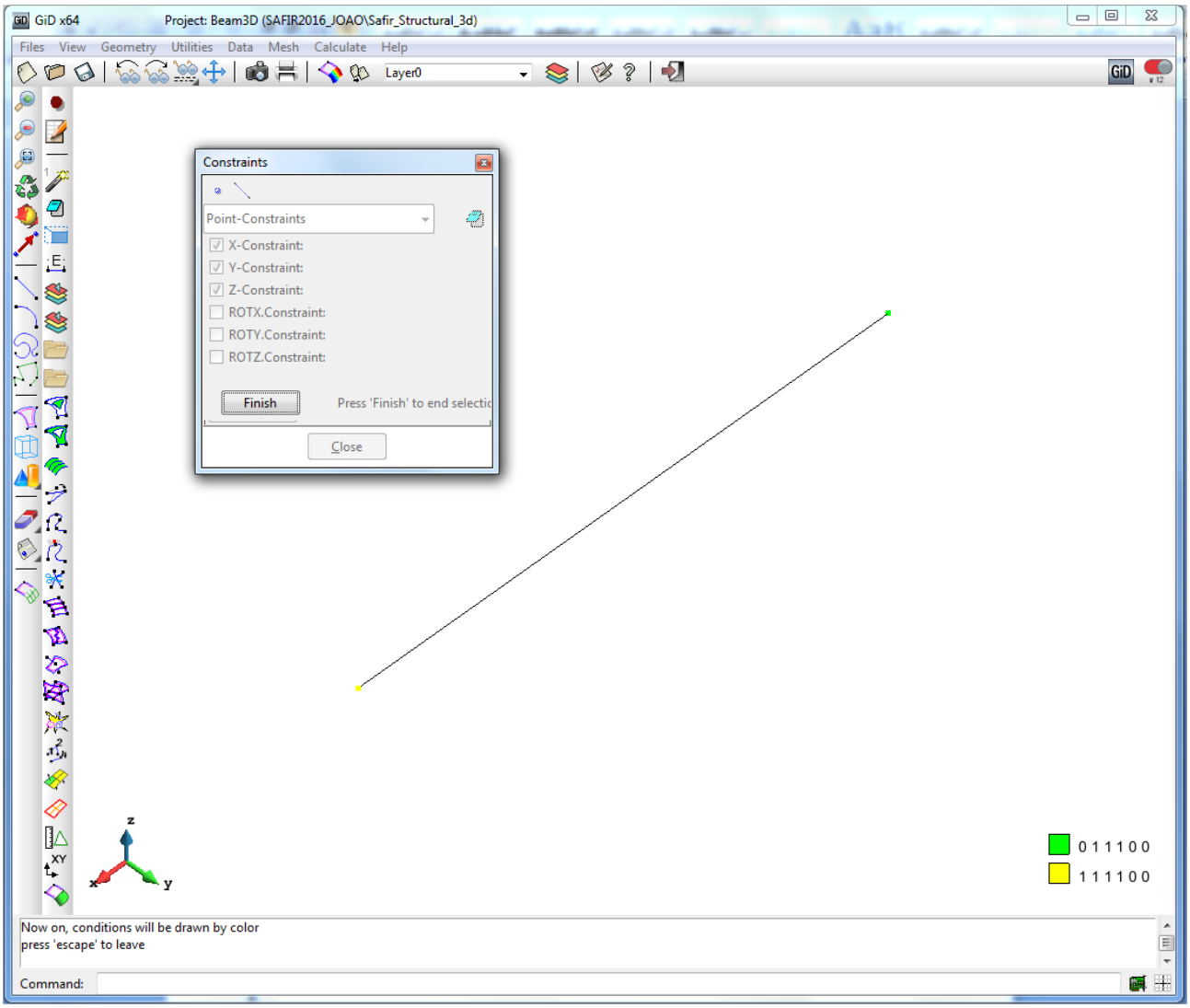
Select Y and Z-Constraint, X rotation constraint, and assign it to the left end point of the beam.



Select X,Y and Z-Constraint, X rotation constraint, and assign it to the right end point of the beam.



In the dialog box, with ***Draw->Colors*** you can display the constraint conditions.

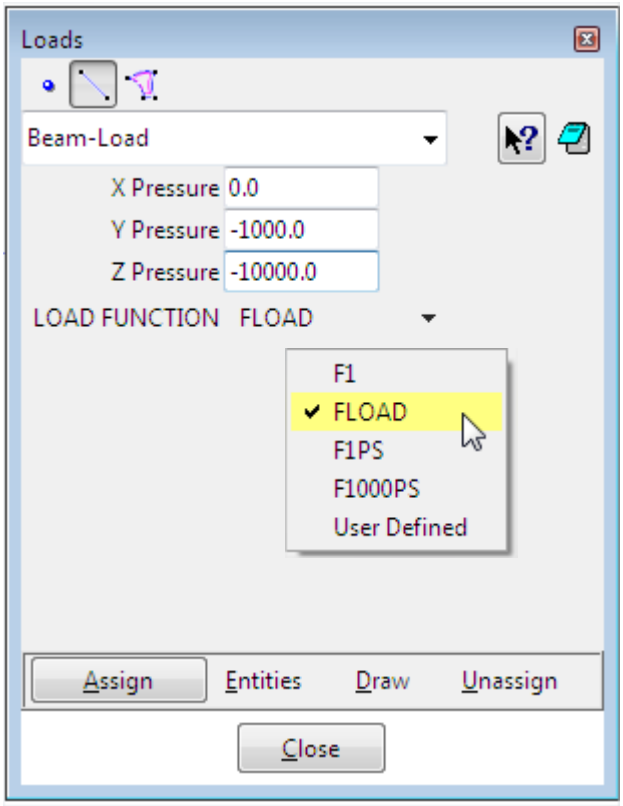


Press ***Finish*** then ***Close*** to leave this view mode.

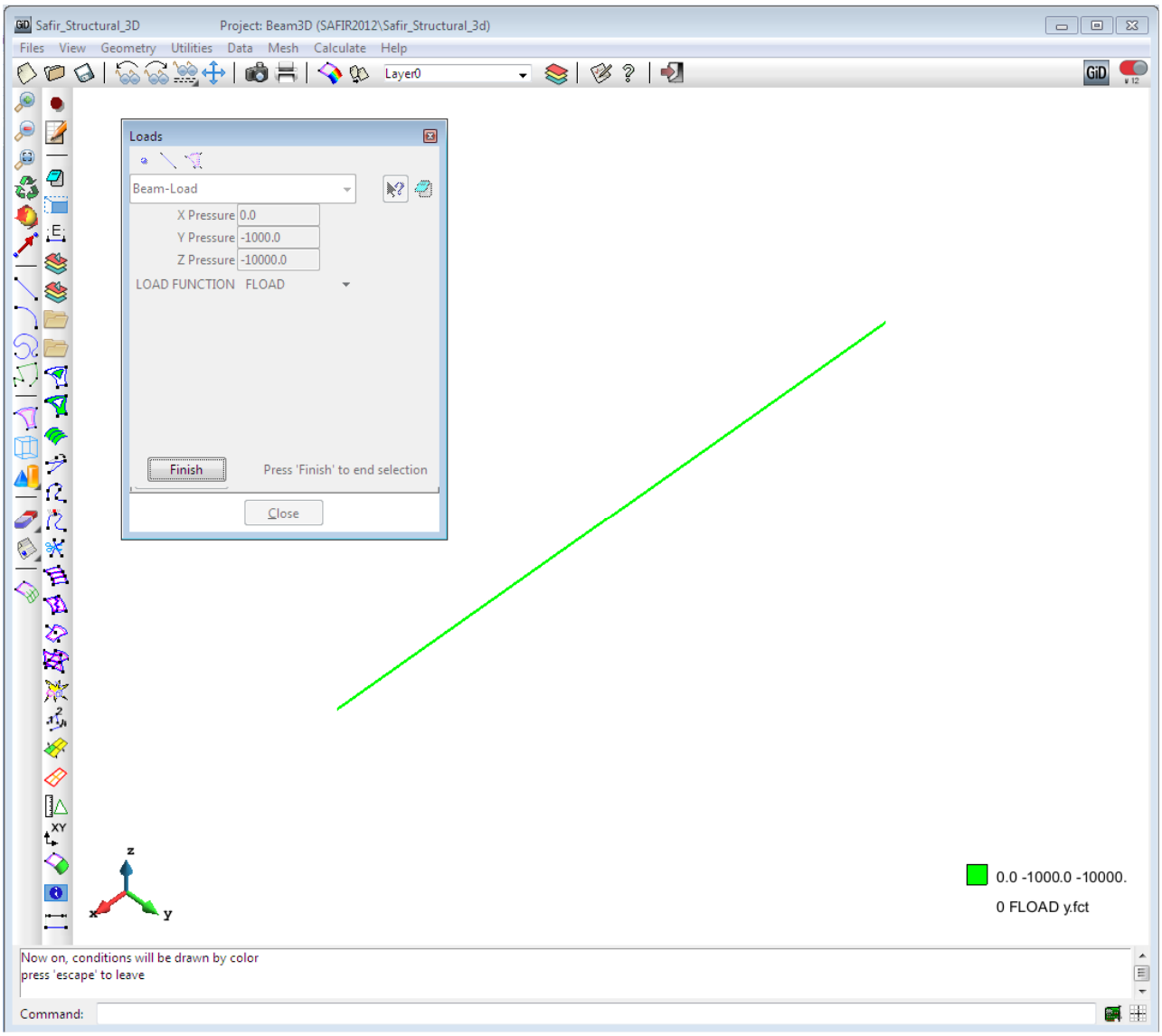
4. Define loads

From the pull down menu select:

- ***Data->Loads***



In the dialog box select ***Beam-Load***  , enter a Y-Pressure of ***-1000N/m*** , a Z-Pressure of ***–10000N/m***, select FLOAD in the LOAD FUNCTION dialog box, then ***Assign*** it to the beam.



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

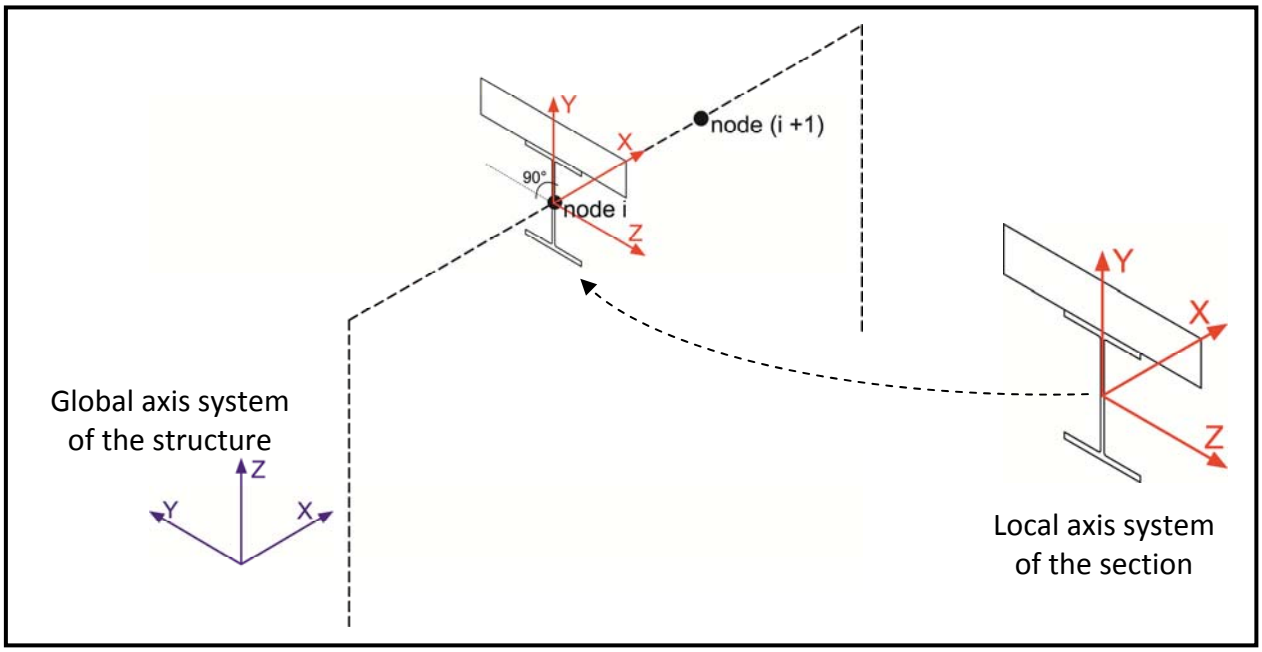
Press ***Finish*** then ***Close*** to leave this view mode.

5. Create local axes

The orientation of the cross-section is controlled by defining his local axis in the global XYZ plan of the structure. The objective is to create a local axis for each different section orientations.

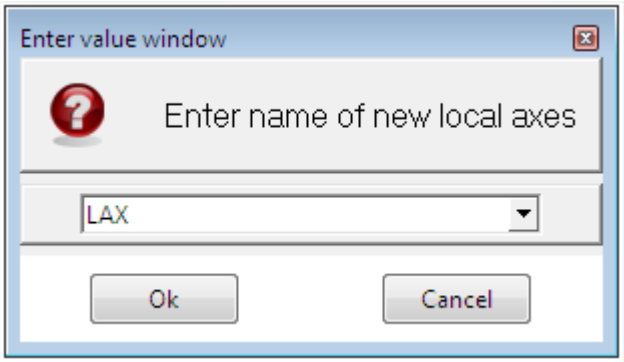
The most used procedure to create a local axis is

- to fix the X direction by given 2 existing nodes of the structure (node i and node i+1)
- to fix the Y direction by given an angle (90°)



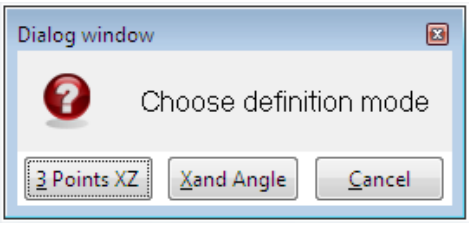
To define the local axis of the section, from the pull down menu select:

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



Enter the local axis name ***LAX*** and click ***OK.***

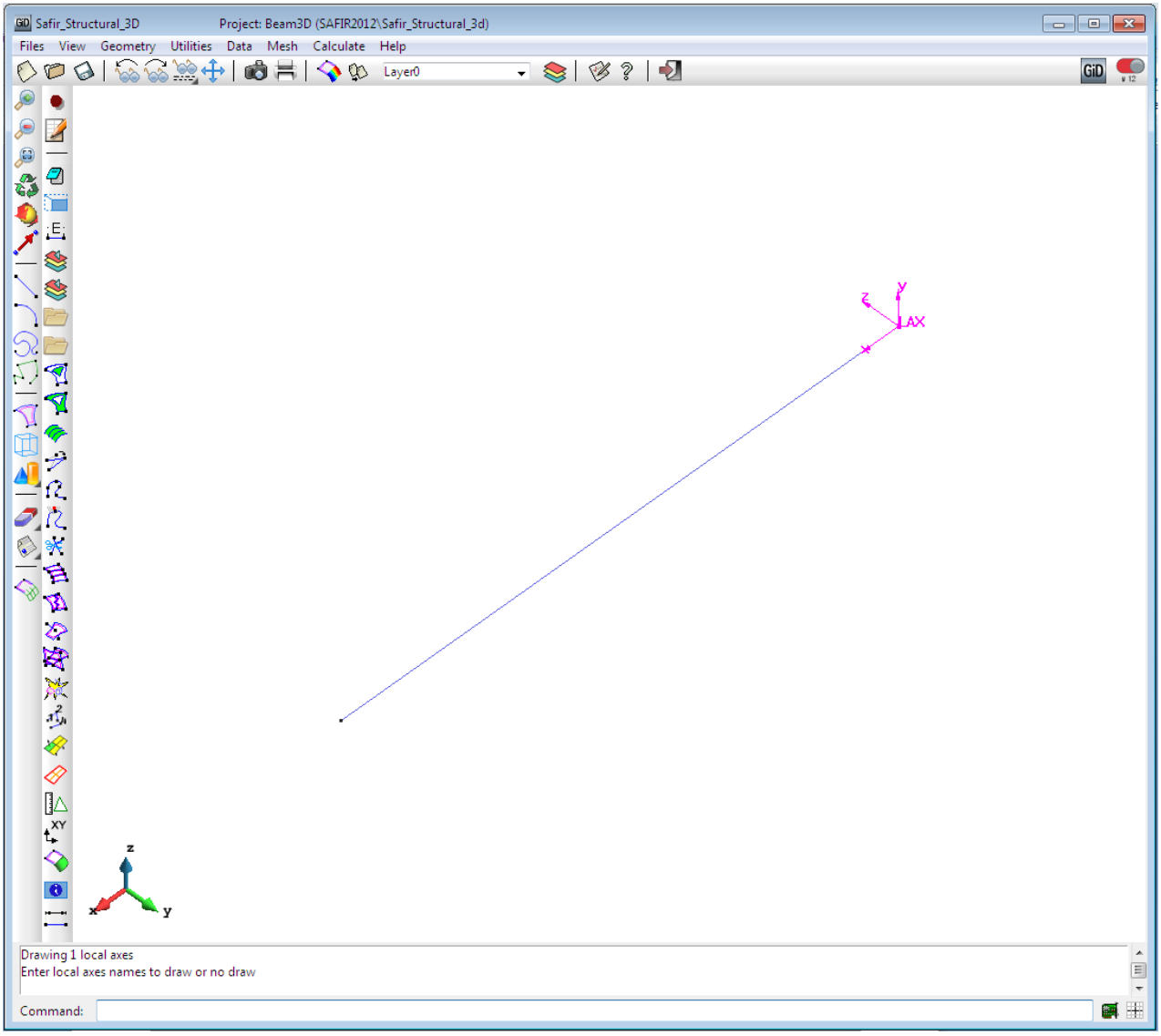
Then choose the definition mode ***Xand Angle***.



Press *[Ctrl+a]* to select the origin of the local axis LAX (left point of the beam). Then press *[Ctrl+a]* to select a point in the positive orientation of the local axis LAX (right point of the beam). Now enter an angle (in this example put 90°) to get the orientation of the Y axis of the local axis LAX, and therefore the orientation of the section.

To draw local axes select:

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



6. Assign the section (temperature file .TEM)

The objective is to assign the section HEB220_3D (defined in the Exercise 2 – Part 2) to the system line.

From the pull down menu select:

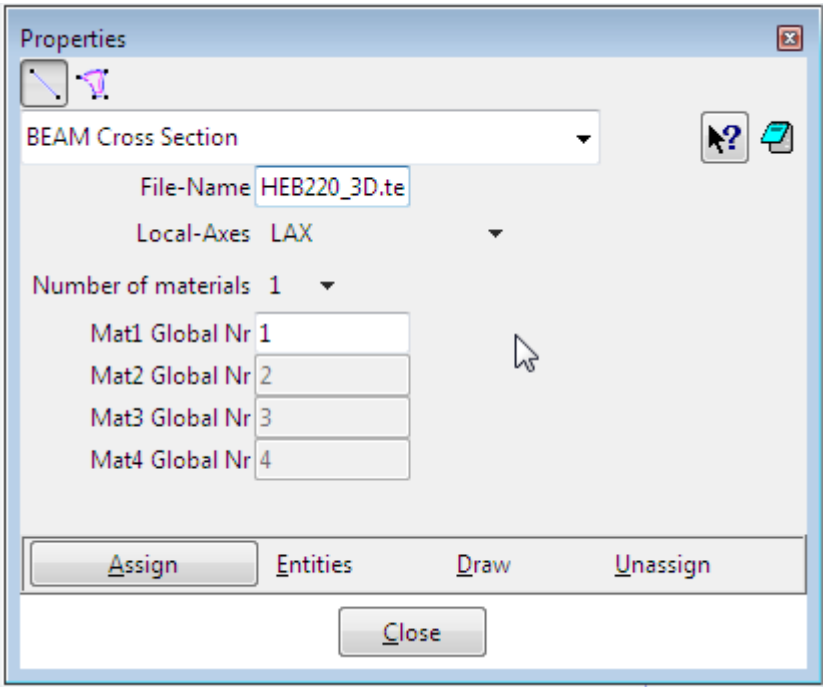
- ***Data-> Properties***

In the File-Name dialog box, give your section file name ***HEB220_3D.tem.***

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

As the section is only steel material, select ***1*** for the Number of materials.

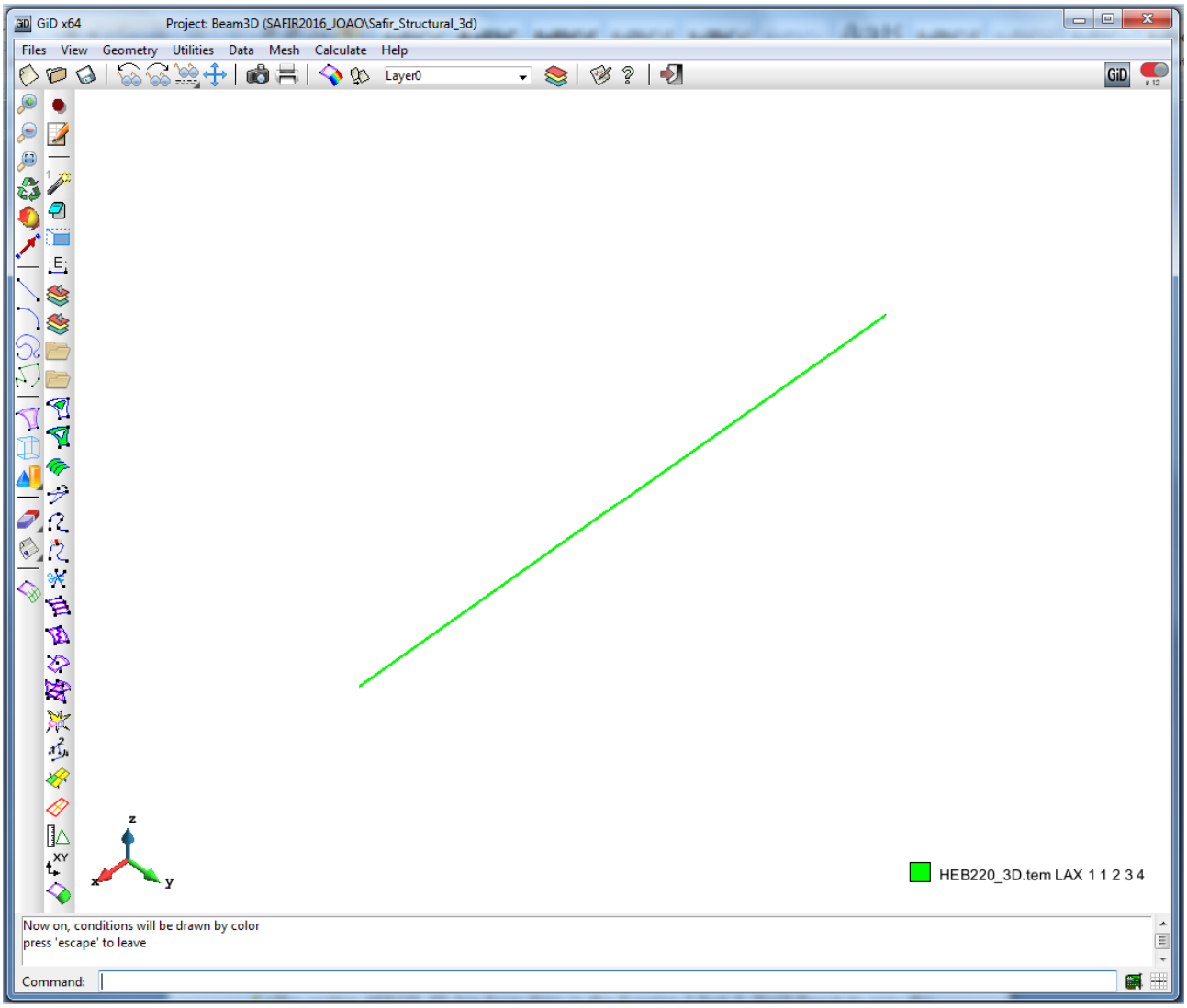
Then put ***1*** for ***Mat1 Global Nr***. This means that the material number 1 defined in the section HEB220_D will have the mechanical properties number 1 in the "structure.in" file.



Number of materials	*The number of material should be the same as defined for the thermal analysis of the considered section (.tem file).*
*Mat**x** Global Nr **y***	*This is the rank **y** in the "structure.in" file (Global Nr **y**) of the Material **x** (Mat**x**) defined in the .tem file.*

Click on ***Assign*** and select the line.

Click on ***Finish*** , then ***Close.***



In the dial box, click on ***Draw-> Colors*** to display the properties and press ***[Esc]*** or click on ***Finish*** to quite this view mode.

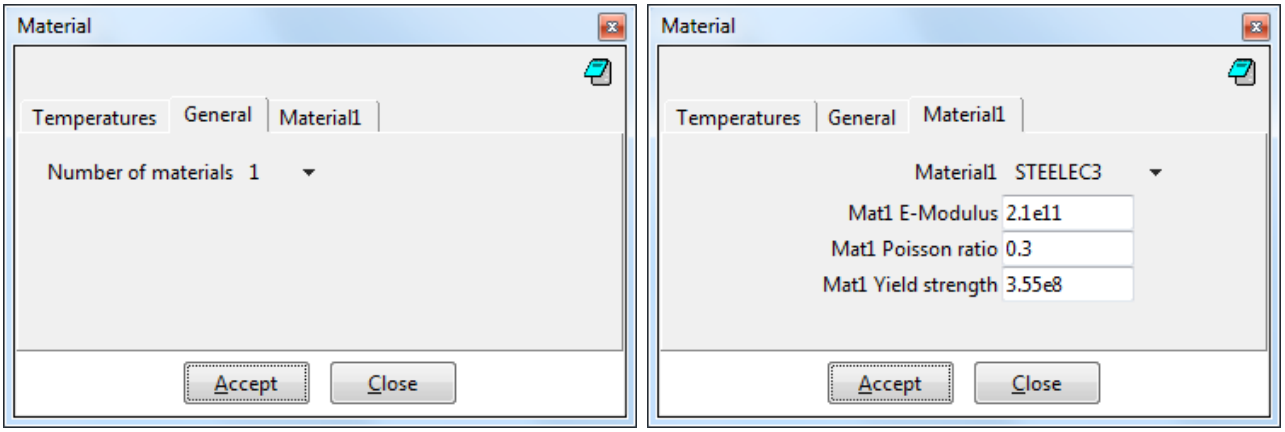
⚠ *The section HEB220_3D has been done in the Exercise 2-Part 2. Don't forget to copy the HEB220_3D.tem file in the Beam3D.gid directory before launch the structural analysis.*

7. Define global materials

To define material, select from the pull down menu:

- ***Data->Material***

In the *General* tab, put ***1*** material and in the *Material1* tab, fill as shown below :



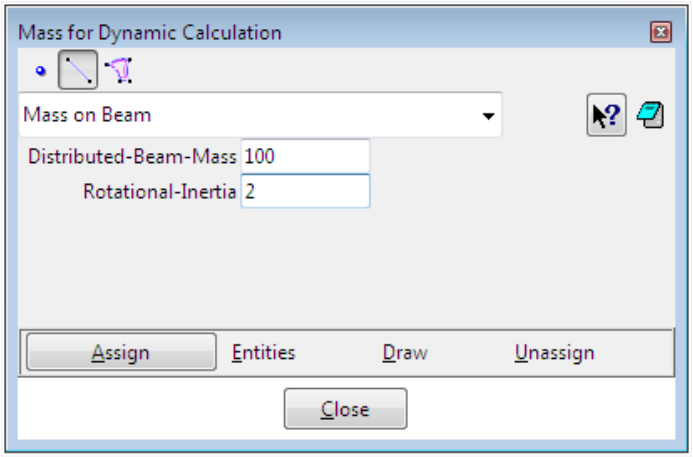
Click on ***Accept*** to confirm.

8. Define the Mass

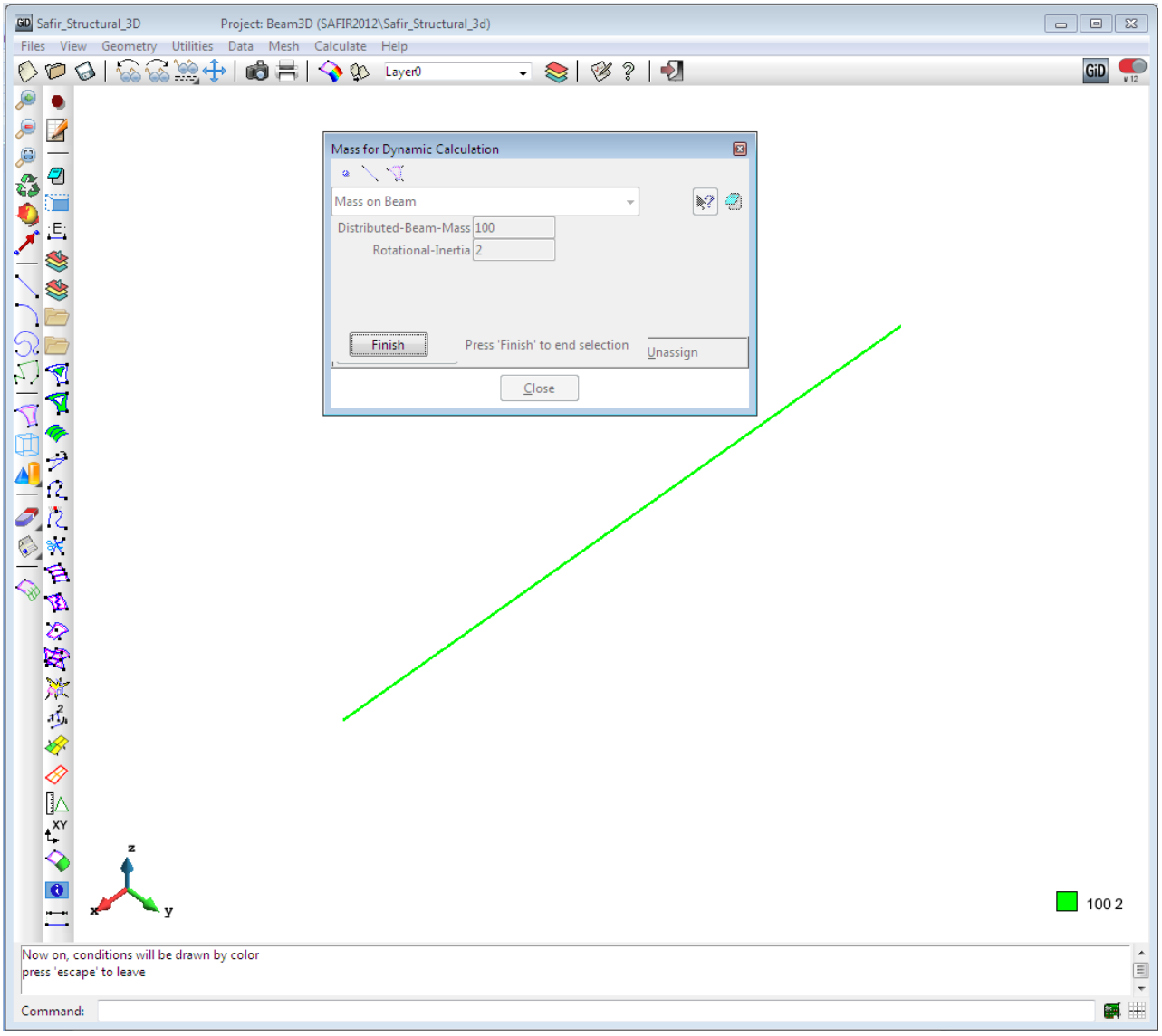
For the dynamic calculation, you need to give a mass to the beam elements. Select from the pull down menu:

- ***Data-> Mass***

To put Mass on Beam, select the  tab and put ***100kg/m*** as Distributed-Beam-Mass and ***2*** as Rotational-Inertia.



Assign the mass to the beam element, and click on ***Finish*** then ***Close.***



To display Property select in the dialog box:

- ***Draw->Colors***

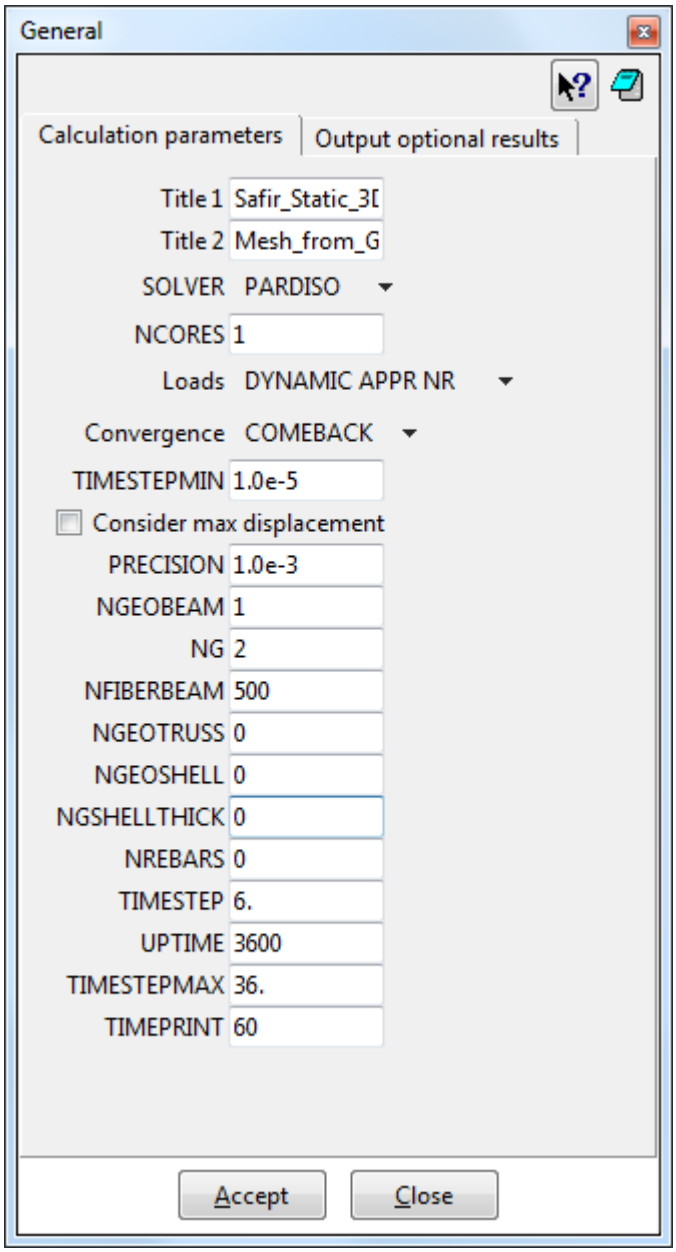
Press on ***Finish*** then ***Close*** to leave this view mode.

9. Define general data

Select from the pull down menu:

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

Fill the dialog box as below :



⚠ *NGEOBEAM is the number of different sections which constitute the structure (.tem files).*

NFIBERBEAM is the number of fibers (meshes) in the section. This value is given in the first line of the .TEM file (in case of several .TEM files, give the maximum value).

Change TIMESTEPMIN, PRECISION, TIMESTEP, UPTIME, TIMESTEPMAX and TIMEPRINT as needed.

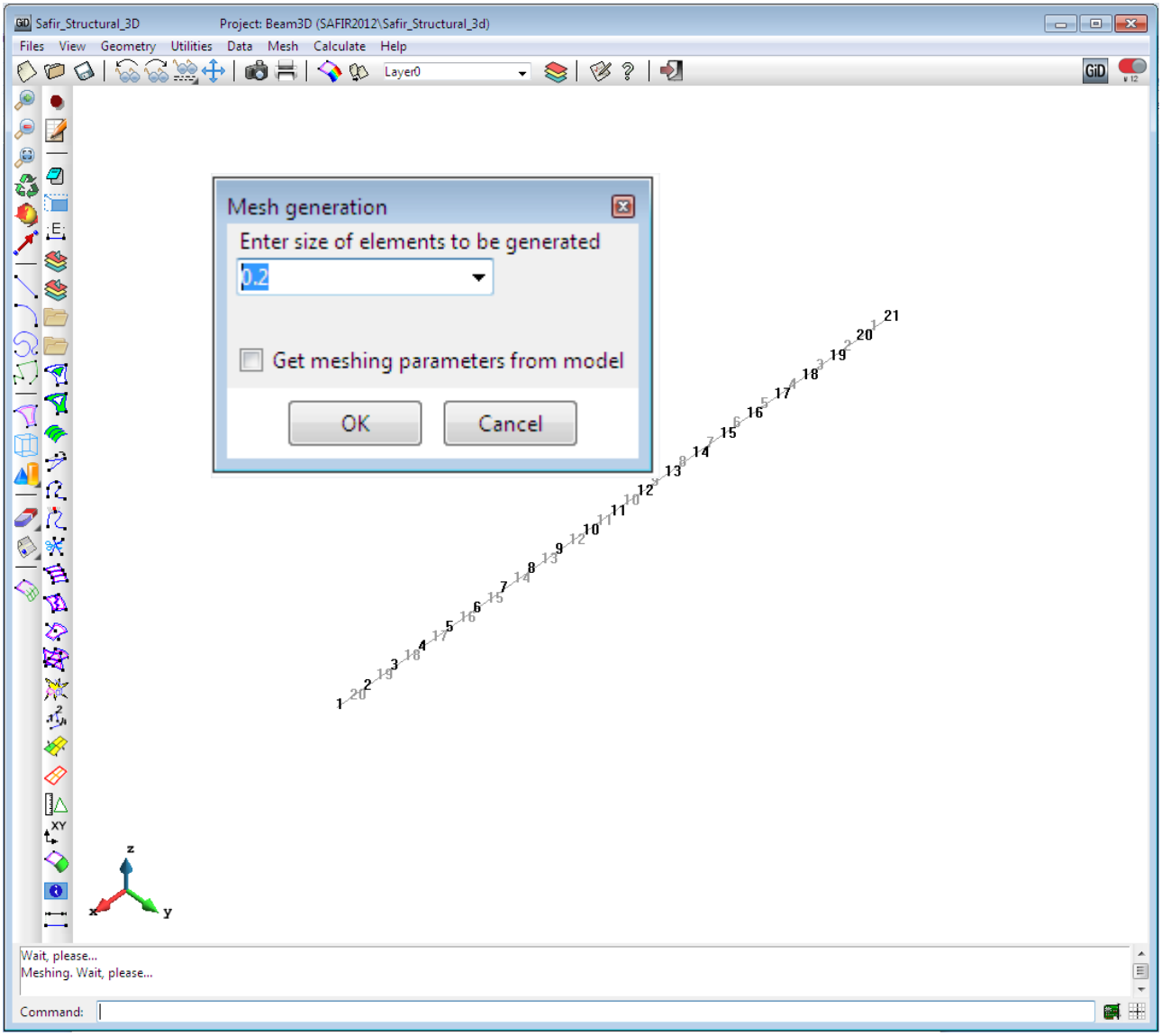
Click on ***Accept*** to save your change, then ***Close***.

10. Generate Mesh

To create meshes select from the pull down menu:

- ***Mesh->Generate mesh*** ***or use [Ctrl + g]***

Enter ***0.2***m as size of elements to be generated.

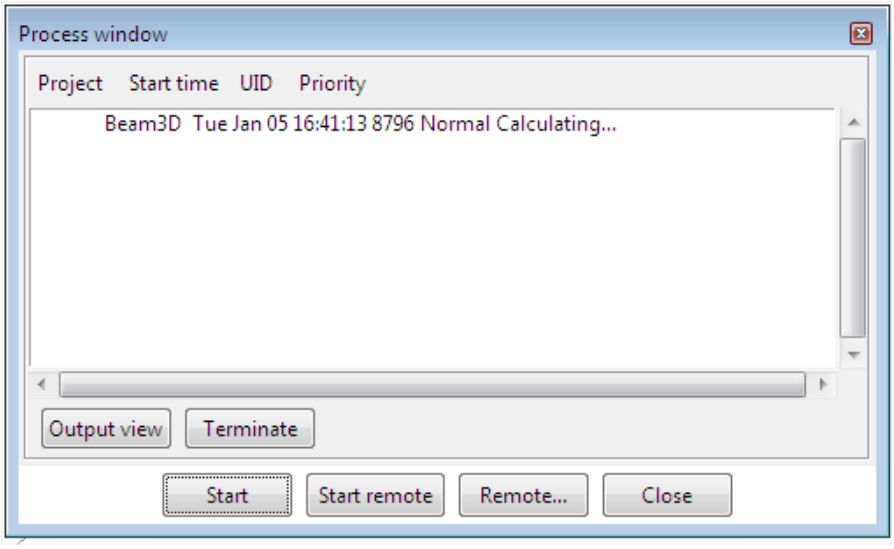


11. Start the calculation

Before starting the calculation, don't forget to copy the HEB220_3D.tem file in the directory "Beam3D.gid".

To start the calculation, select from the pull down menu:

- ***Calculate->Calculate window***



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