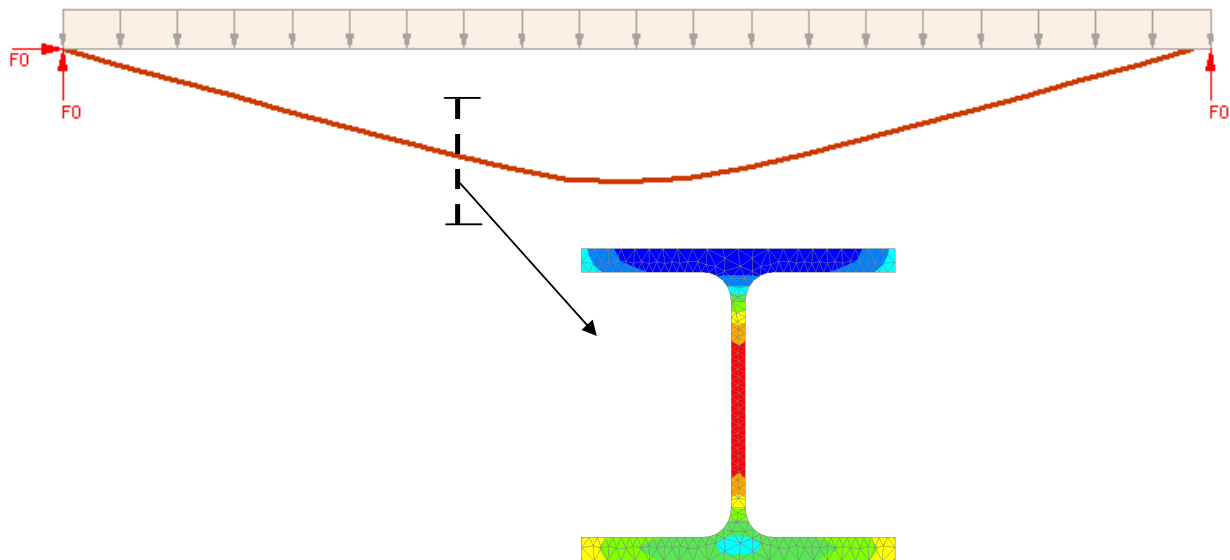


# Example for GiD-SAFIR 2D Structural Analysis

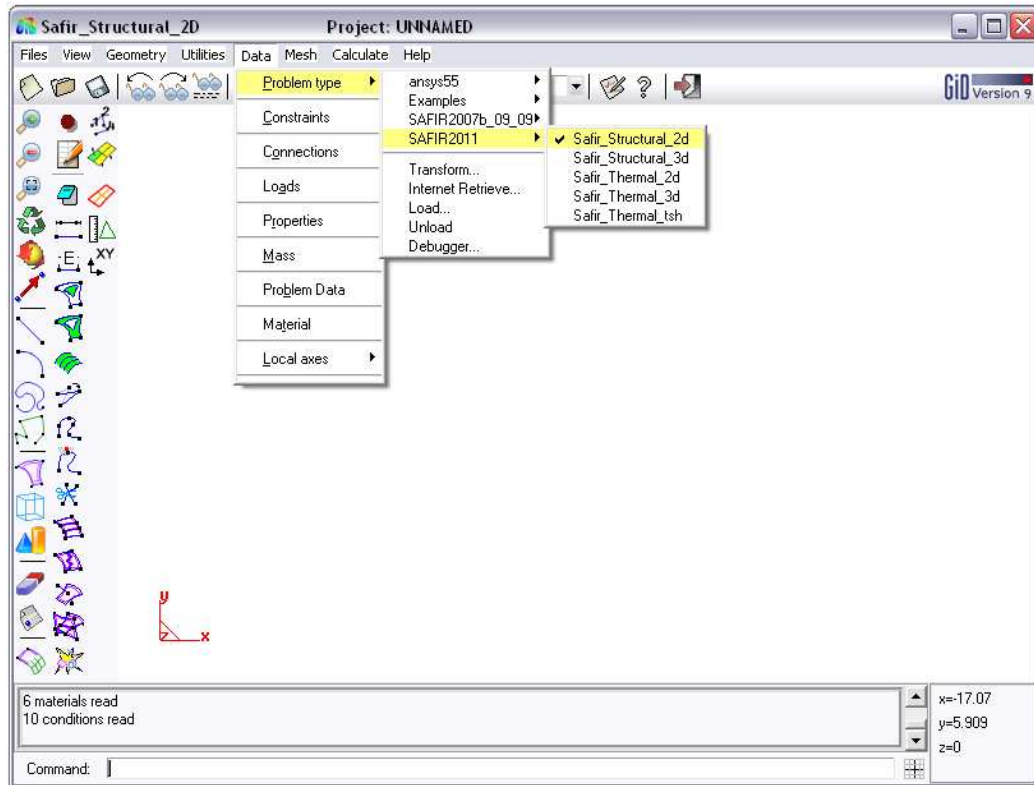
## Exercise n°5 - Beam 2D



## 1. Create a project in 2D for Structural Analysis

From the pull down menu select:

➤ *Data->Problem type->SAFIR2011->Safir\_Structural\_2d*



To save the project select (or use icons 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.: beam2D

GiD creates a directory with the name beam2D.gid

GiD creates a number of system files in this directory.

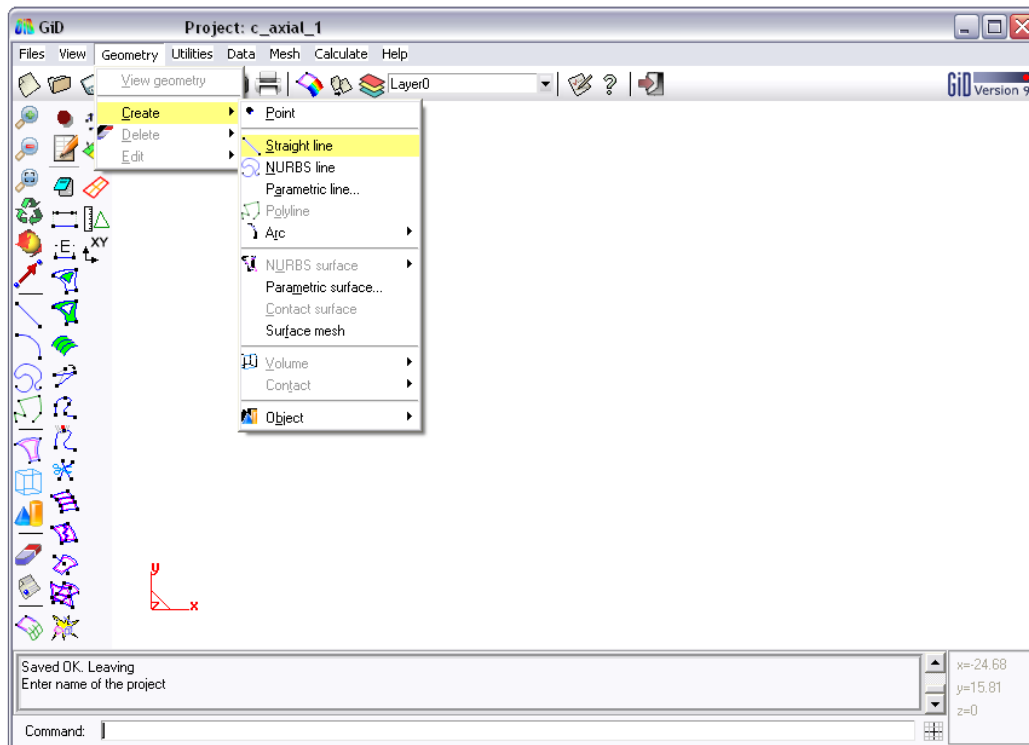
When you start the SAFIR calculation the Safir . IN, .OUT and .TEM files will be created in this directory.

## 2. Create the geometry in the xy-plane

From the pull down menu select:

► **Geometry->Create->Straight Line**

or

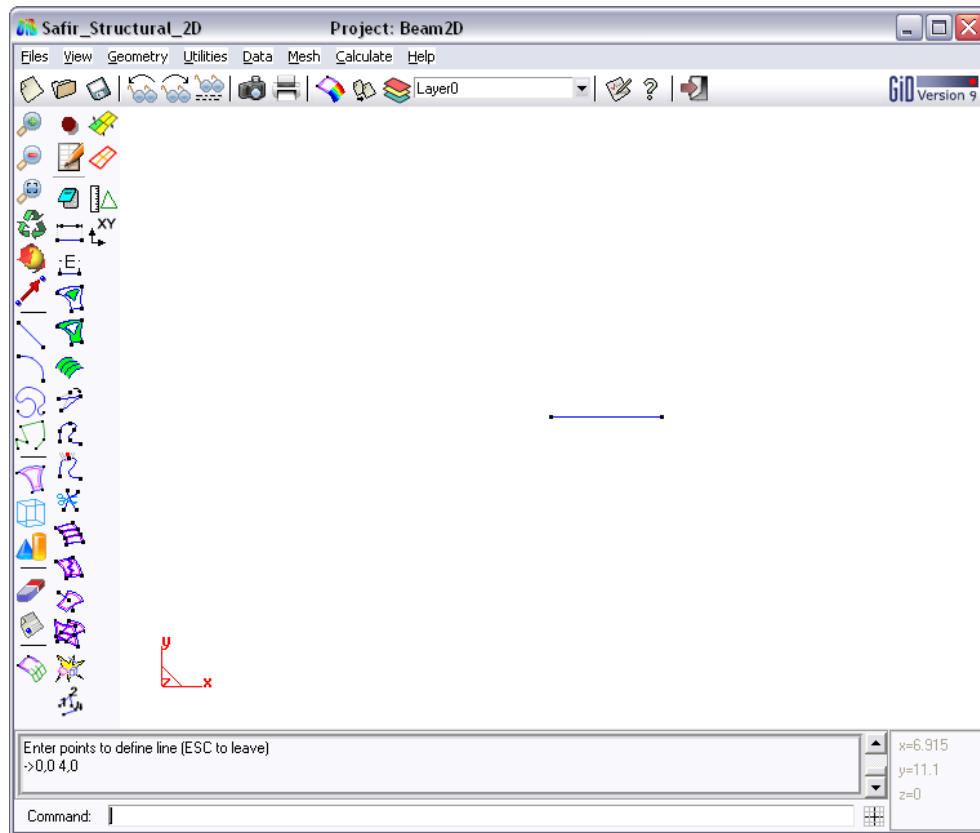


In the command line (in the bottom of the widows) enter the coordinates of the line points:

**0,0 4,0**

Press **[Enter]** to confirm

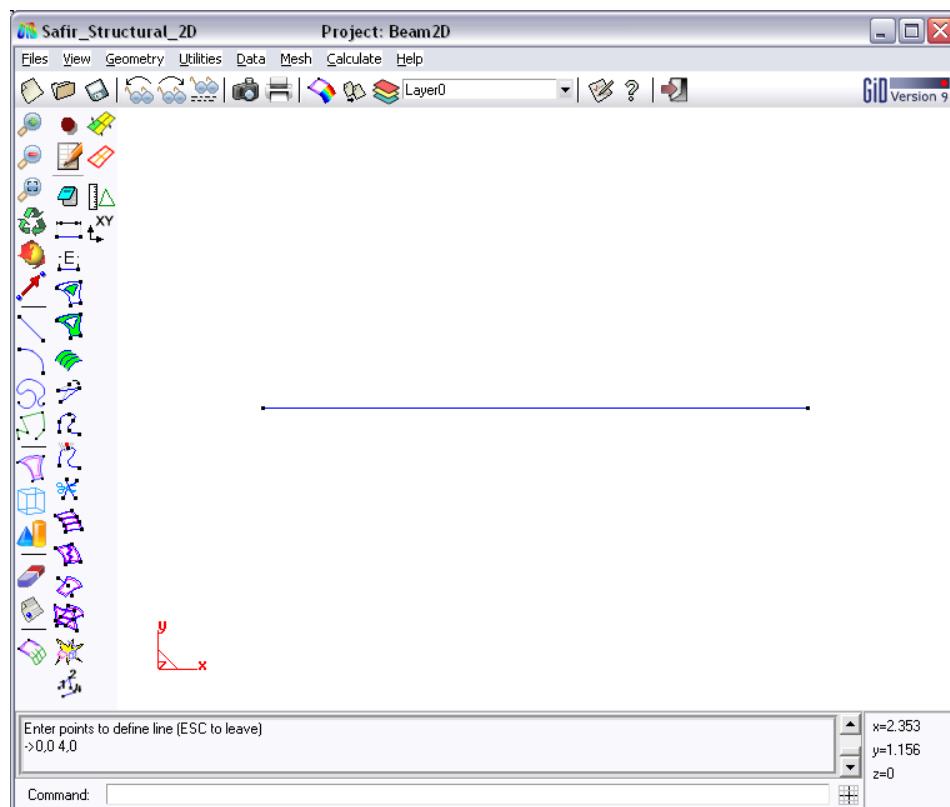
Press **[Esc]** twice to leave line mode



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

► **View->Zoom->Frame**

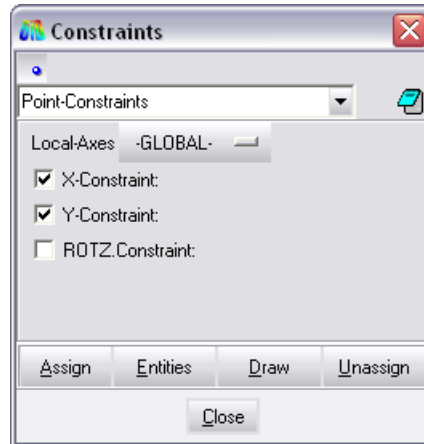
or [F11] or



### 3. Define constraints for the supports

From the pull down menu select:

➤ **Data-> Constraints**

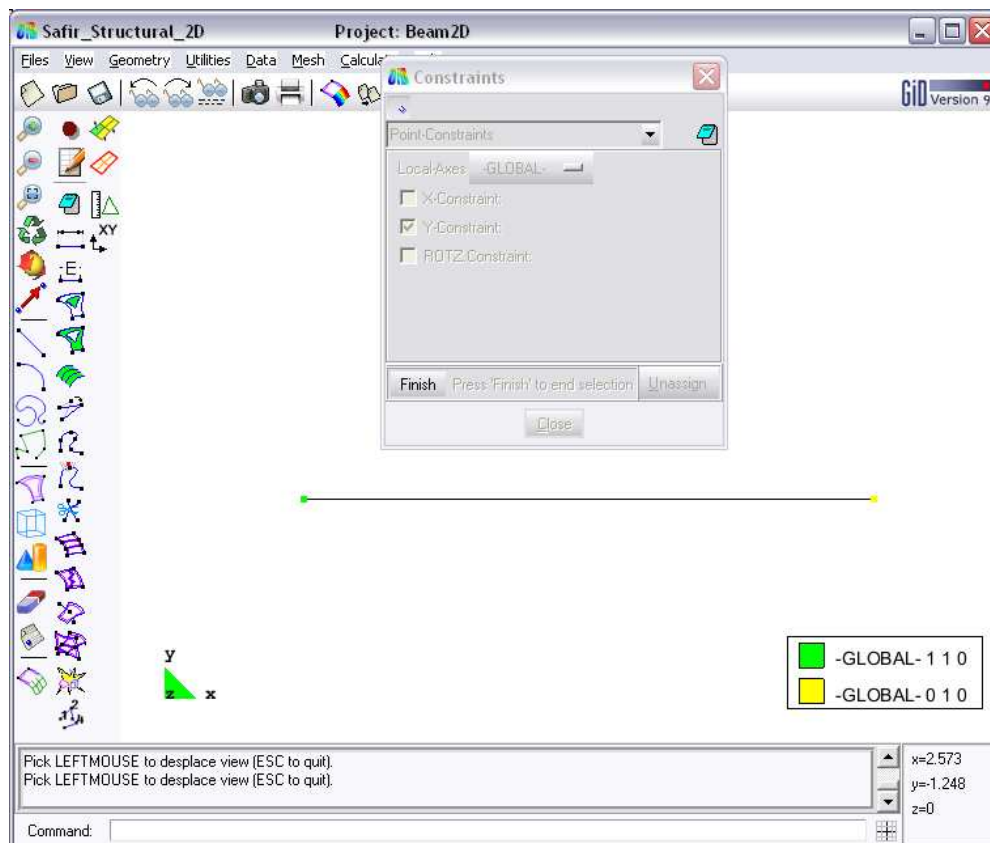


Select x and y constraint and click on **Assign**

Select the left points of the beam

Press **[Esc]** or click on **Finish** to confirm

Select y-constraint and assign it to the right points of the beam




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

#### 4. Define loads

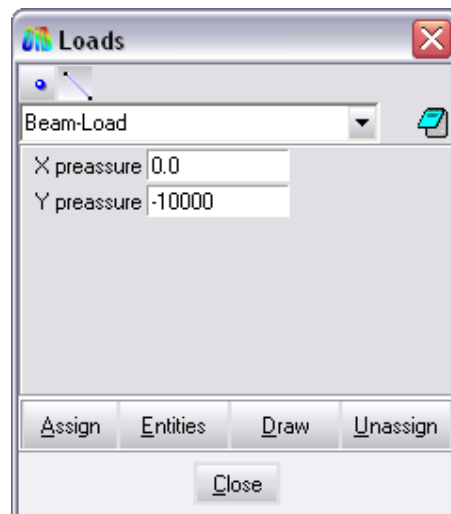
From the pull down menu select:

➤ **Data-> Loads**

Open the line tab (  )

Define beam load for the beam:

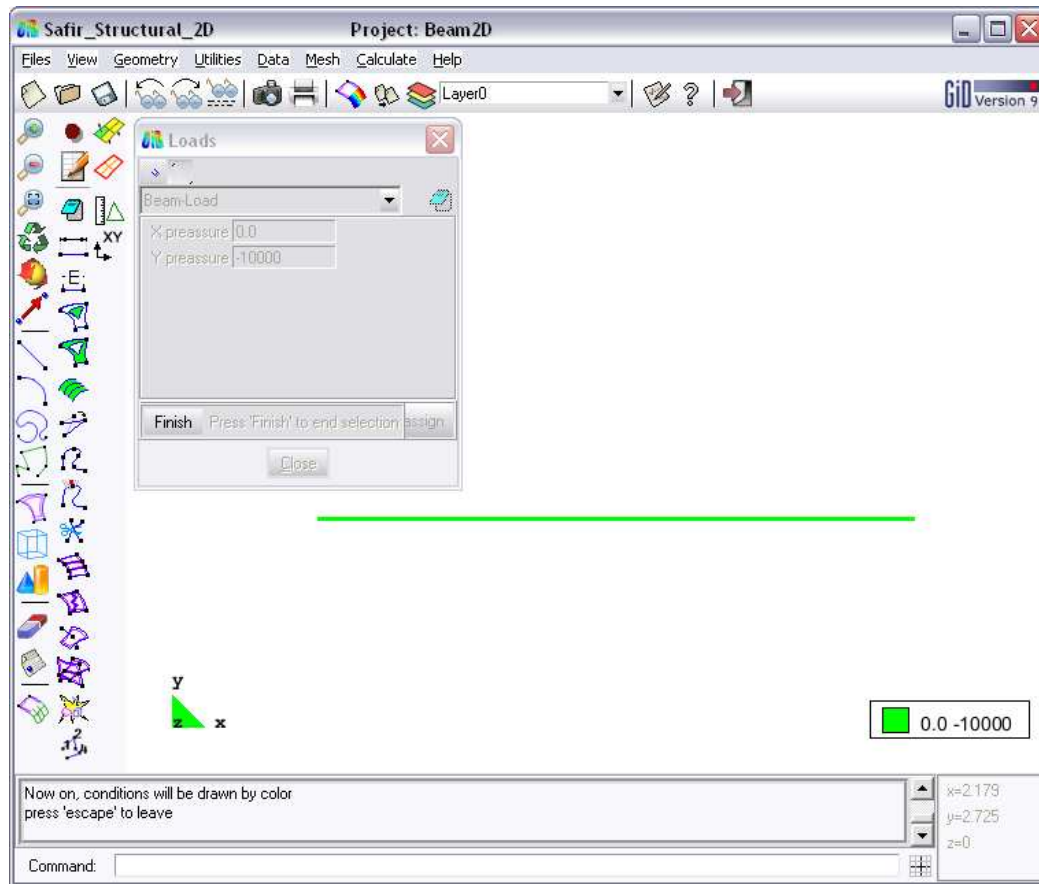
For Y-force, put **-10 000 N/m**



Click on **Assign**

Select the line

Click on **Finish** or press **[Esc]**



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

## 5. Assign temperature files (.TEM files)

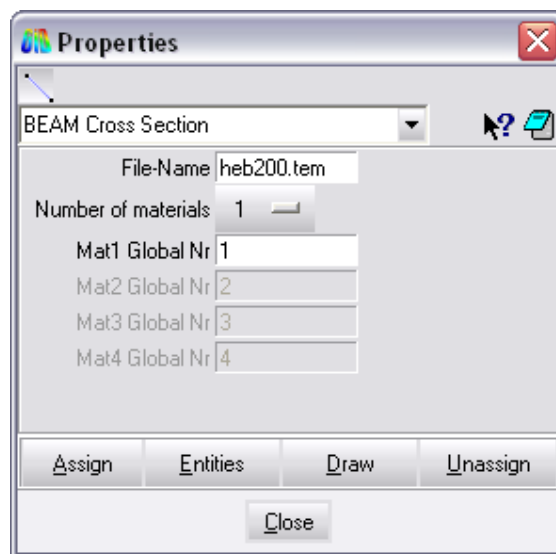
From the pull down menu select:

➤ **Data-> Properties**

In the dial change **Safir.tem** by your section file name **HEB200.tem**

Choose a number of materials of **1**

⚠ *The number of material is the number of material used in the .tem file.*

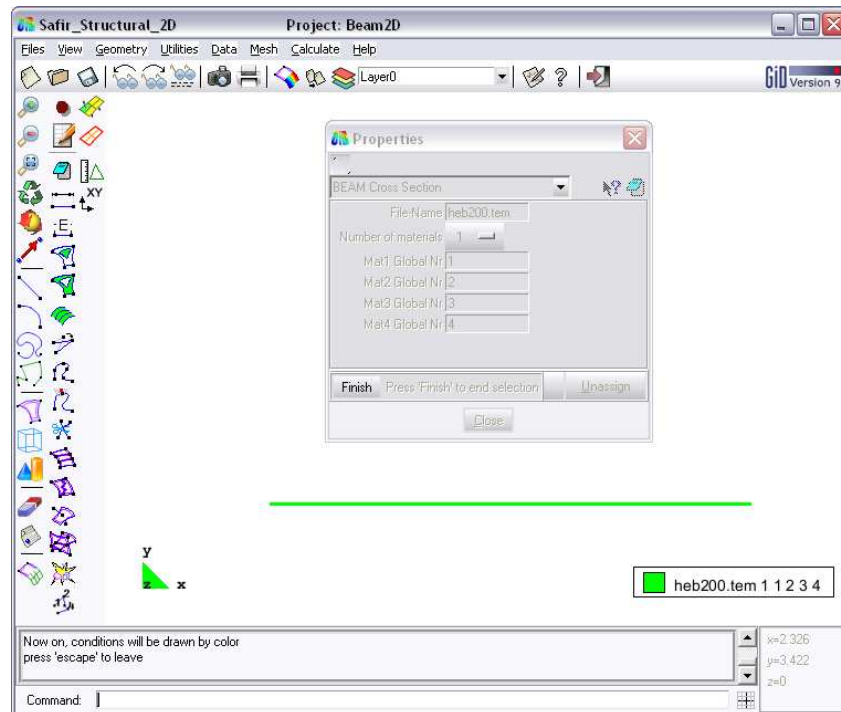


Click on **Assign**

Select the line

Click on **Finish** or press **[Esc]**





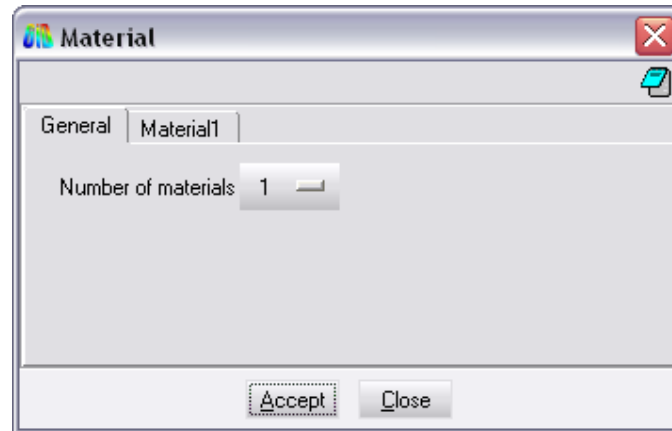
In the dial box, click on **Draw-> Colors** to display the constraints and press on **[Esc]** or click on **Finish** to quite this view mode

⚠ Now you have to open the *heb200.gid* file you already realized (Exercise n°2).  
Select the *.TEM* files (*heb200.tem*) and copy it into your new "Beam2D.gid" directory

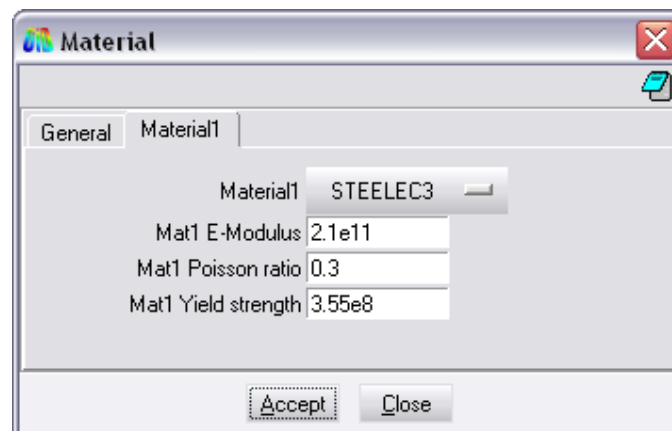
## 6. Define global materials:

To define global material, select from the pull down menu:

➤ *Data->Material*



In this case, the number of material is equal to one and its properties are defined as below



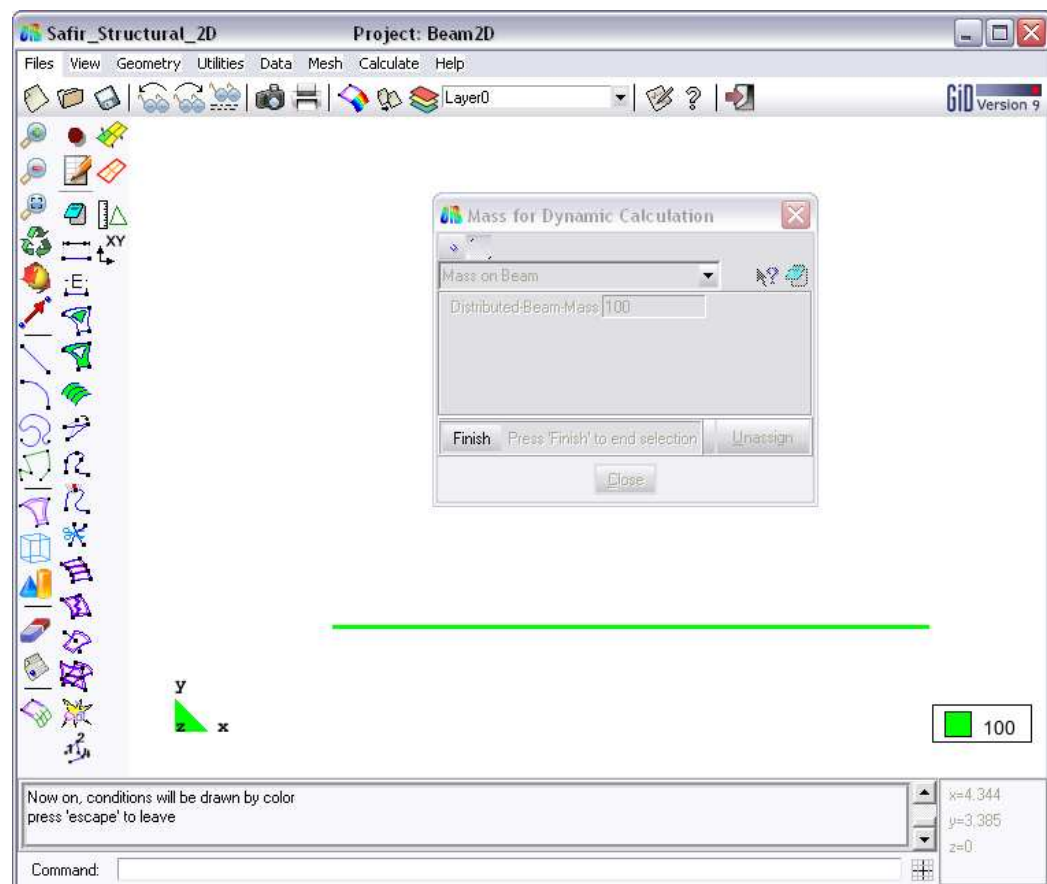
Click on *Accept* to save your modifications

## 7. Define the Mass

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

➤ *Data-> Mass*

Select the  tab and put **100 kg/m** as Distributed-Beam-Mass



Assign the mass to the element

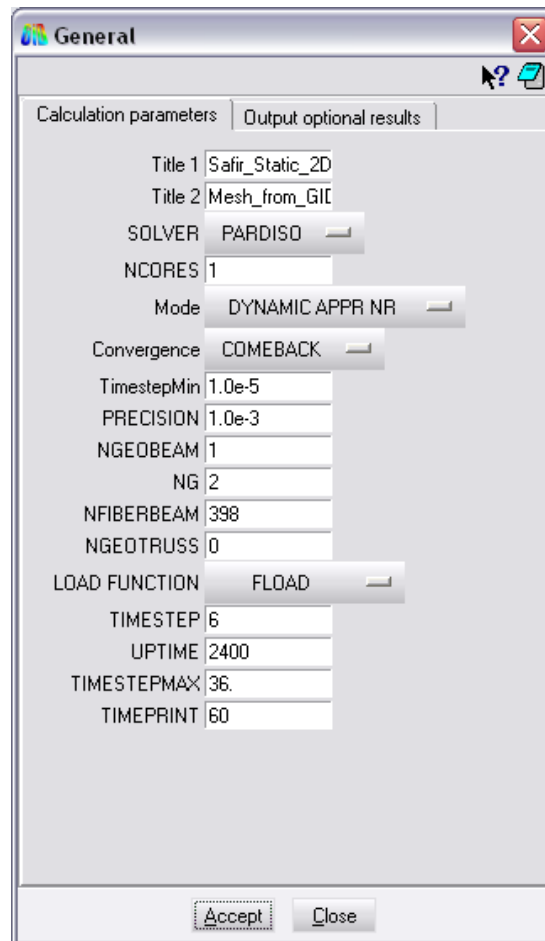
Press on **[Esc]** or click on *Finish*

## 8. Problem data

Select from the pull down menu:

➤ **Data->Problem Data**

Fill the dial box as below



⚠ *Ngeobeam is the number of .TEM file used (1 in this case)*

*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 of each of them (on the first line) and use it as the NFIBERBEAM in your problem data.*

⚠ *You can change TimestepMin, Precision, Timestep, Timestepmax 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 change


## 9. Generate Mesh

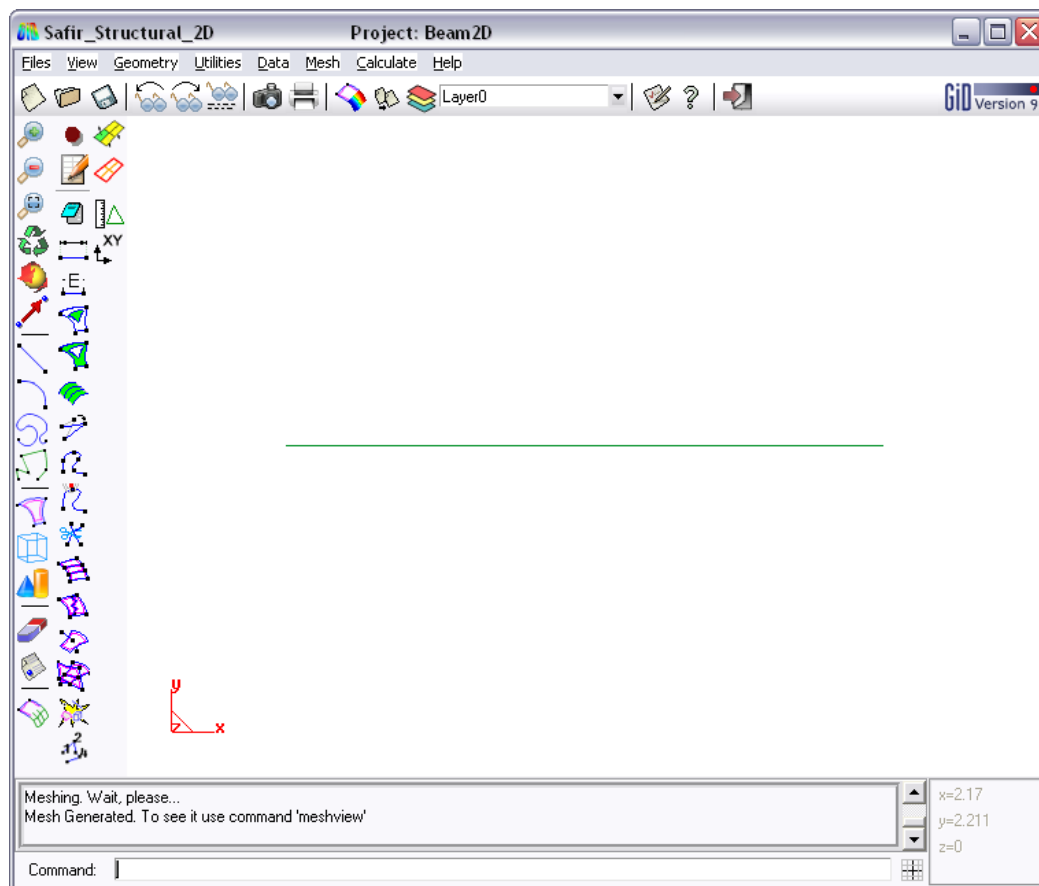
To create meshes select from the pull down menu:

 **Mesh->Generate mesh**

*or use [Ctrl + g]*

Enter 0.2 m as size of elements to be generated

 *A message with the number of nodes and the number of elements will appear. If you are using an evaluation version of GiD, remember that the maximum of nodes allowed is 1010 nodes*



To display element and node numbers select:

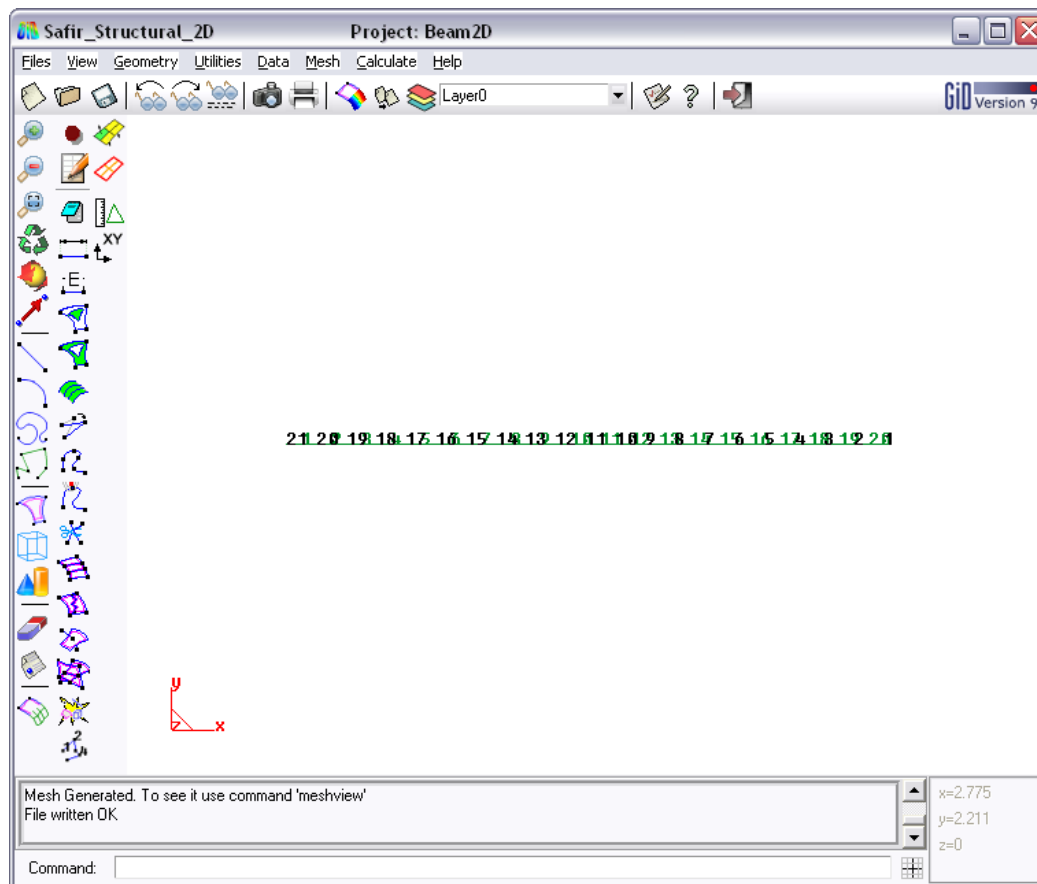
 **View->Label->All**

If you want to zoom, select:

 **View->Zoom->In**

*or* 

Select the area you want to check

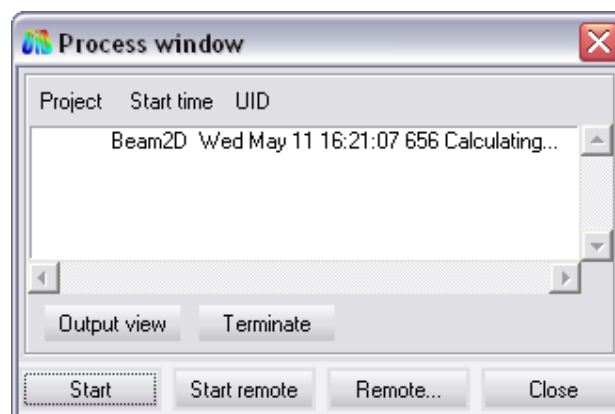


➤ **View-> Lapel-> Off** to quit the label mode

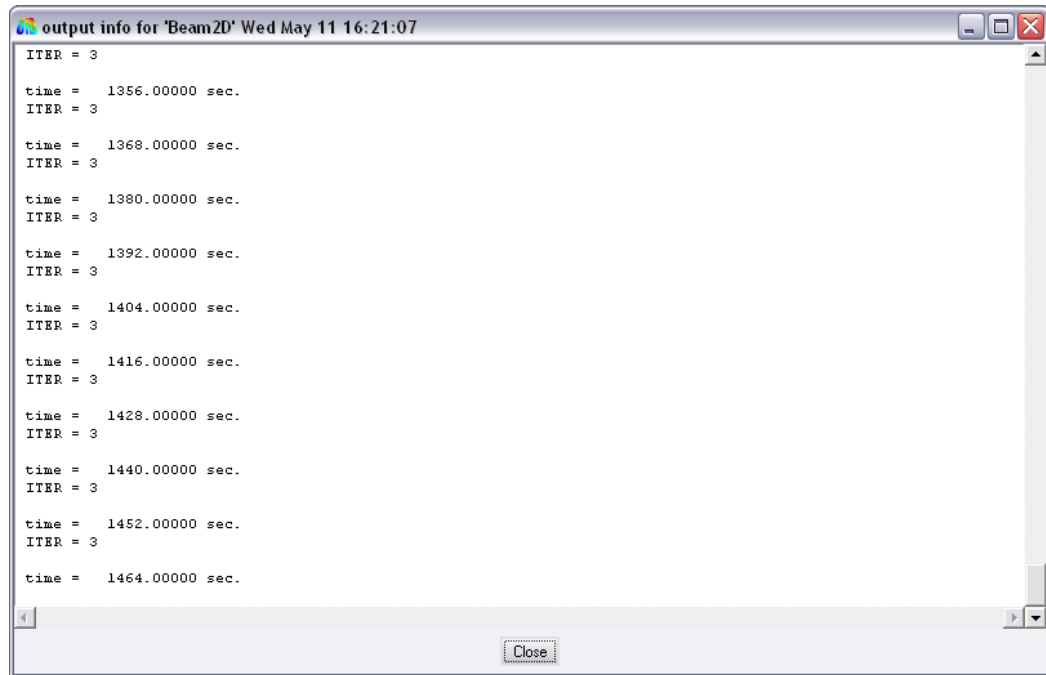
## 10. Start the calculation

Select from the pull down menu:

➤ **Calculate-> Calculate window**



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



⚠ *GiD creates a .IN file in the project directory and starts the calculation. In the output window you can see 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 see the error message in this window.*