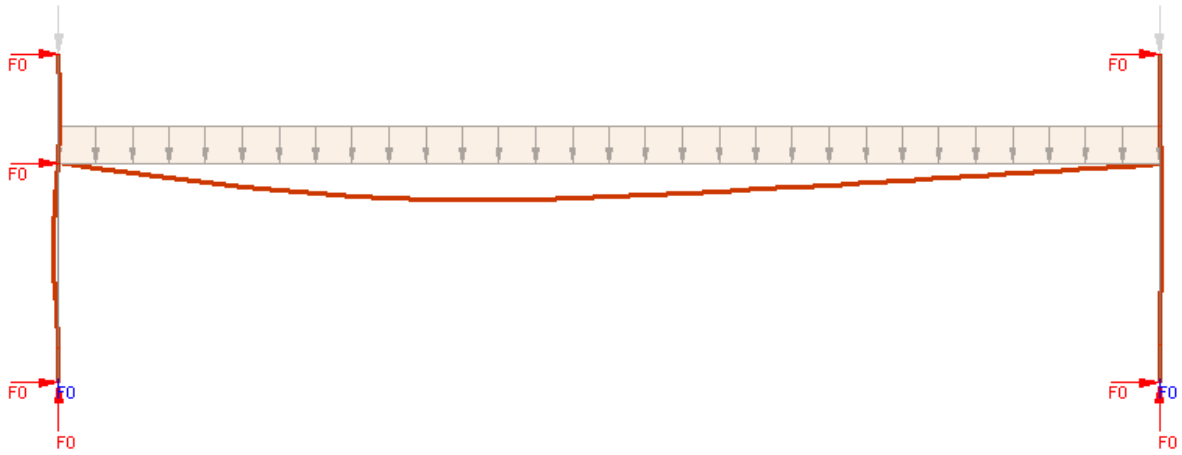
# Example for GID-SAFIR 2D Structural Analysis

## Exercise n°7 - Frame 2D



This example is a 2D-frame consisting of a 15 m composite member (IPE550 plus concrete slab) and 2 columns (HEB220) 3m high. The member has 3 different temperature zones (IPE550B1.tem, IPE550B2.tem and IPE550B0.tem). The left column has temperature HEB220B1.tem, the right column HEB220B0.tem. The load on the member is 30kN/m, the load on the columns is a point load of 900kN on the left side 450 kN on the right side.

# A. Create a frame

## 1. Create sections

In this exercise, we will suppose that you want to import your own fire curve of temperature.
In this case we are going to use 2 temperature curves: ***B1.txt*** and ***B2.txt.*** B0 will be the .TEM file with cold calculation.

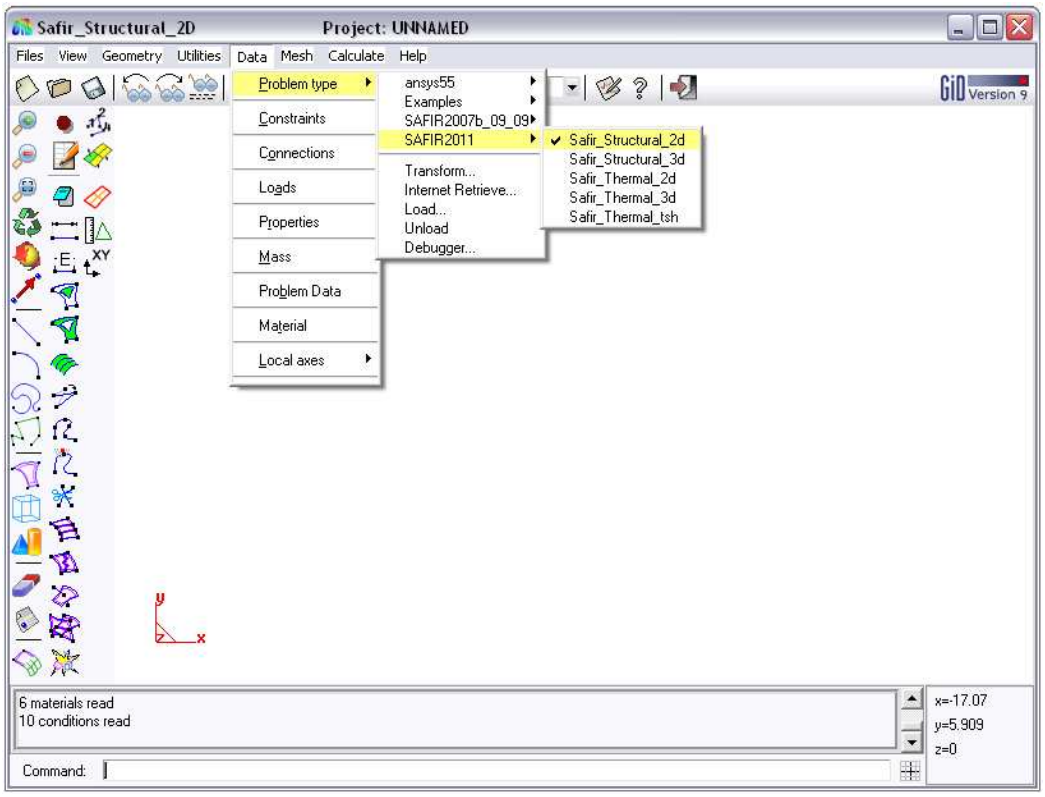
In order to create new sections, you will have to do exactly the same operations that shown in Exercise 2 and 3.

Now create 5 .Tem files for 2D calculation:
- One IPE550B1 file, temperature curve B1 with a section partially exposed to fire. This section is the section created in exercise 3.
- One HEB220B1 file, temperature curve B1 with a section totally exposed to fire. To create this section, do exactly the same steps than in exercise 2, with the difference that the section is a HEB220 fully exposed to fire and not exposed to ISO fire, but to the temperature curve defined in the ***B1.txt*** file
- One HEB220B0 and one IPE550B0 file (in this case B0 is the cold calculation, to create those sections you just have to skip the part 3 of exercises 2 and 3).
- One HEB220B2 file with temperature curve B2 and a section fully exposed to fire (as section HEB220B1)
- One IPE550B2 file with temperature curve B2 (as section IPE550B1)

## 2. 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 doesn't work*

Enter a file name, eg.: Frame2D

GiD creates a directory with the name Frame2D.gid

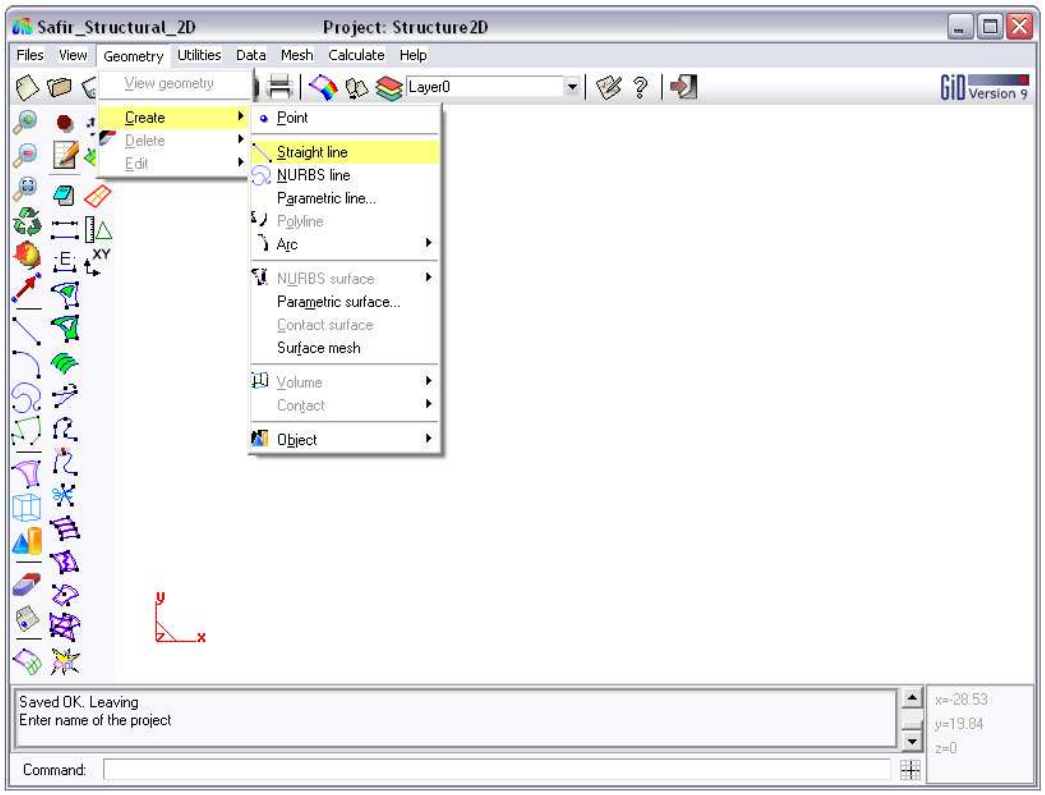
GiD creates a number of system files in this directory.

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

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

From the pull down menu select:

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

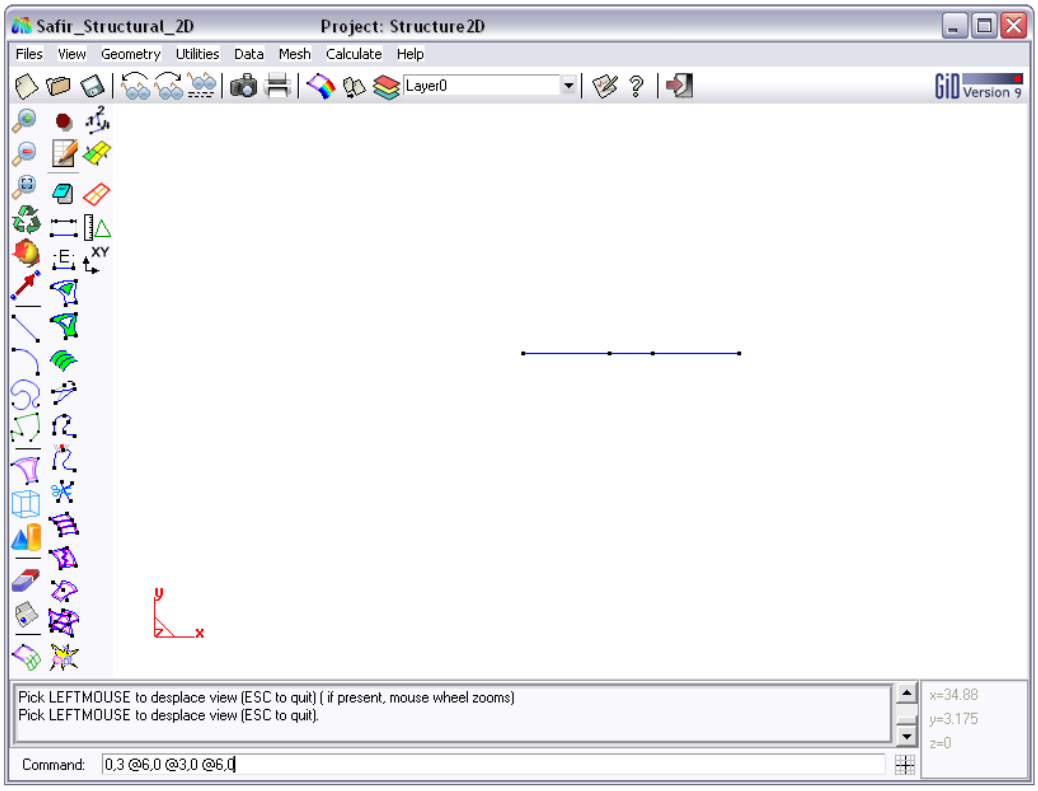



In the command line (bottom of the widows) enter the coordinates of the line points:

***0,3 @6,0 @3,0 @6,0***

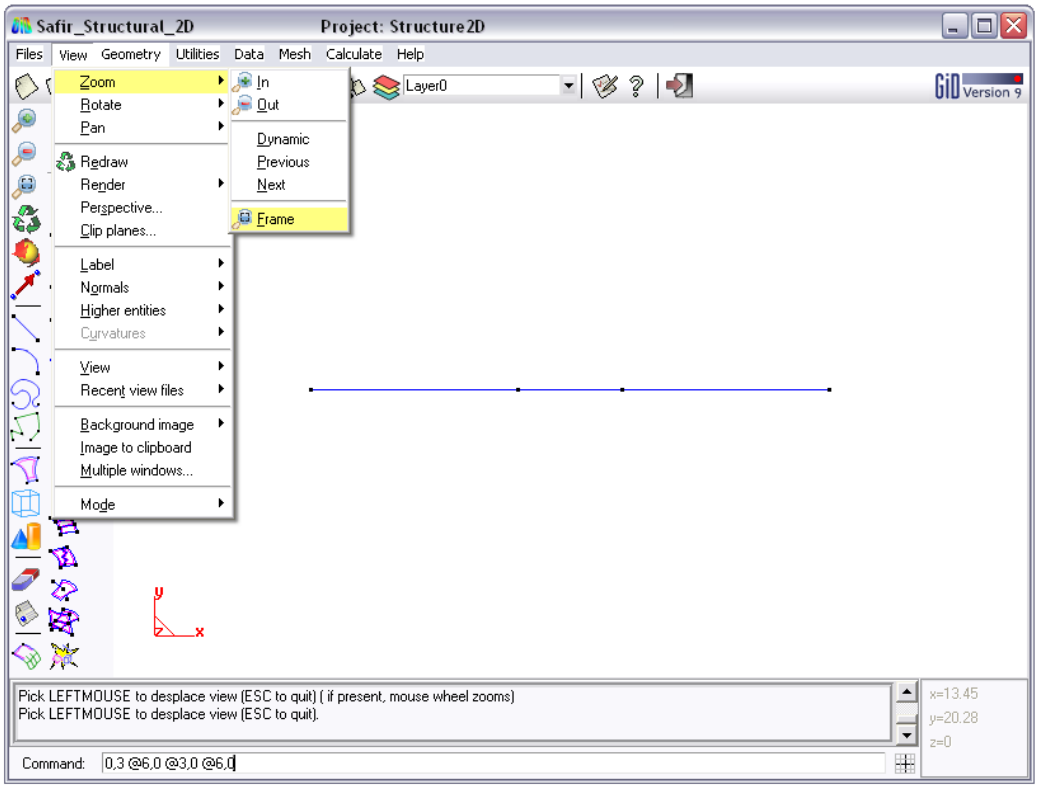
Press ***[Enter]*** to confirm

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




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

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



Select

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

In the command line enter:

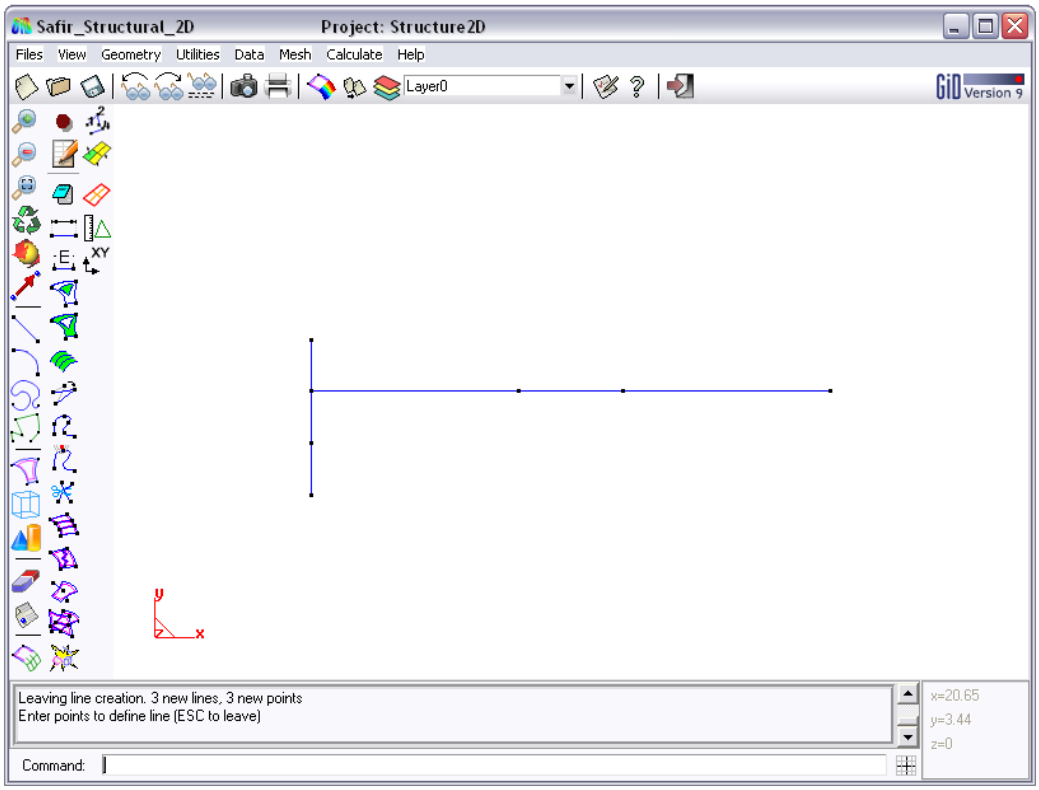
***0,0***

Then Press ***[Enter]***

***0,1.5***

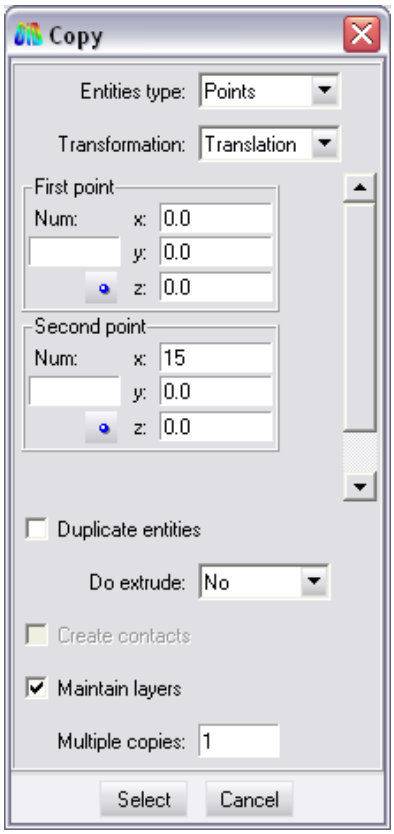
Then Press ***[Enter]***

Then type ***[Ctrl + a]*** and pick the left end of the member line, then press ***[Ctrl + a]*** to quit the selection mode and type in the command line ***@0,1.5*** and press once on ***[Enter]*** and twice on ***[Esc].***



Select from the pull down menu:

- ***Utilities-> Copy*** ***or [Ctrl + c]***

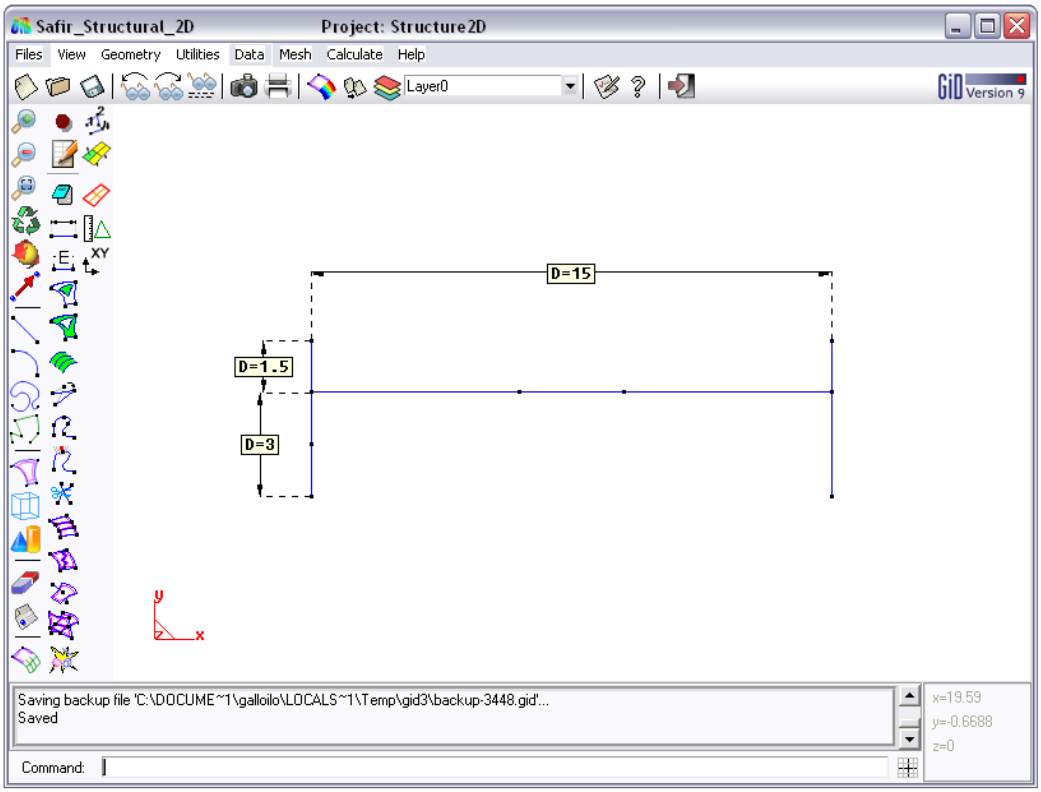



In the dial box, for the second point, put ***x = 15 m*** and select the upper and the lower point of the left column and click on ***Finish*** or press ***[Esc].***

Select

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

Use the node selection mode ***[Ctrl + a]***, pick up the top then the middle and the lower point of the right column and press ***[Esc]*** twice to leave this mode.




⚠ *We divided the system line for the member according to the temperature zones and the left column because we want to assign a x-constraint at 1.5m high.*

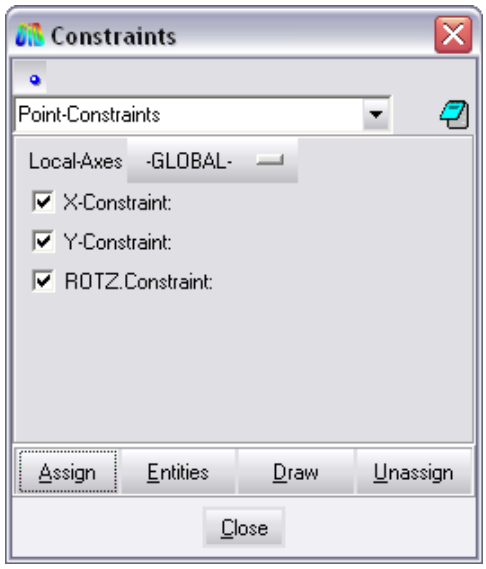
⚠ *If you want to see a distance between 2 points, use the*  *button and click on the two points of your choice and click in the place you want to put the dimension text.*

*To delete a dimension you have to select Geometry-> Delete-> All types or*  *to click on the target you want to delete and press [Esc]*

## 4. Define constraints for the supports

From the pull down menu select:

- ***Data-> Constraints***



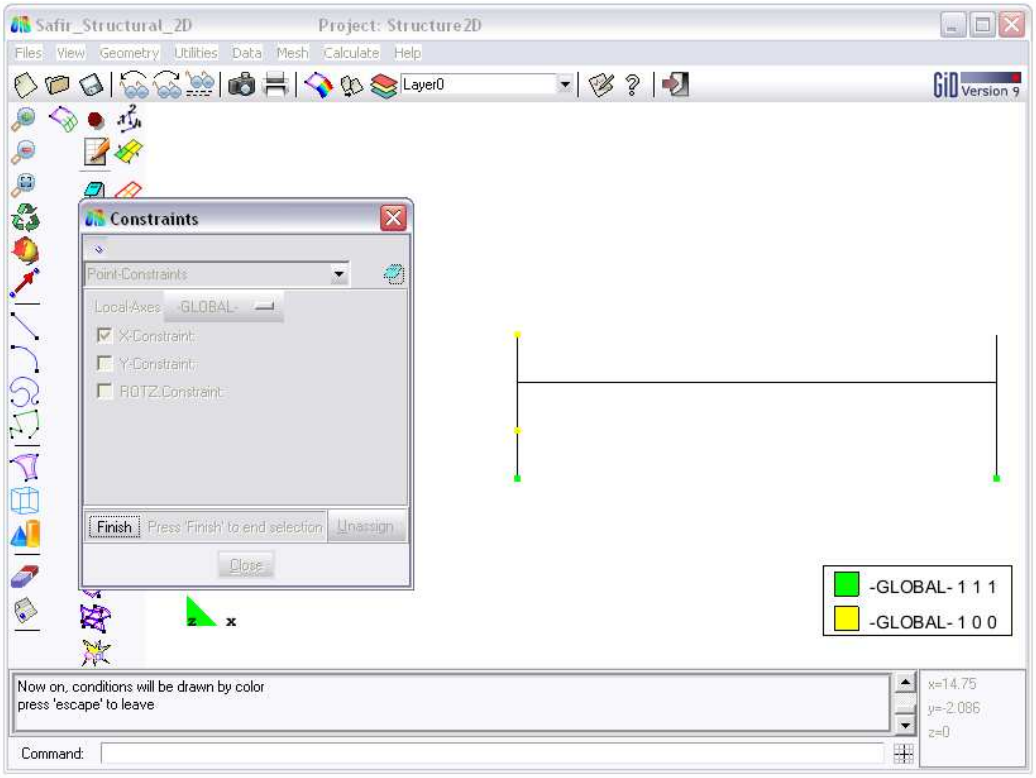

Select x,y constraint and z rotation constraint and click on ***Assign***

Click on the columns base points

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

Select x-constraint and assign it to columns top points and to the mid point of the left column.

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



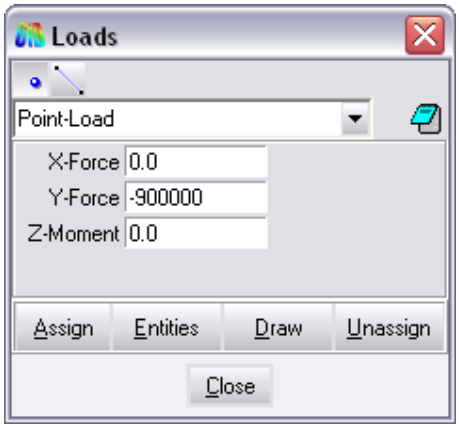

## 5. Define loads

From the pull down menu select:

- ***Data-> Loads.***

Define point loads for the columns:

For Y-force, put ***-900 000 N***

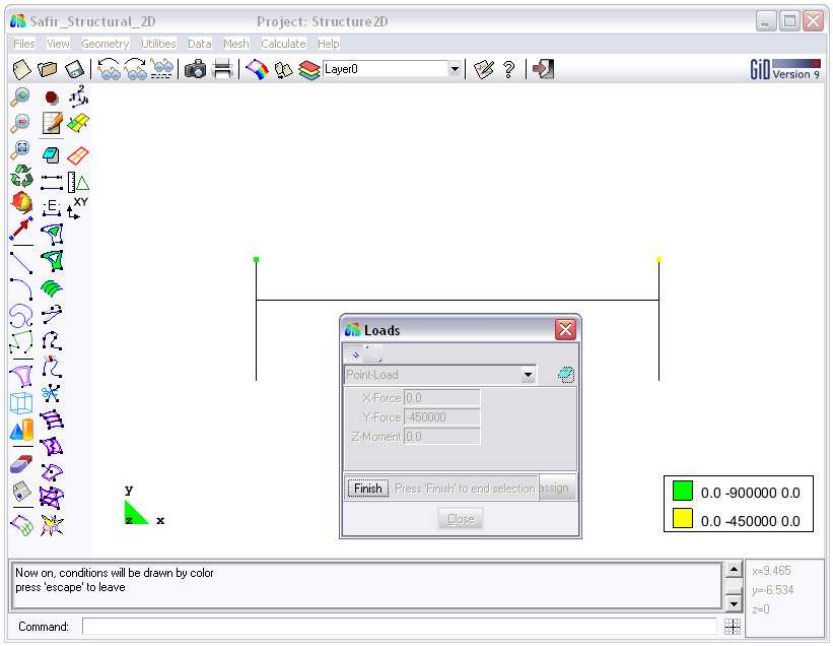



Click on ***Assign***

Select the node on the top of the left column

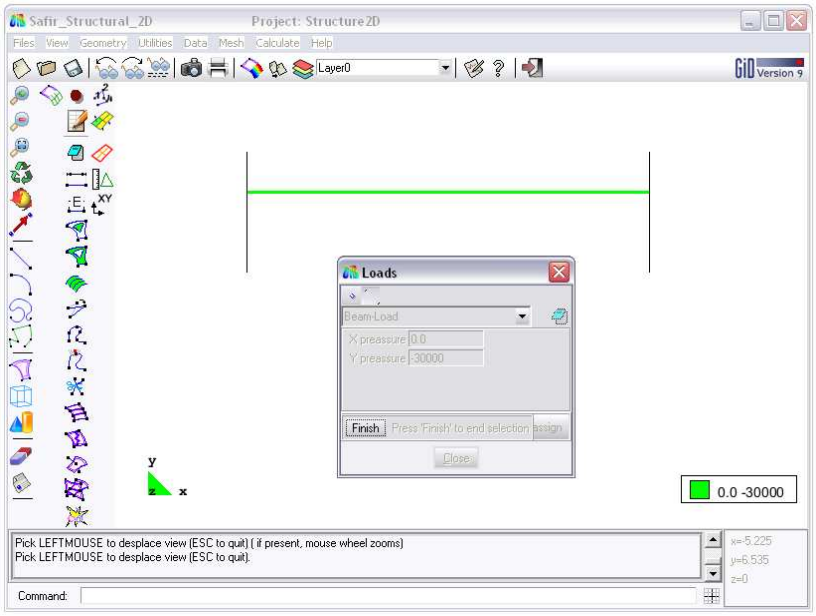
Click on ***Finish*** or press ***[Esc]***

Do the same operation to assign an Y-force of ***-450 000 N*** on the top of the right column




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

In the Loads dial box, select the  tab. As for a point-load, assign a Y pressure of -30 000 N to all the member line



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

From the pull down menu select:

- ***Data-> Properties***

*The objective is to associate section files (.TEM) to lines. In that regard, in the first part of this exercise you created files IPE550B1, IPE550B2, IPE550B0, HEB220B0 and HEB220B1.*

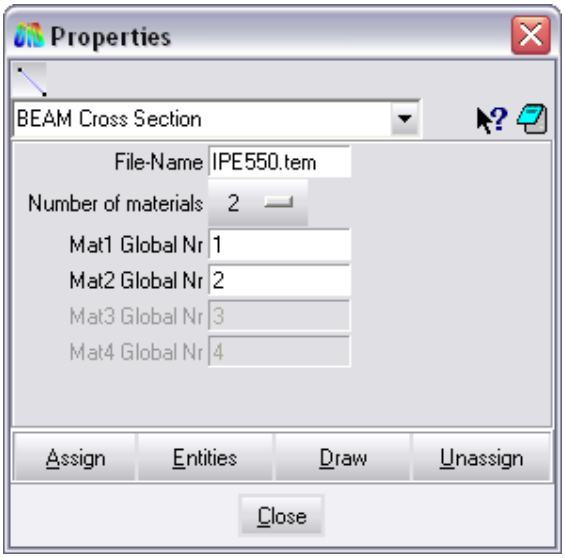
*IPE550 files have 2 materials (Siliconc en and steelec3).*

*HEB files have 1 material (steelec3).*

For the IPE550B0 file:

In the dial box change ***Safir.tem*** by your section file name (***IPE550B0.tem*** in this case)

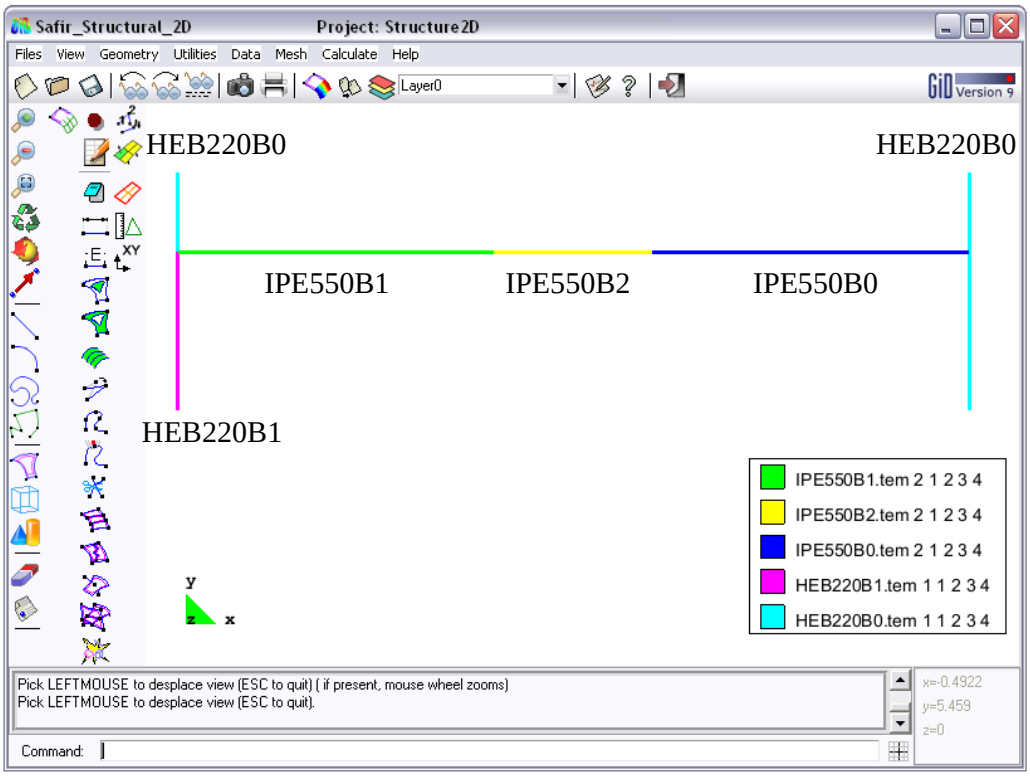
Choose a number of materials of ***2***




Click on ***Assign***

Select the right part of the member line as shown in the figure below

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




Then do the same operation to assign IPE550B1, IPE550B0, HEB220B0 and HEB220B1 section as shown in the figure.

⚠ *Don't forget to change the number of material from two to one when you wil assign HEB sections*

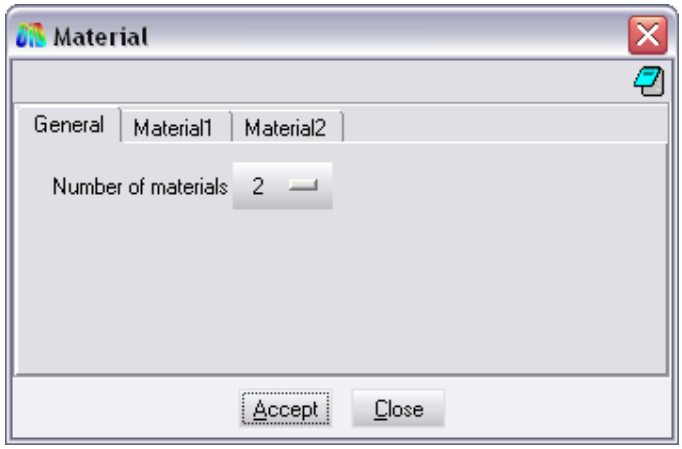
In the dial box, click on ***Draw-> Colors*** to display the constraints and press on ***[Esc]*** or click on ***Finish*** to quite this view mode

⚠ *Now you have to open the GiD-Safir tutorial file and select the 5 .TEM files (IPE550B1, IPE550B2, IPE550B0, HEB220B0 and HEB220B1) and copy them in this directory*

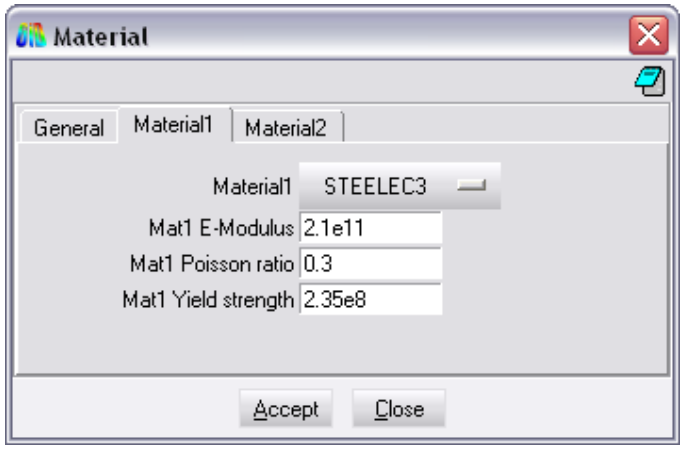
## 7. Define global materials:

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

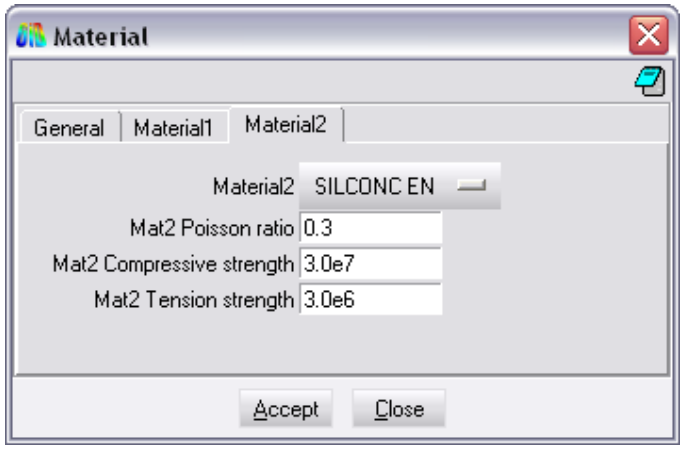
- ***Data->Materials***




Change the number of material from one to two and complete as below




Do the sam operation with the tab ***Material2***



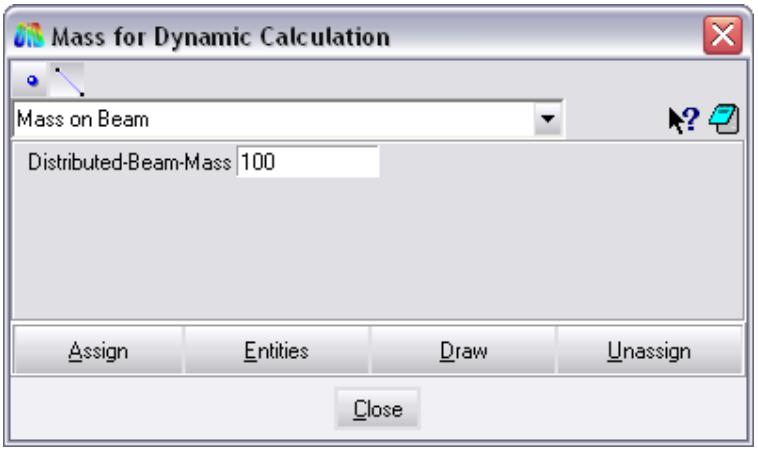

Click on ***Accept*** to save your modifications

## 8. Define the Mass:

To define 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



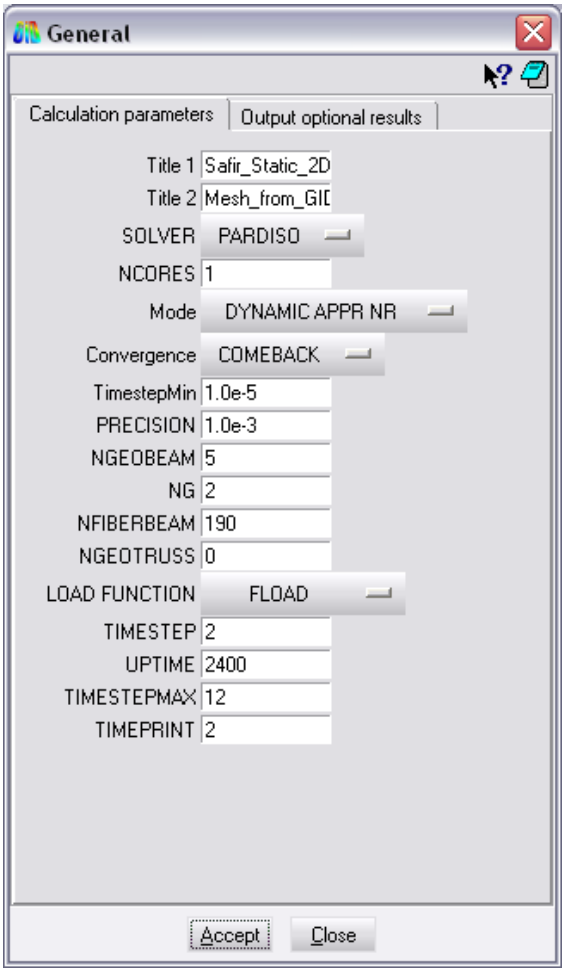

Assign the mass to all the elements.

## 9. Problem data

Select from the pull down menu:

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

Fill the dial box as below




⚠ *Ngeobeam is the number of .TEM files (5 in this case). Nfiberbeam is equal to 190 in this case, the only way to know this number is to open .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.*

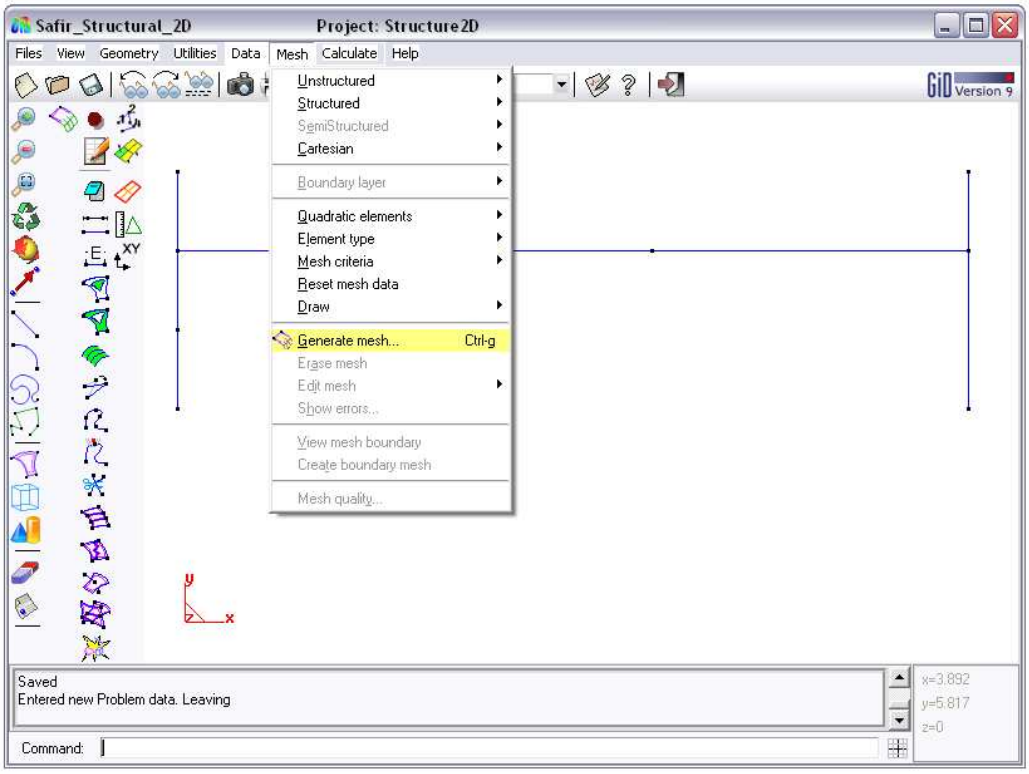
⚠ *You can change TimestepMin, Precision, Timestep, Timestepmax and Timeprint as needed but you have to be careful that your UPTIME is less or equal to the UPTIME used for sections 2D calculation*

Click on ***Accept*** to save your change

## 10. Generate the Mesh

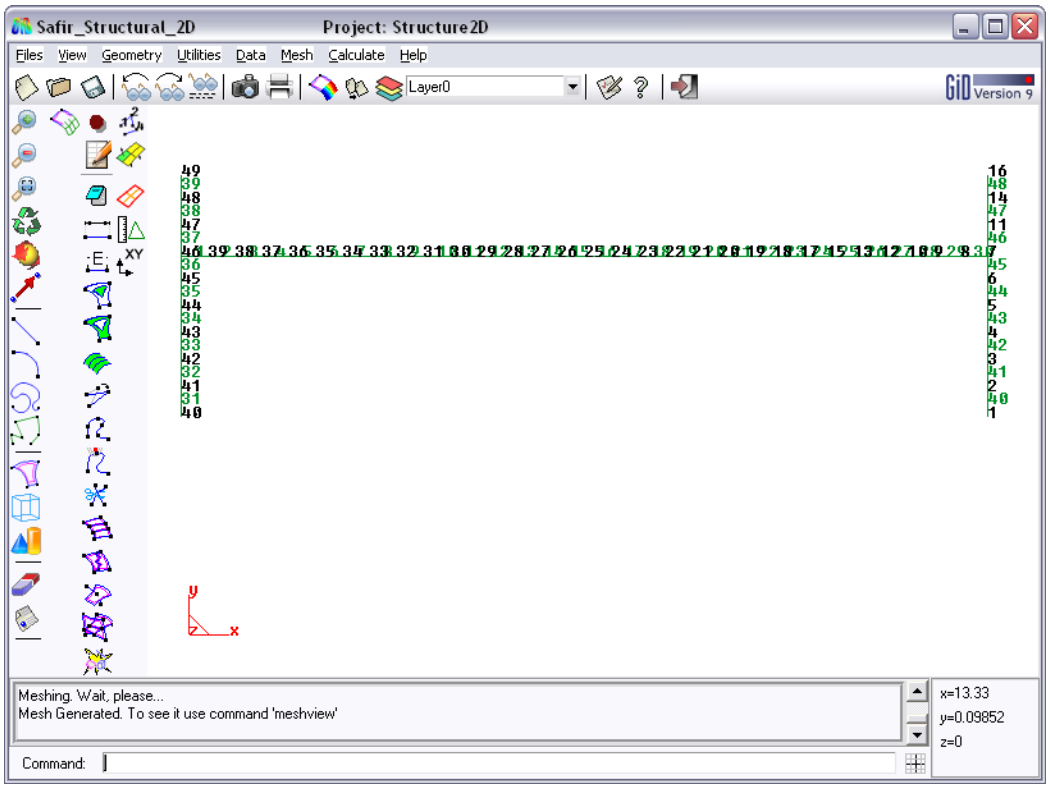
To create meshes select from the pull down menu:

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




Enter 0.5 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*



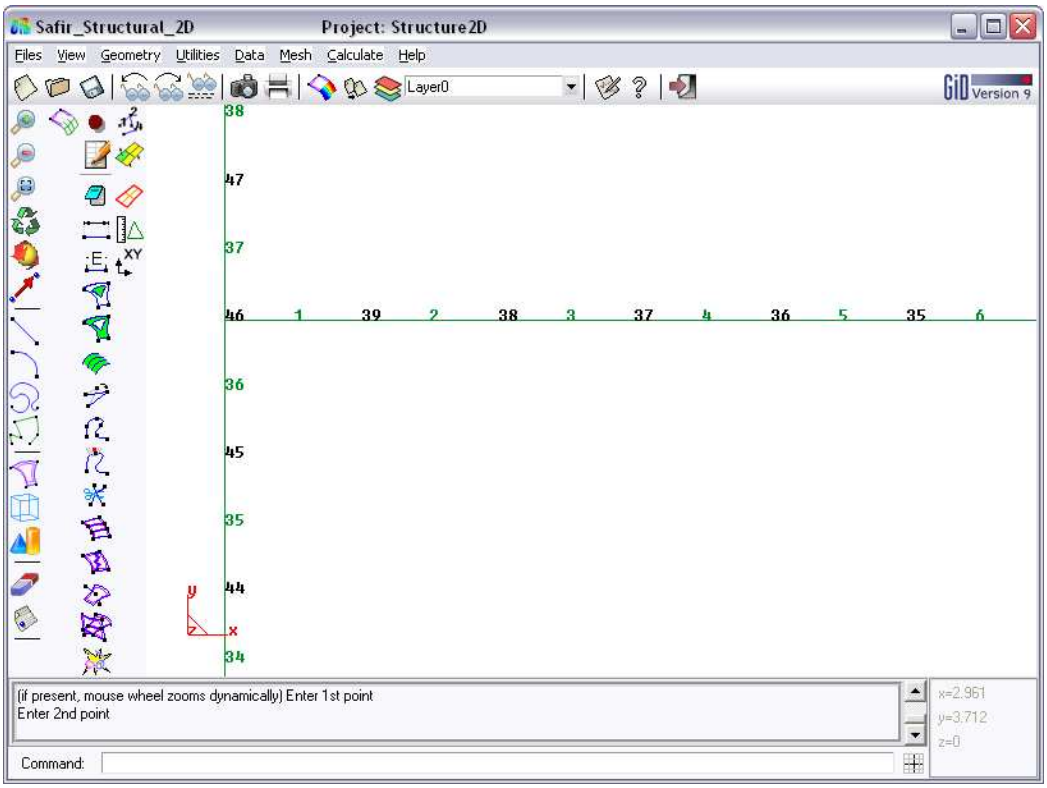

To display element and node numbers select:

- ***View->Label->All***

If you want to zoom use select:

- ***View->Zoom->In*** ***or*** 

Select the area you want to check

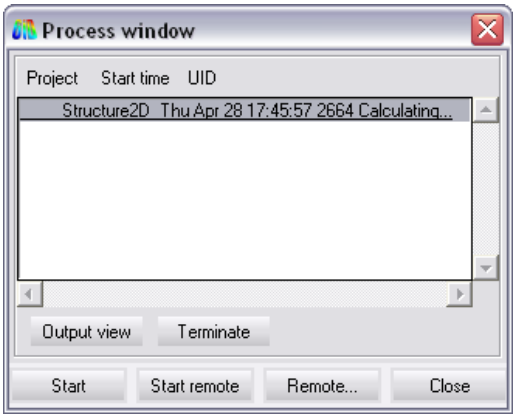



- ***View-> Lapel-> Off*** to quit the label mode

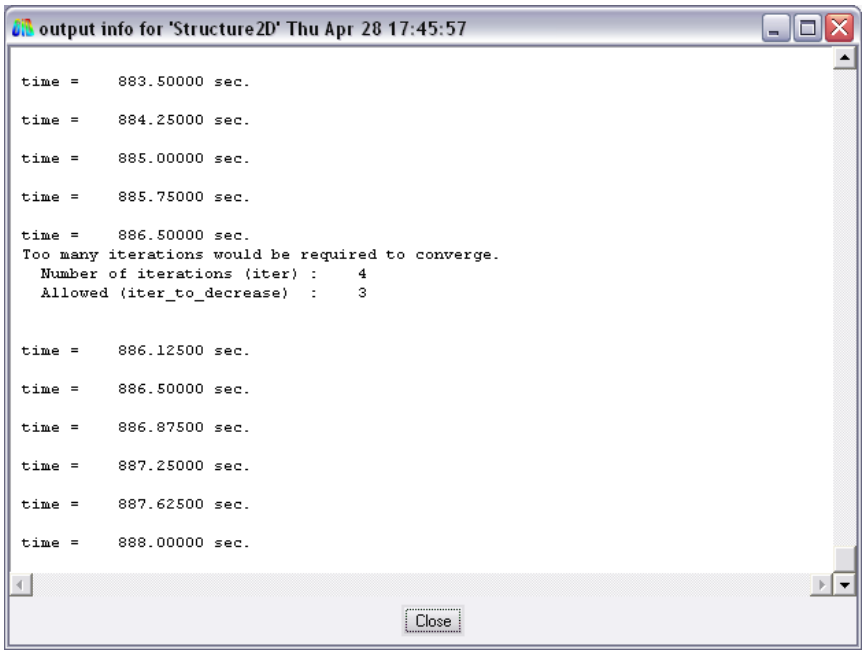
## 11. Start the calculation

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.*

# B. Create a frame with modified connection

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

The objective is now to save this file with another name in order to compare effects of connections modifications on the frame resistance.

- ***Files->Save as***

Enter a file name, eg.: ***Frame2D_R***

GiD creates a directory with the name Frame2D_R.gid

GiD creates a number of system files in this directory.

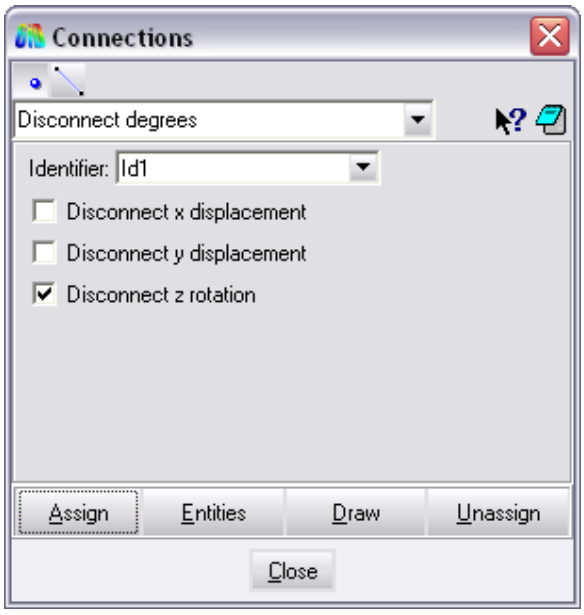
When you start the SAFIR calculation the Safir . IN, .OUT and .TEM files will be created here in this directory.

## 2. Use the "Connection" function

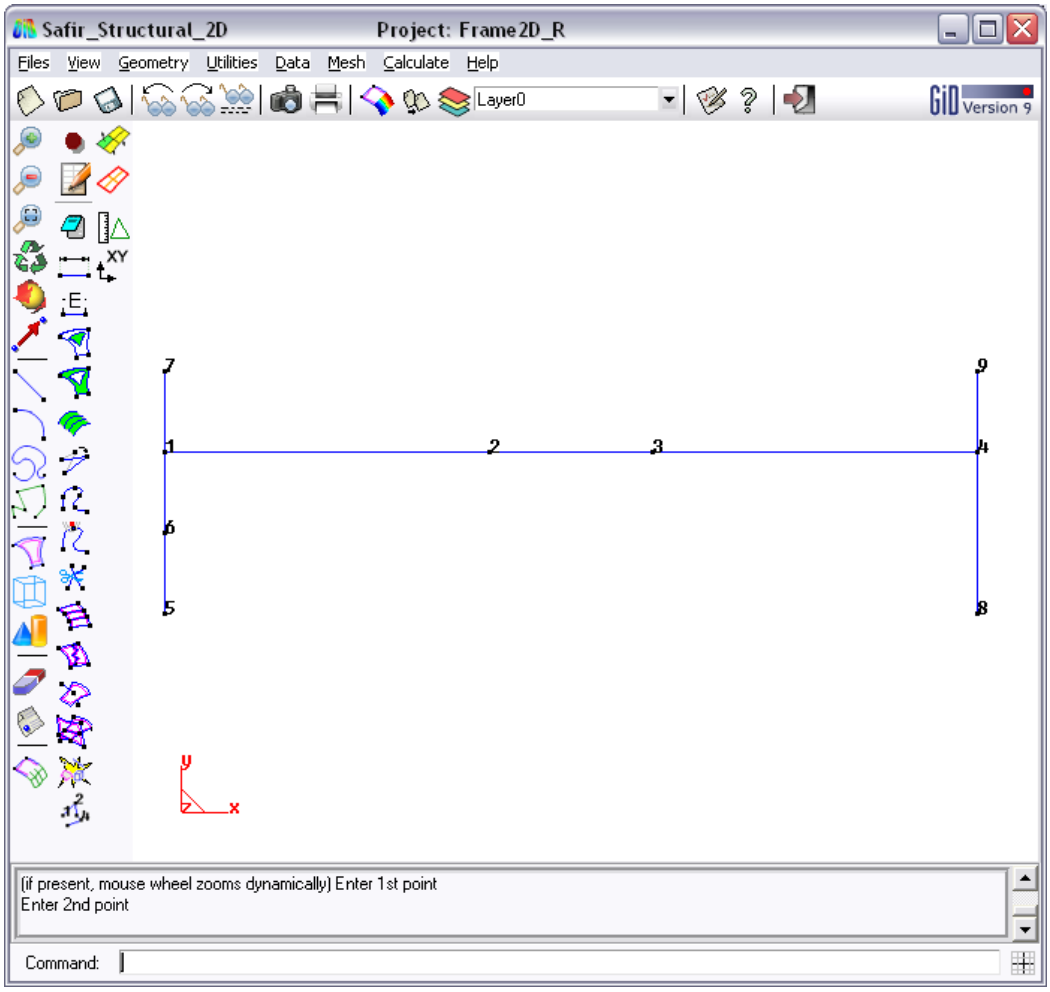
To disconnect certain degrees of freedom on the end-node of beams or shells select from the pull down menu select:

- ***Data->Connections***

Then fill as shown below:

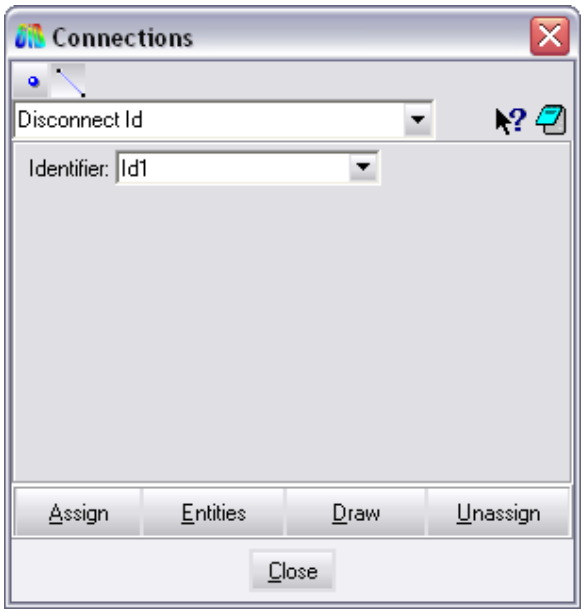


Click on ***Assign*** and select points ***1***



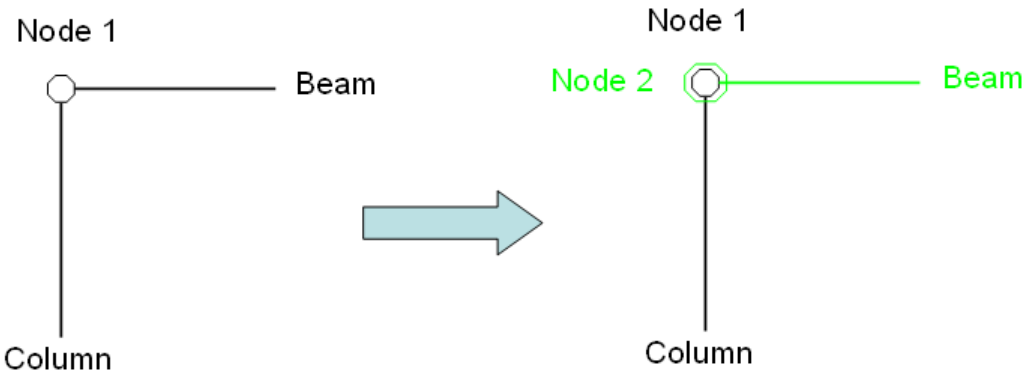

Press ***[Esc]***

Now on the beam tab select ***Id1*** as Identifier



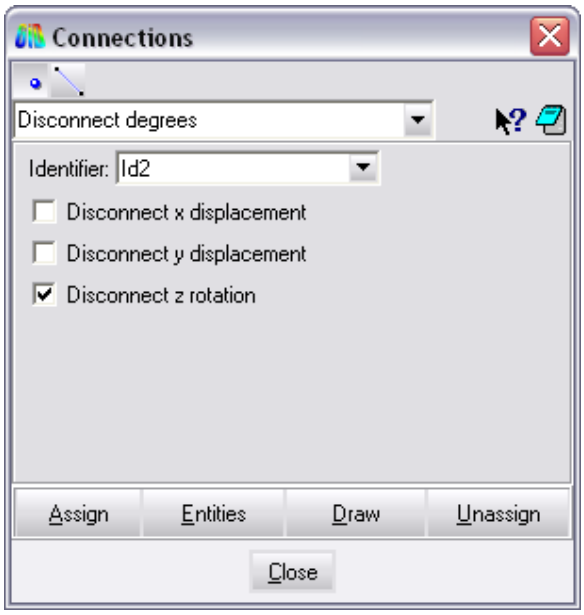

Click on ***Assign*** and select beams ***1-2*** and press ***[Esc]***

⚠ *The software has now created a new point in the same position than node 1. Node 1 is now only include in the column "5-7" and the new node "1bis" is only include in the beam "1-2" fully connected to the column 5-7, excepted for rotation in z direction.*



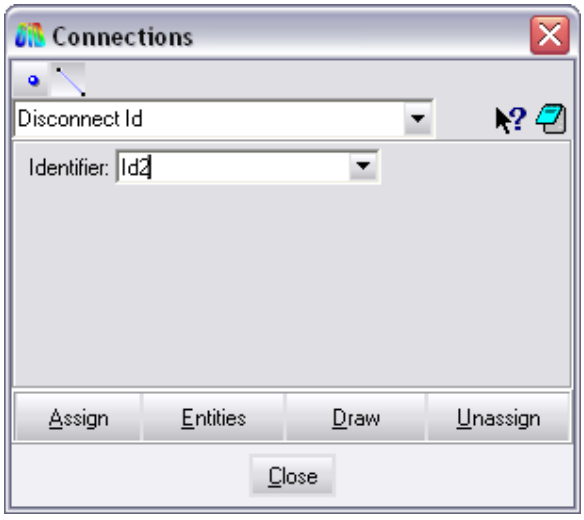
Now, the objective is to disconnect the Z rotation on node 4

Fill the connections windows as shown below:




Click on ***Assign***, select node 4 and press ***[Esc]***

Now open the beam tab and select ***Id2*** as identifier




Click on ***Assign*** and select lines ***3-4*** and ***[Esc]***

⚠ *You have now disconnected Z rotation on node 4 from beam 3-4 to column 8-9*

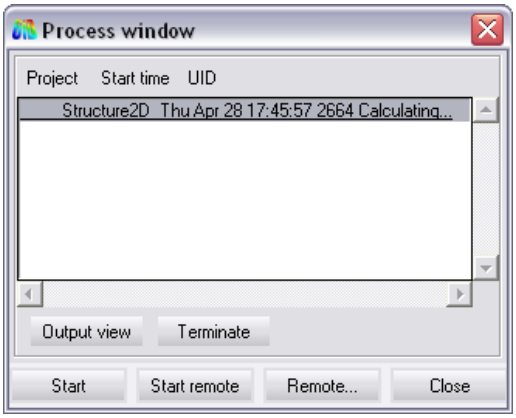
To create meshes select from the pull down menu:

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

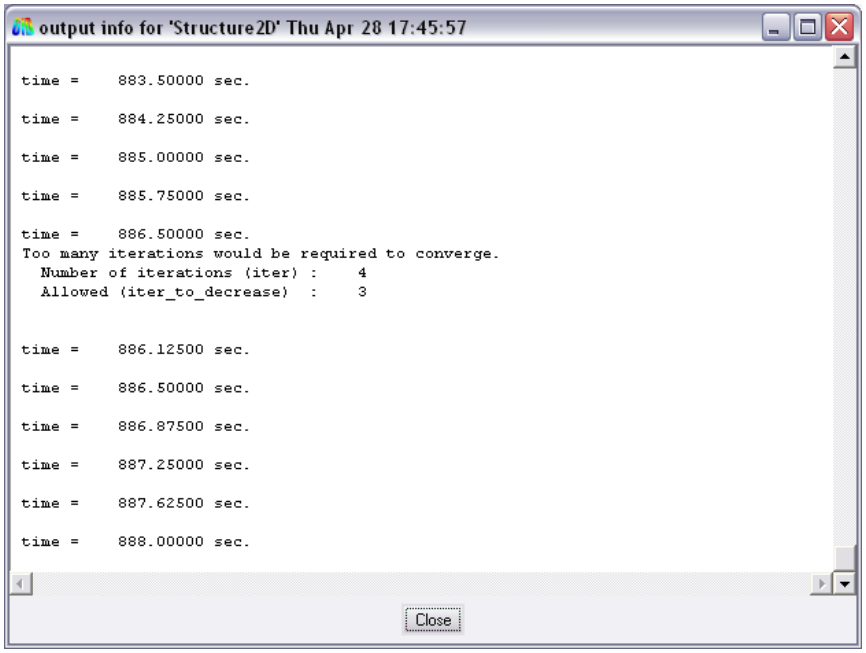
## 3. Start the calculation

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.*