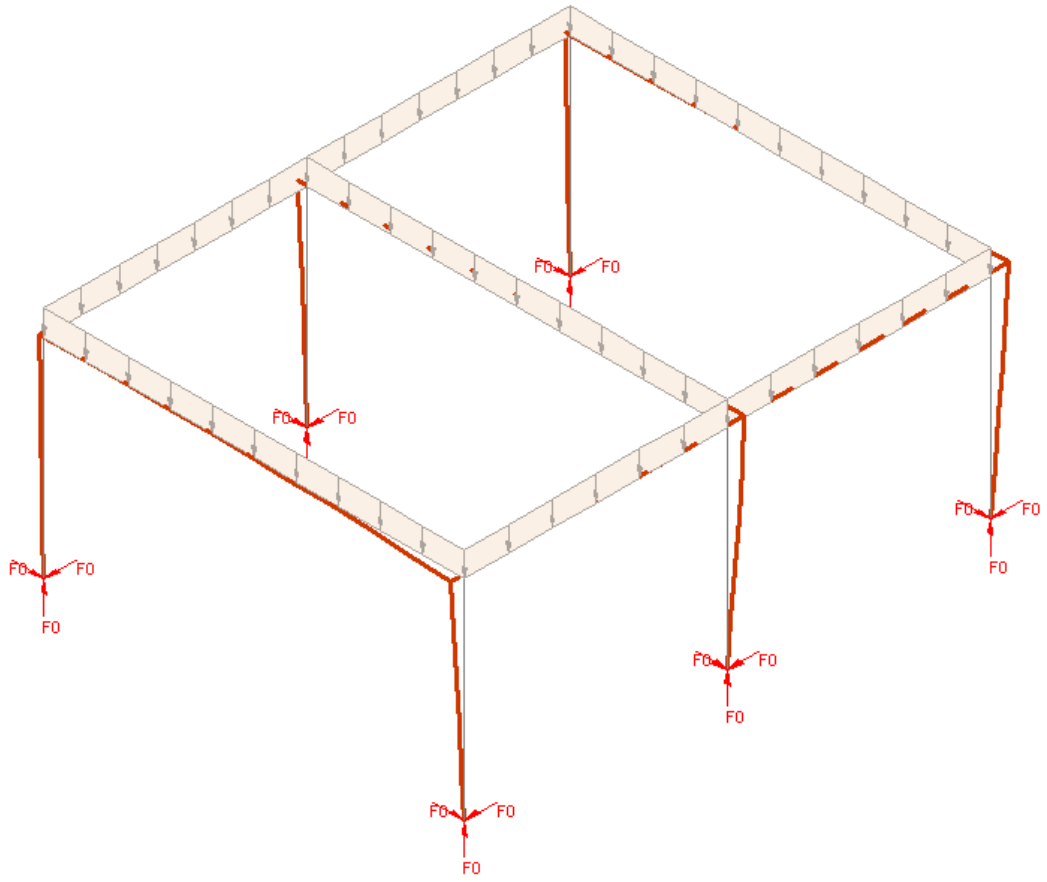
Tutorial for GID-SAFIR 3D Structural Analysis

Exercise n°8 Hall 1 3D



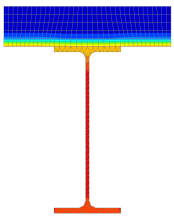
1. Create sections

This example is a 3d frame with the following system geometry and beam profiles. There are 3 temperature zones, the .tem files used are HEB220B0.tem (cold), HEB220B1.tem (715° max), HEB220B2.tem (476°max); IPE550B0 (cold); IPE550B1.tem (715° max), IPE550B2.tem (476° max).

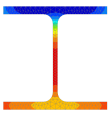
IPE550B1 section is the same section than created during exercise 3. B0 are cold section and B2 is an other user-temp curves attached in the « section » file. We add torsion for 3D calculation, as explained in exercise 2, part 2.

HEB220.tem and IPE550.tem steel Mechanical properties: E-Modulus = 2.1e11, Poisson ratio = 0.3 and Yield strength = 3.55e8 N/m^2.

IPE550.tem concrete properties: Poisson ratio = 0.3, Compressive strength = 3.0e7 and tension strength = 0



IPE550

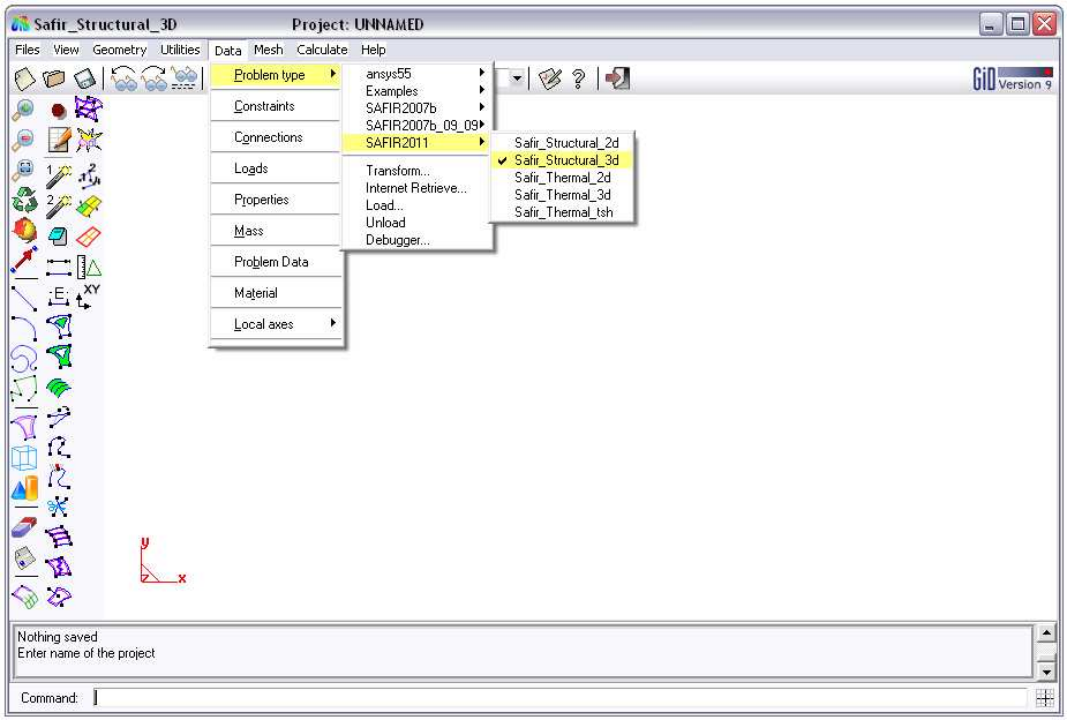


HEB220

2. 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.: ***Hall1_3D***

GiD creates a directory with the name ***Hall1_3D.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 placed in this directory.

3. Create the system geometry

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

- *View->Rotate->isometric*

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

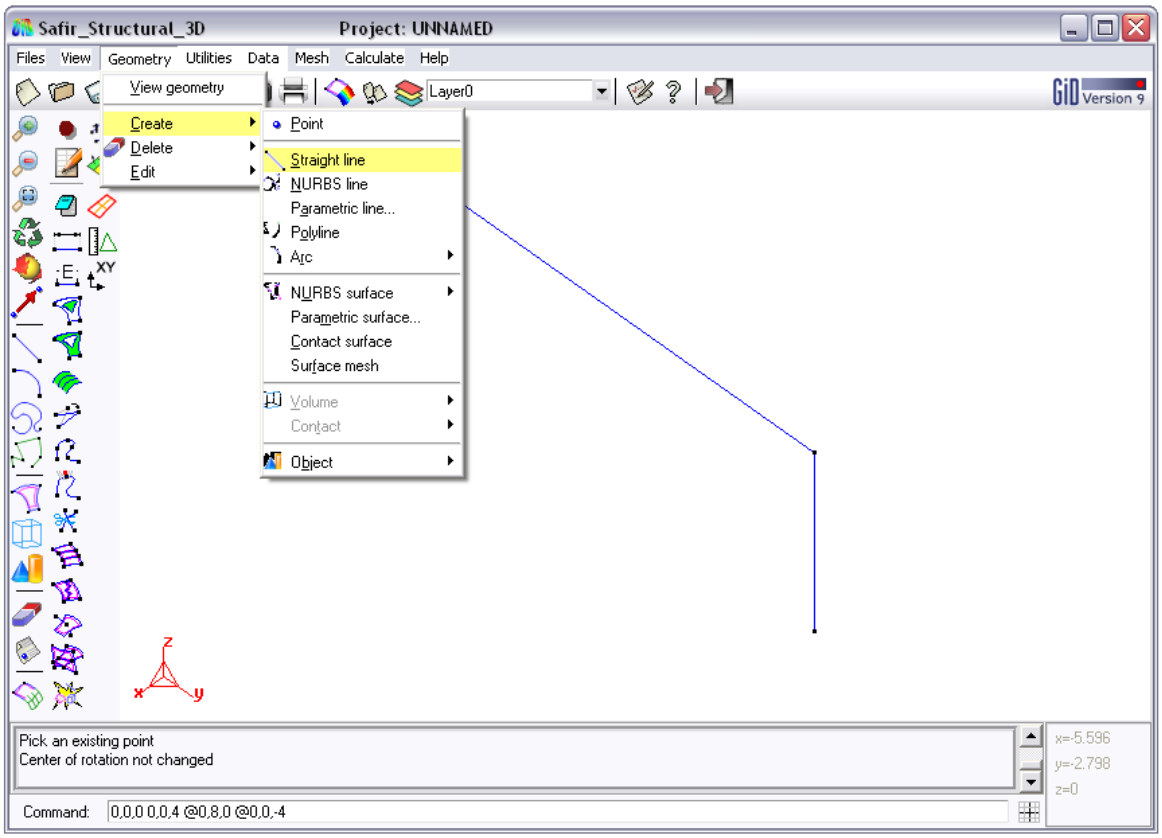
- *View->Rotate->Trackball* *or [F7] or* 

Create the system lines:

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

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

0,0,0 0,0,4 @0,8,0 @0,0,-4 and press *[Enter]*



The first frame is created.

To create the 2 other frames Select:

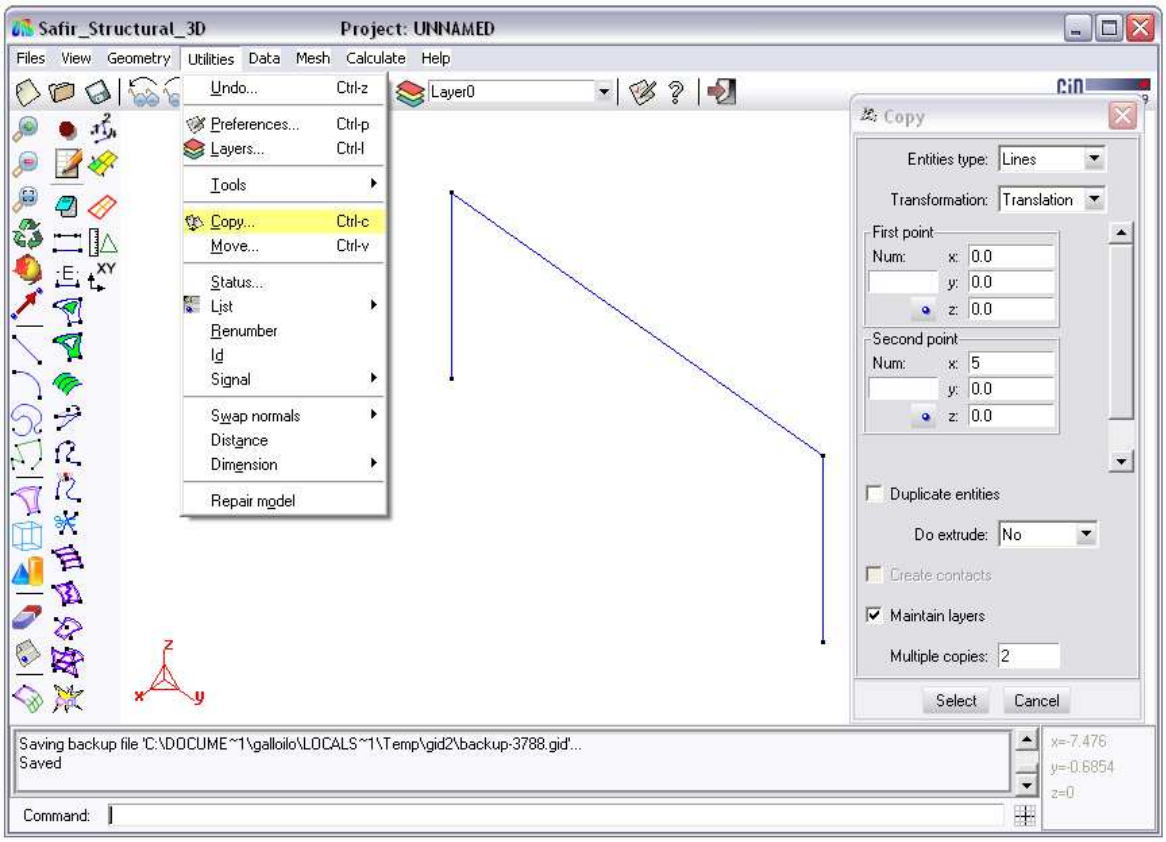
- ***Utilities->Copy*** ***or [Ctrl + c]***

For Entity type, use: ***Lines***

Enter for Second point: ***x = 5***

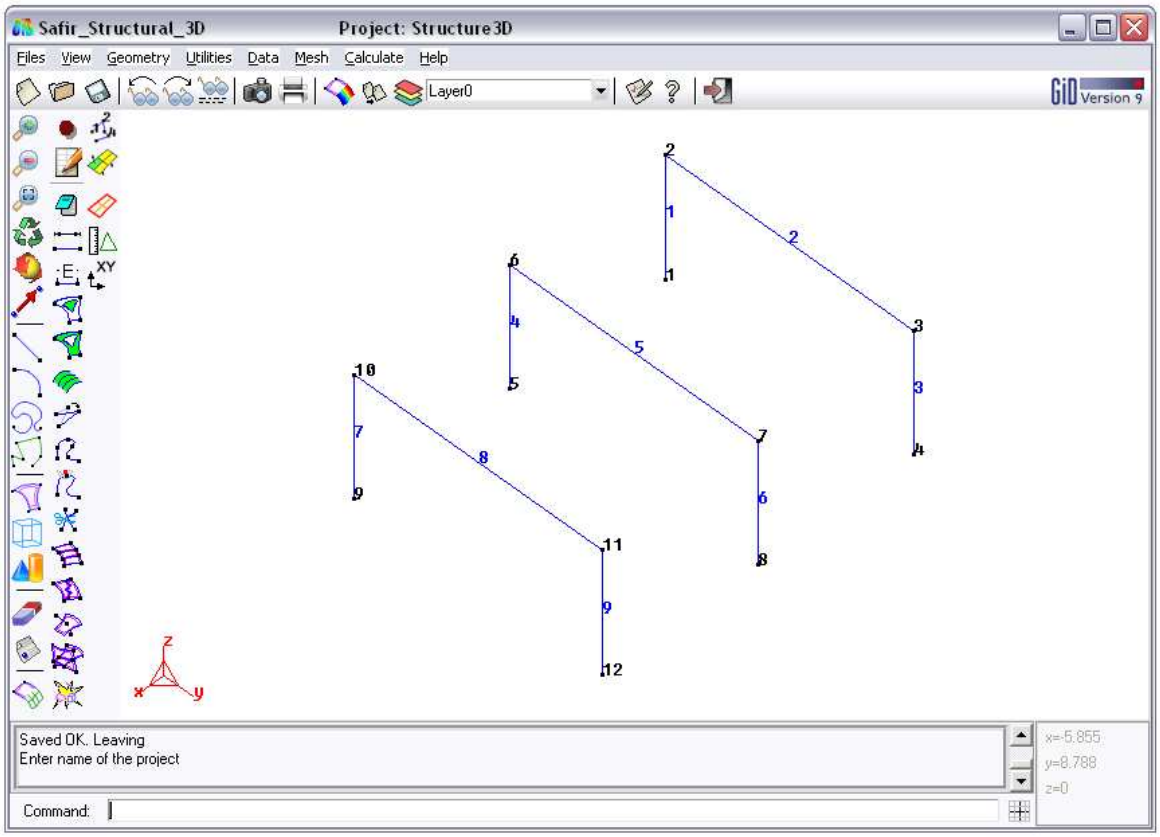
Enter for multiple copies: ***2***

Select all lines and press ***[Esc]***



To see nodes and beams numbers select:

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



To connect the endpoints of columns in x direction, select:

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

Press *[Ctrl-a]* and pick points 2, 6 and 10 and press *[Esc]*

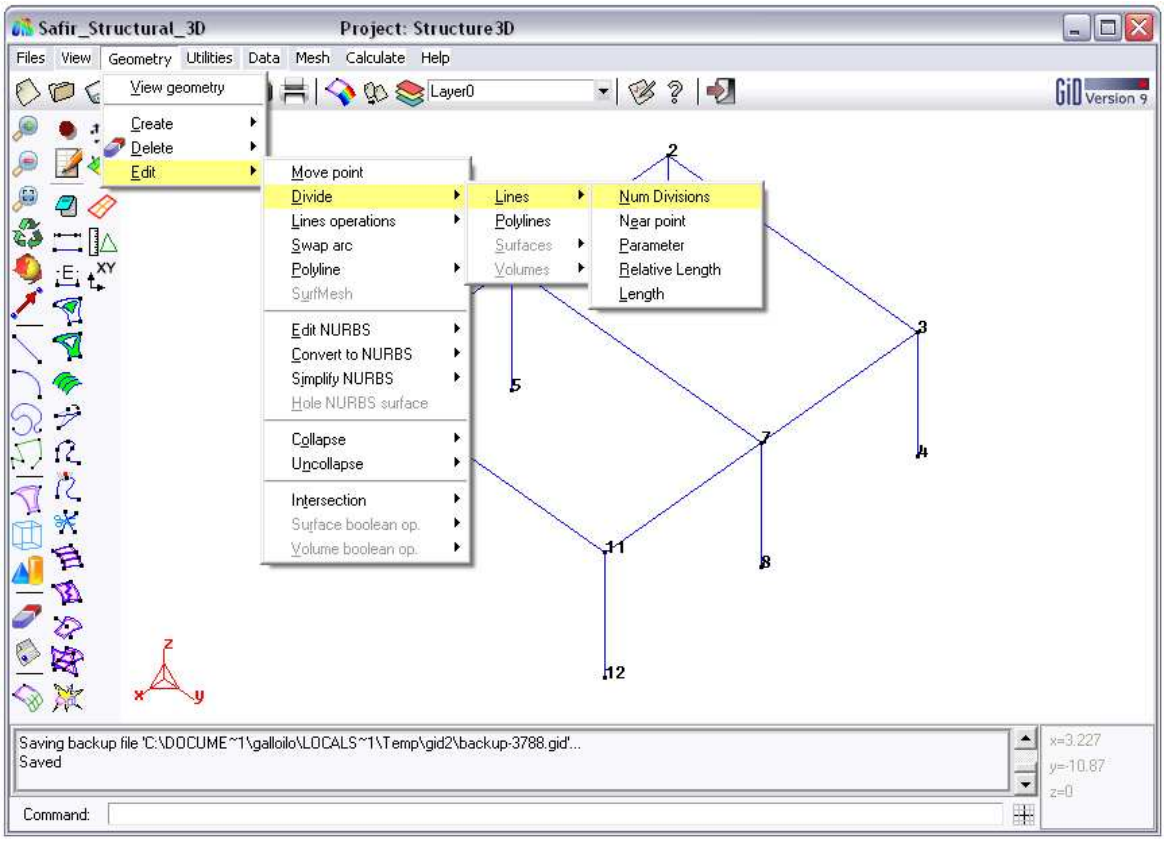
Then do the same operation with points 3, 7 and 11

Divide beams 2-3, 6-7, 10-11, 3-7, 7-11 in 2 pieces by using:

- ***Geometry->Edit->Divide->Lines->Num Divisions***

Put a number of divisions of 2

Select all beams you want to divide and press *[Esc]*



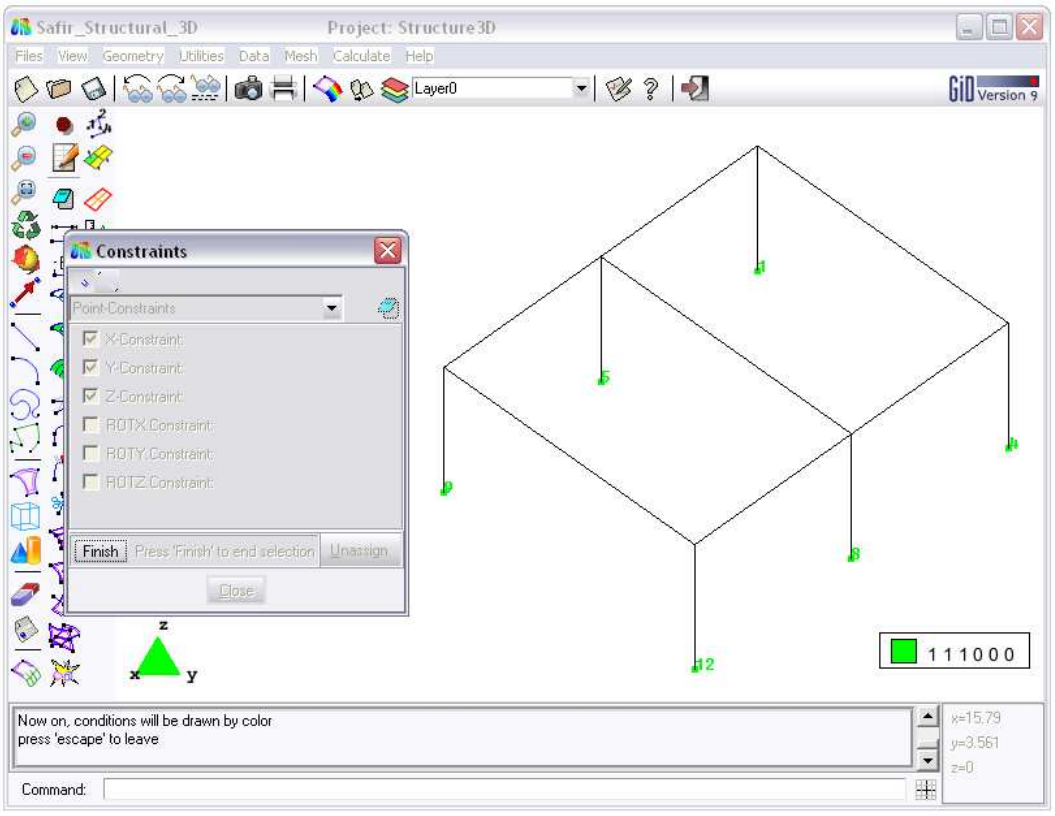
4. Define constraints for the supports:

From the pull down menu select

- ***Data->Constraints***

Select x,y and z constraints and assign them to the base points of all columns.

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

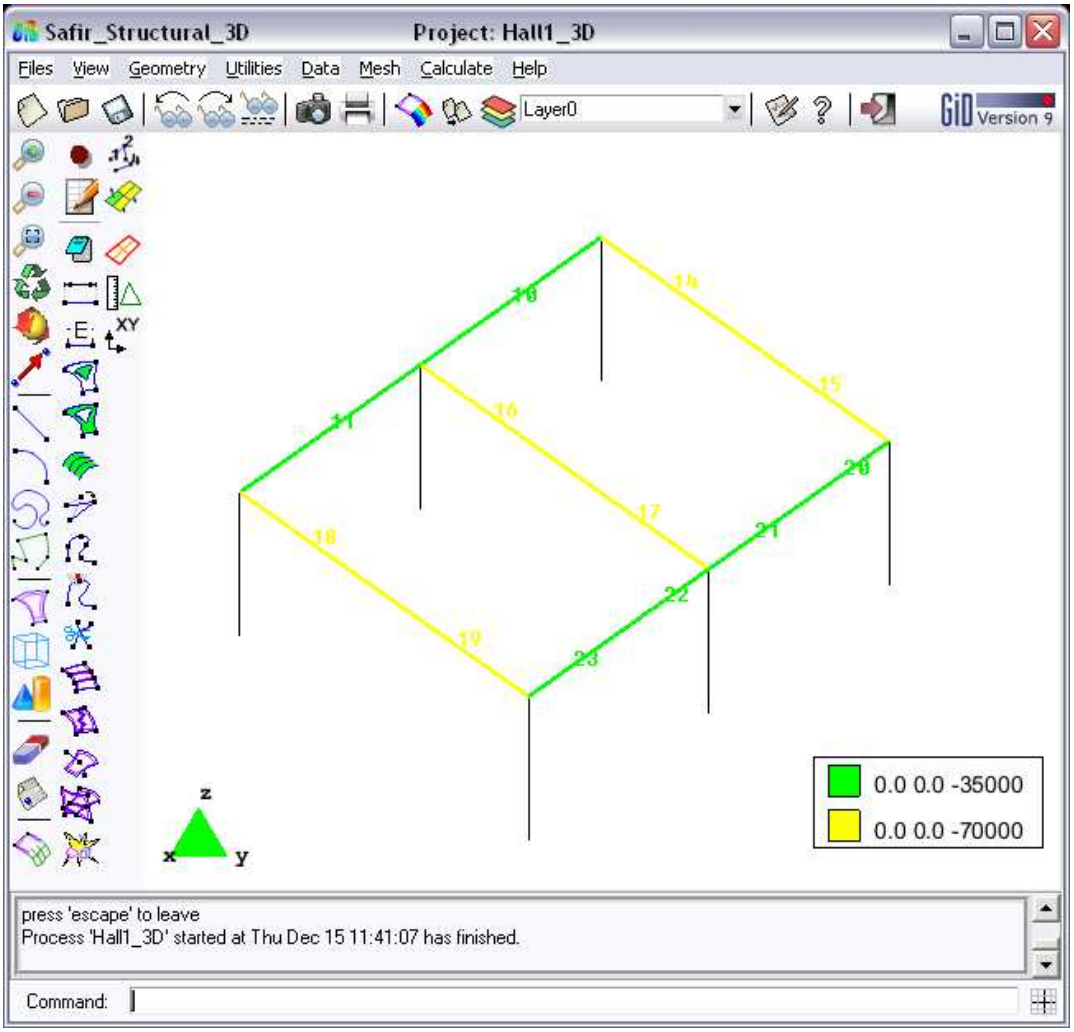


Press ***Finish*** or ***[Esc]*** to leave this view mode

5. Define loads

From the pull down menu select:

- *Data->Loads*



In the dial box select ***Beam-Load***  and enter a Z-Pressure of –70 000 N/m and ***Assign*** it to all beams in y-direction (beams from 14 to 19).

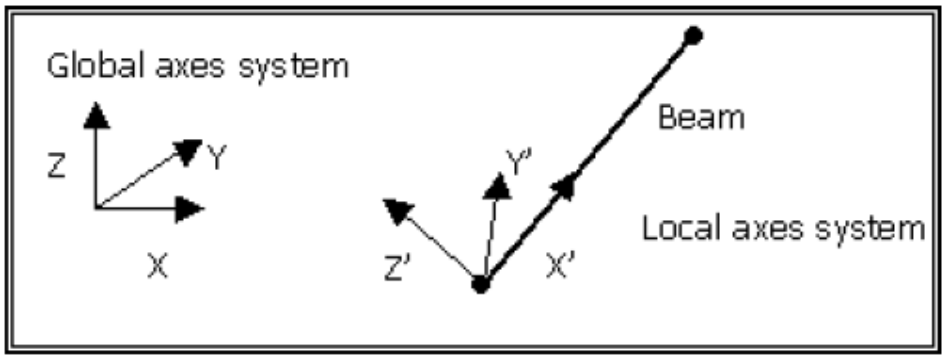
Enter a Z-Pressure of –35000 N and ***Assign*** it to all beams (10, 11, 20, 21, 22 and 23) in x-direction.

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

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

6. 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 3 named local axes (LAX, LAY, LAZ) with a X' direction and an angle as shown in the figure below.

From the pull down menu select:

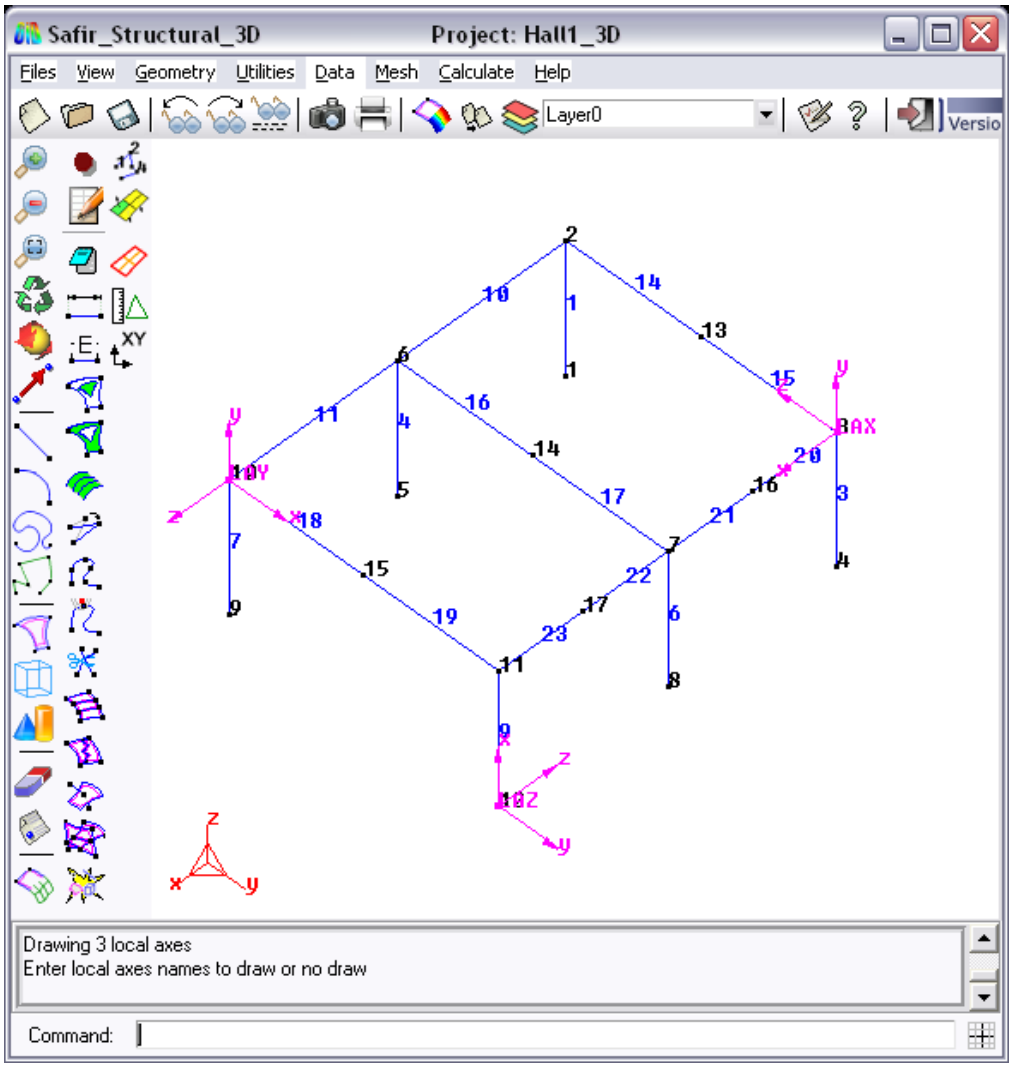
- ***Data->Local Axes->Define***

Enter the name of the new local axe ***LAY***
Select ***XandAngle***

Press ***[Ctrl+a]*** in order to select the local axe center then do the same operation to select a point in the positive x' axe of your local axe (in this case you have to select a point in Y direction).
Set an angle in order to get your y' direction (4th node)

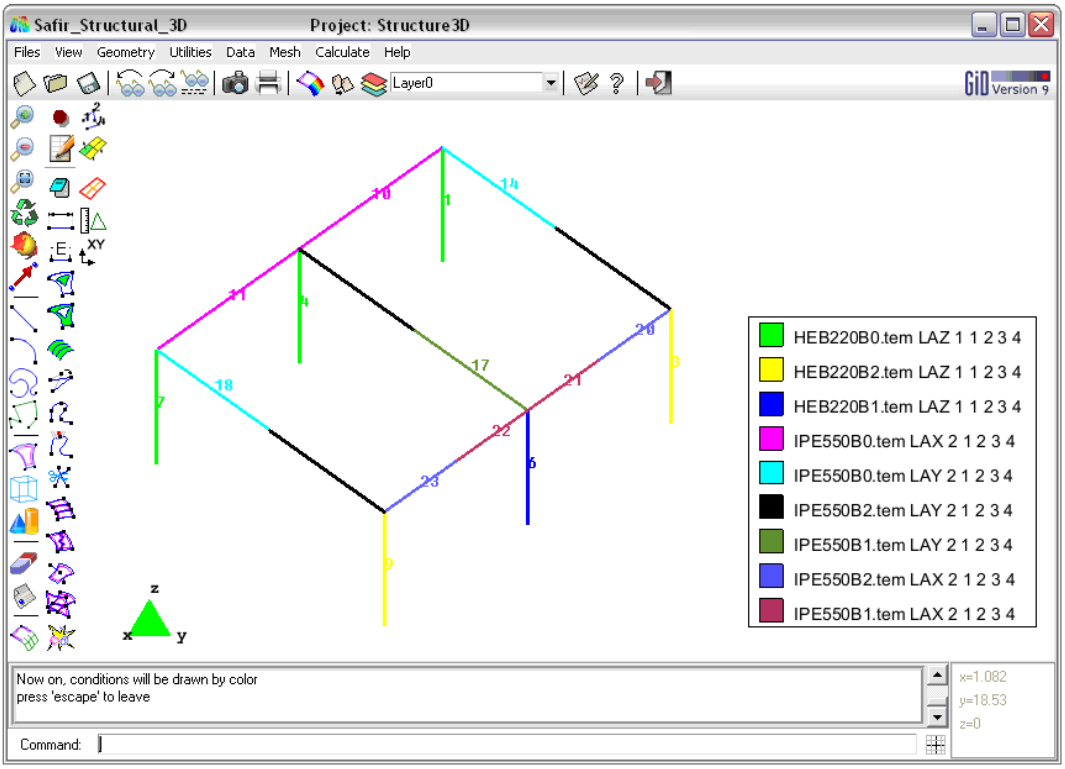
Do the same operations in order to create local axes LAX and LAZ. To draw local axes select:

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


Safir_Structural_3D
Project: Hall1_3D
Files View Geometry Utilities Data Mesh Calculate Help
Layer0
Drawing 3 local axes
Enter local axes names to draw or no draw
Command:

7. Assign temperature files (.TEM files)

The objective is to assign the .tem file named (IPE550B0; IPE550B1; IPE550B2; HEB220B0; HEB220B1; HEB220B2) to the system lines



From the pull down menu select:

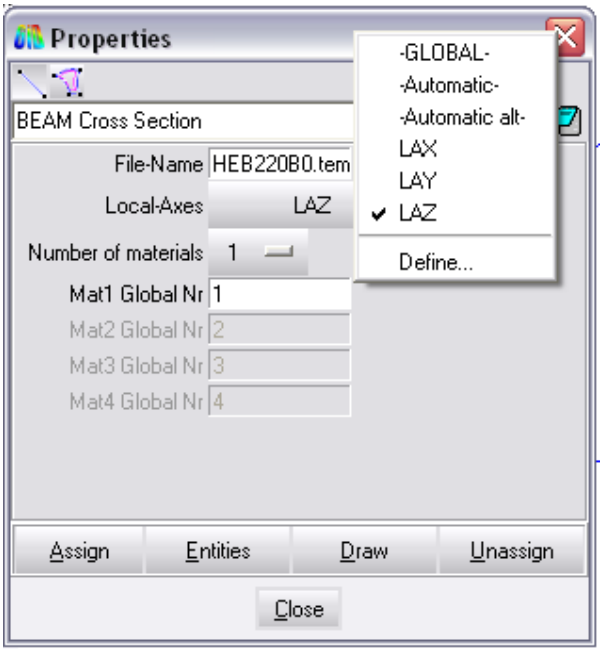
- ***Data->Properties***

In the dial box change the File-Name: ***safir.tem*** to the temperature file (.TEM file) of the cross-section, in this case ***HEB220B0.tem***.

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

In this case all cross-section have just one material

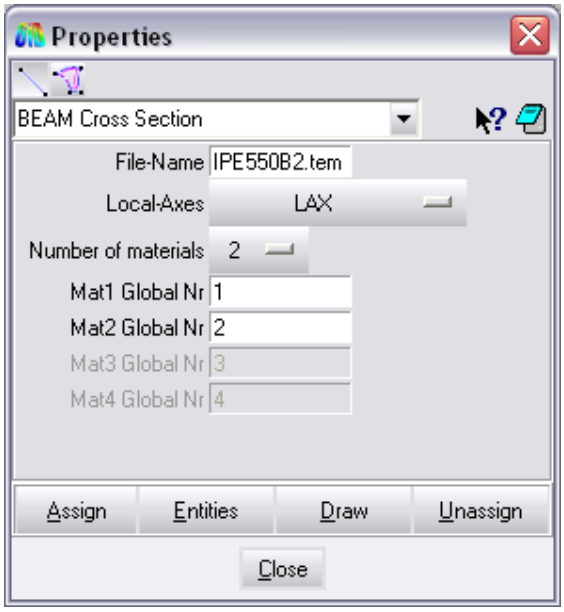
⚠ *Sometime, GiD use white color for element, that why you might not see one element type when you use the "Draw color" function*



Assign the ***HEB220B0.tem*** section to beams in HEB220B0 who have a local axe ***LAZ*** (in this case beams 1, 4 and 7).

Do the same operations in order to assign ***IPE550.tem*** file to other systems lines.

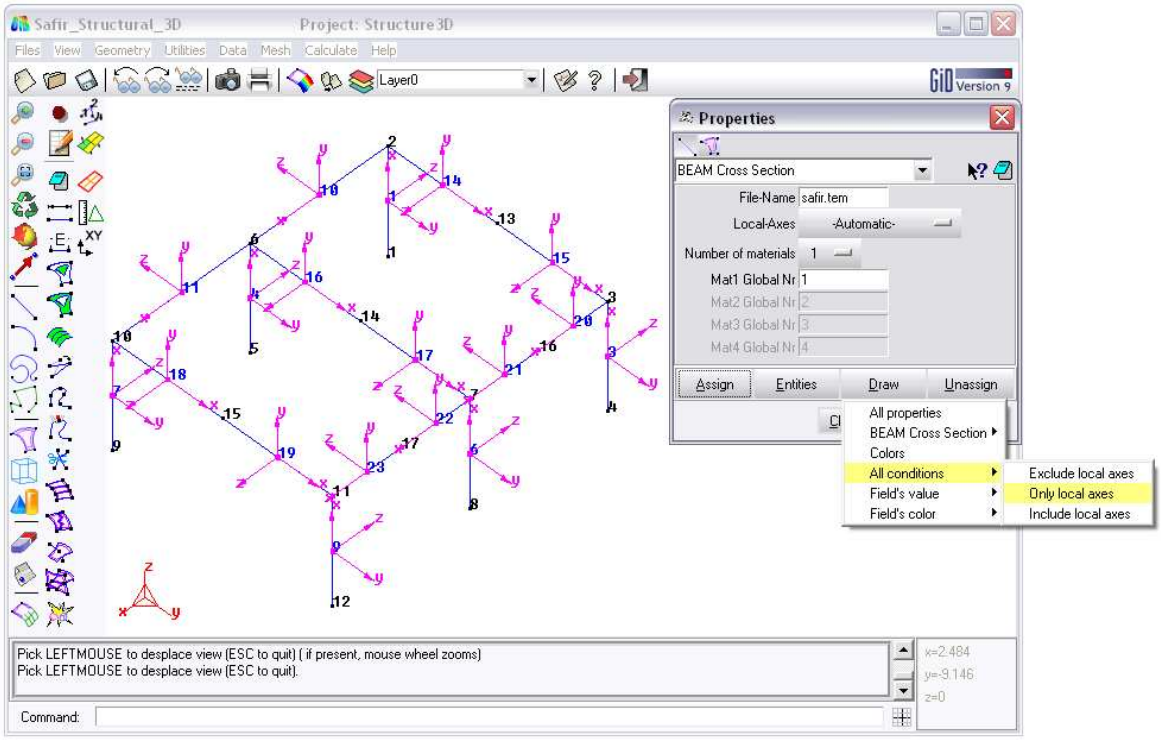
⚠ *Don't forget to change the local axes and the number of materials (IPE550.tem have 2 material)*



To display Property select in the dialog box:

Draw->Colors

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



To display local axes select in the property dialog box:

Draw->All conditions->Only local axes

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

Now you have to open the GiD-Safir tutorial file and the Hall1_3D file

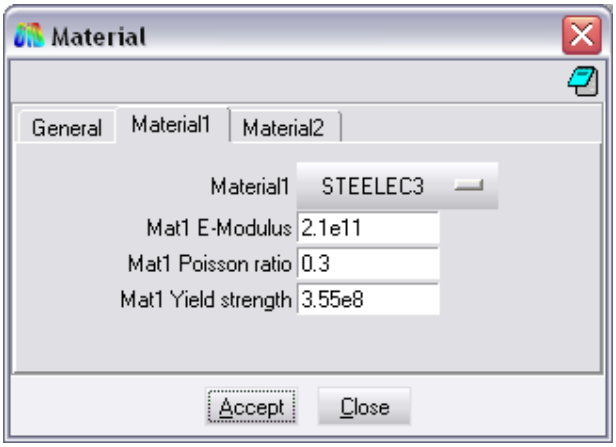
Select the 6 .tem files and copy them into the Hall1_3D.gid file you created

8. Define global materials:

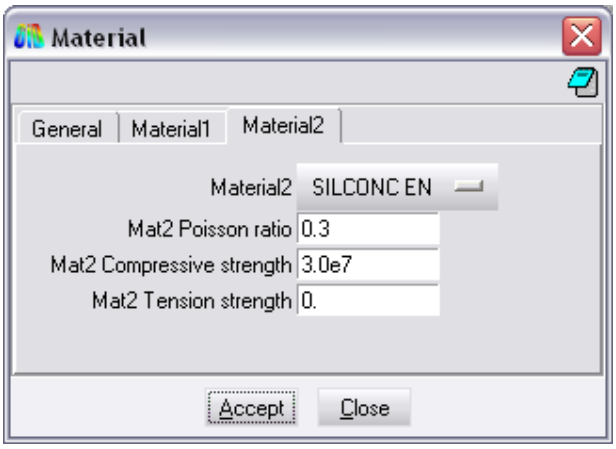
To define materials select from the pull down menu:

- ***Data->Material***

In the general tab, put 2 materials. In the material1 tab, fill as shown below



In the material2 tab, fill as shown below:

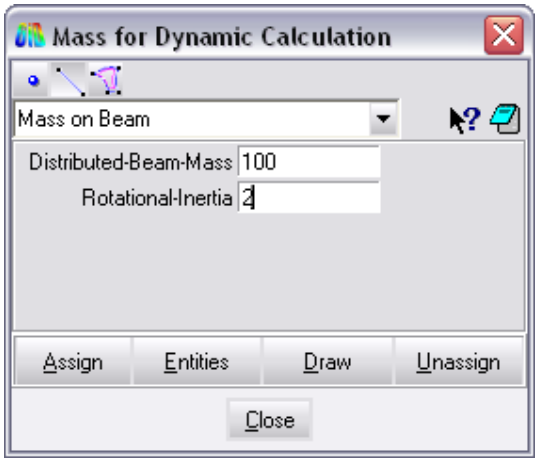


9. Define the Mass

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

- ***Data-> Mass***

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



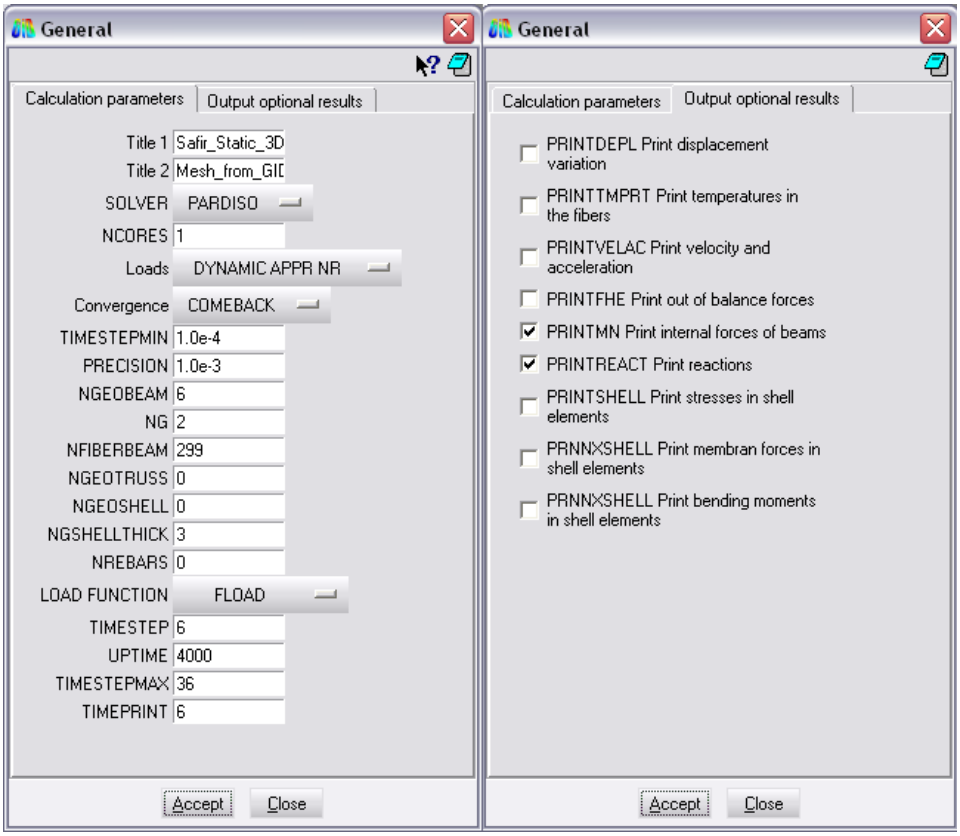
Assign the mass to all the elements

10. Define general data

Select from the pull down menu:

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

And fill as shown below



Enter the following

NGEOBEAM = 6 (the number of .tem files)

NFIBERBEAM = 299 (max. number of fibers)

TIMSTEP, UPTIME, TIMEPRINT as needed

In the Output optional results tab, you can change what GiD-Safir will print during the calculation

⚠ *Ngeobeam is the number of .TEM files (6 in this case)*

Nfiberbeam is equal to 299 in this case, the only way to find 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 out 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 Timeprint 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 thermal 2D calculation*

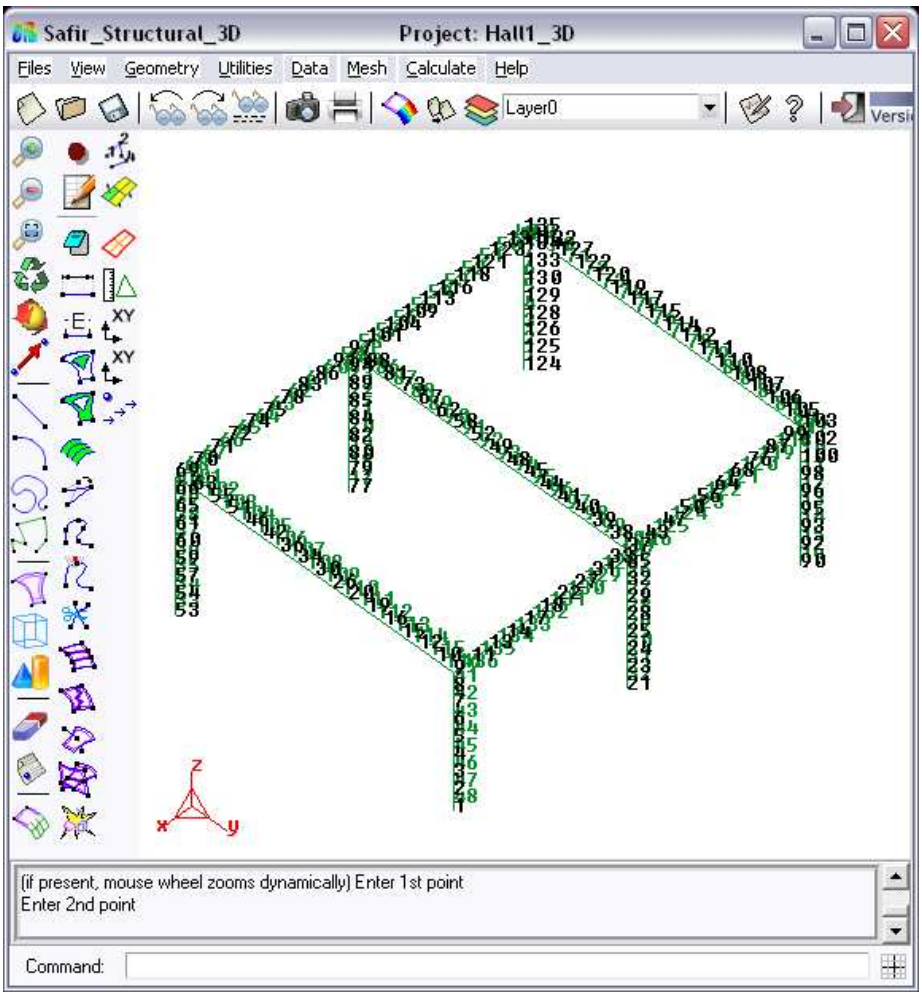
Click on ***Accept*** to save your modification

11. Generate the mesh:

Select from the pull down menu:

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

Enter the element size of 0.5 m



To display elements and nodes numbers select from the pull down menu:

- ***View->Label->Al***

If you want to zoom in use select:

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

And zoom on the area you want to check

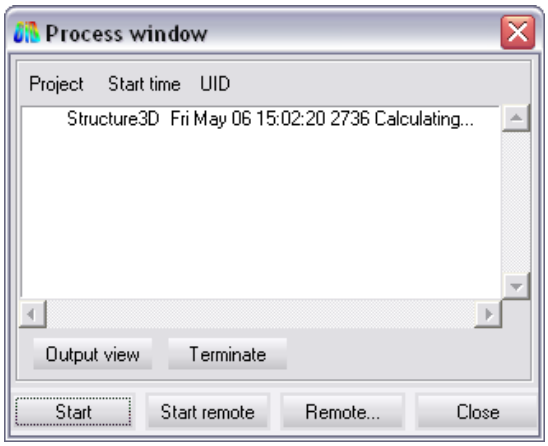
To quit the label mode use

- ***View-> Lapel-> Off***

12. Start the calculation:

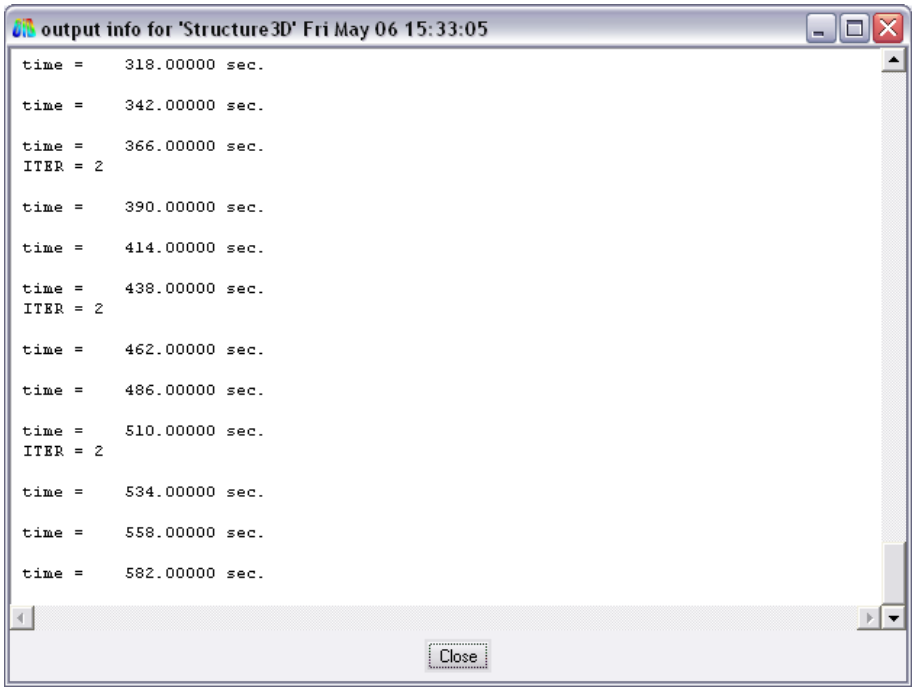
Select from the pull down menu:

- *Calculate->Calculate window*



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

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



Select from the pull down menu:

- *Calculate->Calculate window*

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 when you forgot to copy all .TEM files into the project directory, or if you entered a wrong number for NGEOBEM or NFIBERBEAM.*

⚠ *Post processing can be done with Diamond2011. The .OUT file is located in the project-name.gid directory . The file name is project-name.out*
For post processing with GiD select from the pull down menu:
Files->Postprocess or click the Postprocessor Icon in the tool box.