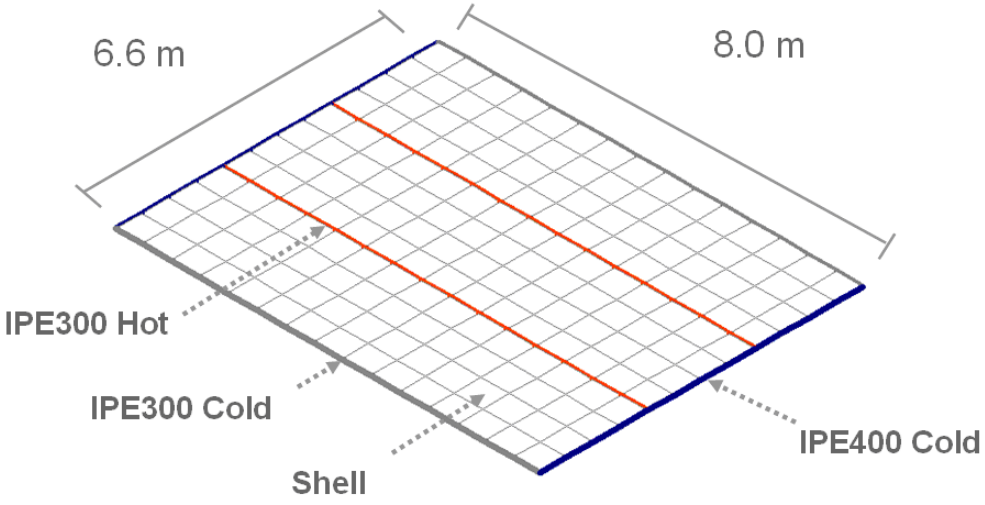
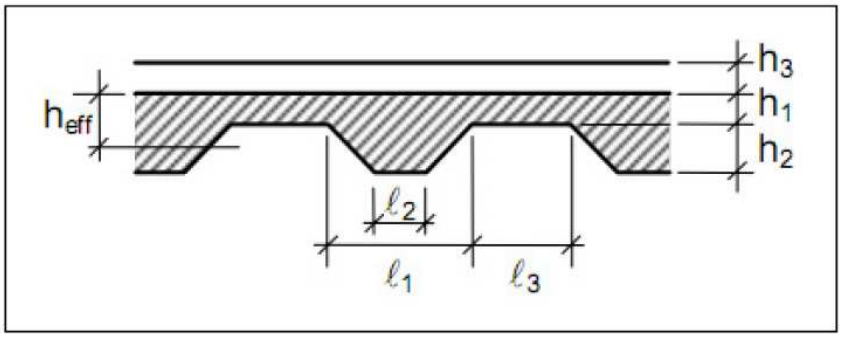
Example for GID-SAFIR 2D and 3DThermal Analysis

Exercise n°11 – Cofraplus 60

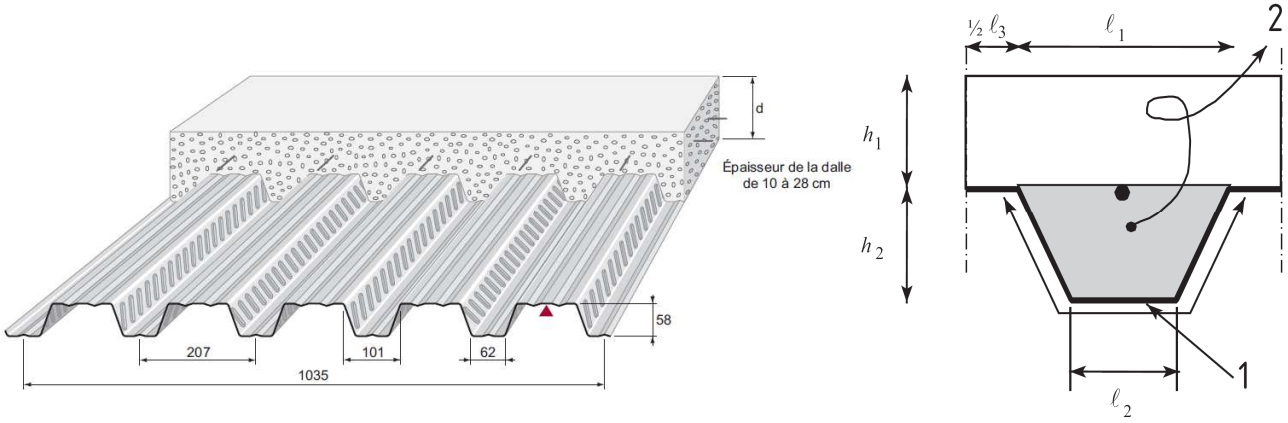


A. Create .TSH Cofraplus 60 section

In this example we are going to design a slab with Cofraplus 60 with concrete C 30/37 of 62mm high and smooth mesh with a diameter equal to 6 mm 150/150 localized 31 mm from the top of the slab for a slab total thickness of 120 mm.



If we refer to the EN 1994-1-2 §D.1:



For thermal analysis, the effective thickness of a composite slab h_{eff} is given by the formula:

$$h_{eff} = h_1 + 0.5 \cdot h_2 \left(\frac{l_1 + l_2}{l_1 + l_3} \right) \qquad for\ h_2 / h_1 \le 1.5\ and\ h_1 > 40\ mm$$

$$h_{eff} = h_1 \cdot \left(1 + 0.75 \left(\frac{l_1 + l_2}{l_1 + l_3} \right) \right) \qquad for\ h_2 / h_1 > 1.5\ and\ h_1 > 40\ mm$$

$$h_{eff} = h_1 \qquad for\ l_3 > 2 \cdot l_1$$

For a Cofraplus 60:

l_1 = 101 mm; l_2 = 62 mm; l_3= 106 mm; h_1 = 62 mm and h_2 = 58 mm (d = 120 mm)

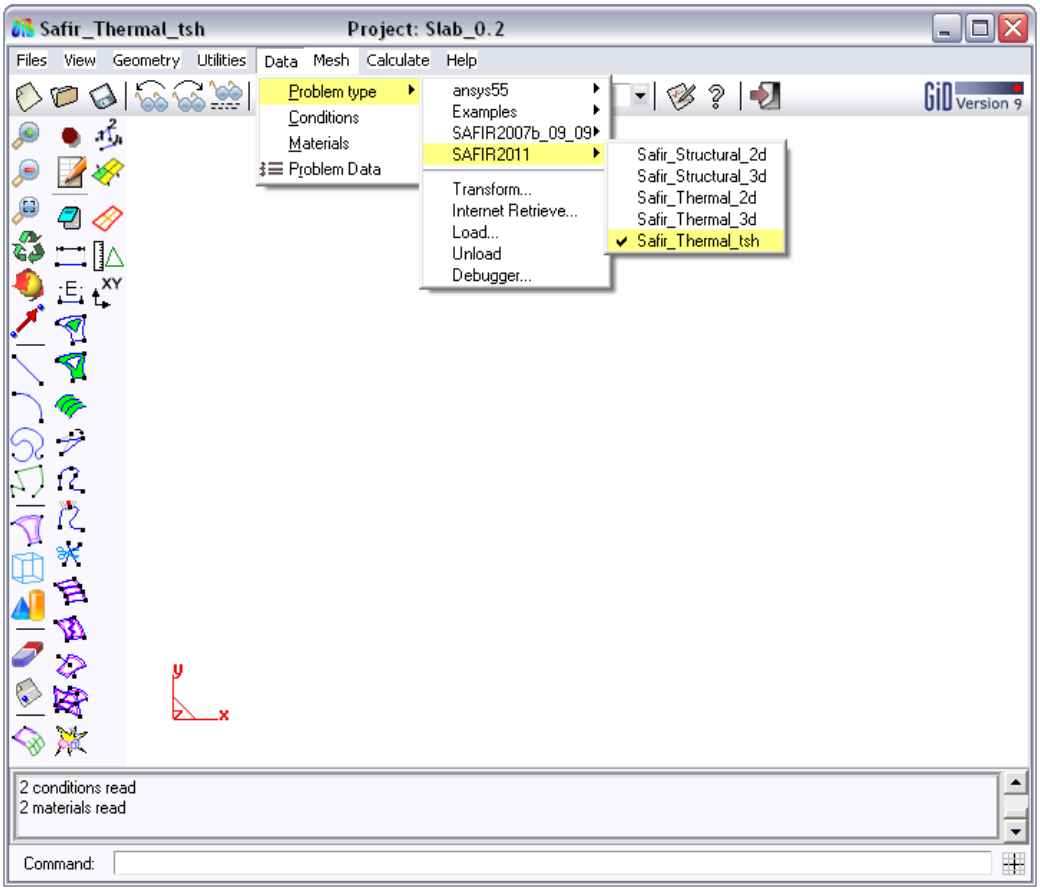
We obtain h_{eff} = 84.8 mm

Now we are going to create a .TSH section with a shell thickness of 0.0848 mm of concrete

1. Create a project in 2D for TSH Thermal Analysis

From the pull down menu select:

- *Data->Problem type->SAFIR2011->Safir_Thermal_TSH*



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.: Cofra60

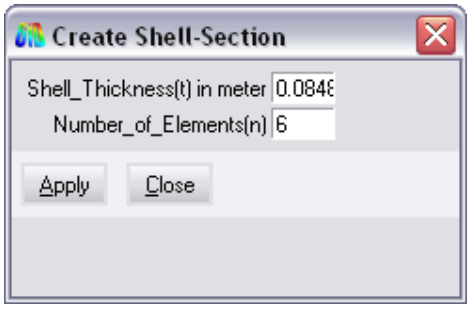
GiD creates a directory with the name Cofra60

GiD creates a number of system files in this directory.

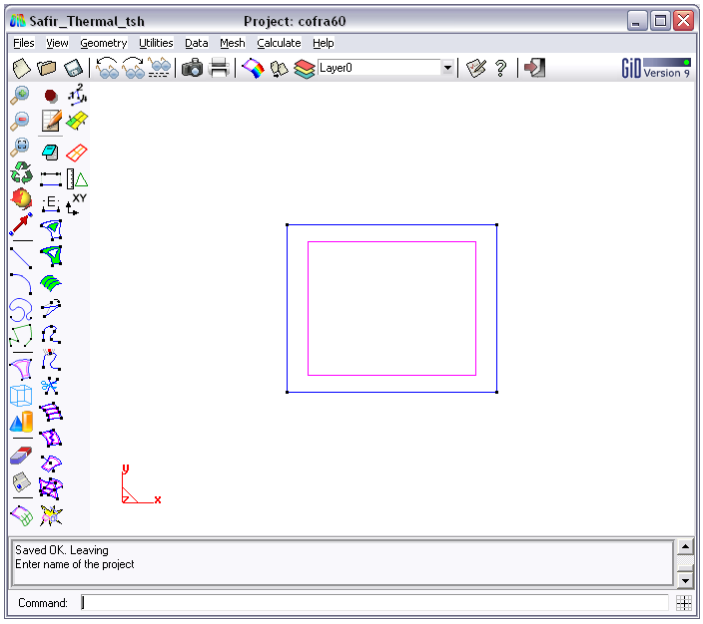
When you start the SAFIR calculation the Safir . IN, .OUT and .TEM files will be created in this directory.

2. Create the geometry in the xy-plane

GiD will open automatically a new window.
Fill as below:



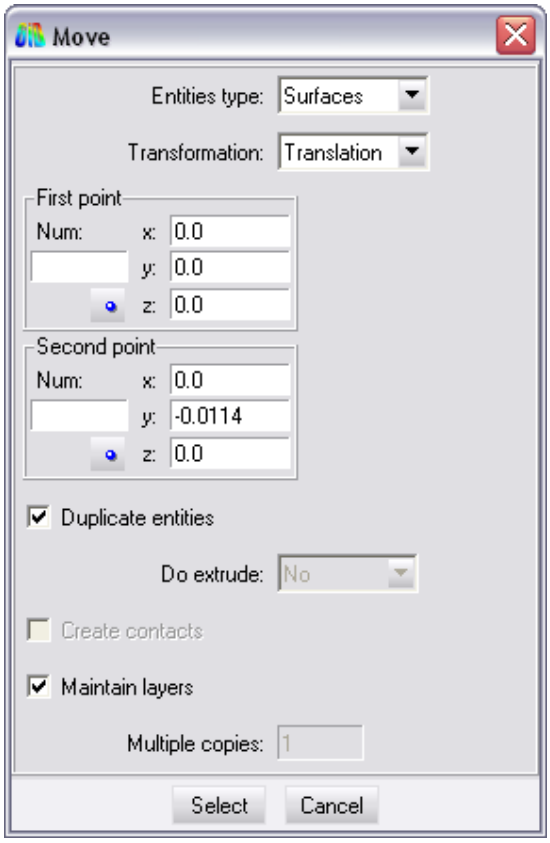
Click on ***Apply***



⚠ *It is very important to center all the element*

In order to center correctly this shell element, we have to move it on the y direction:

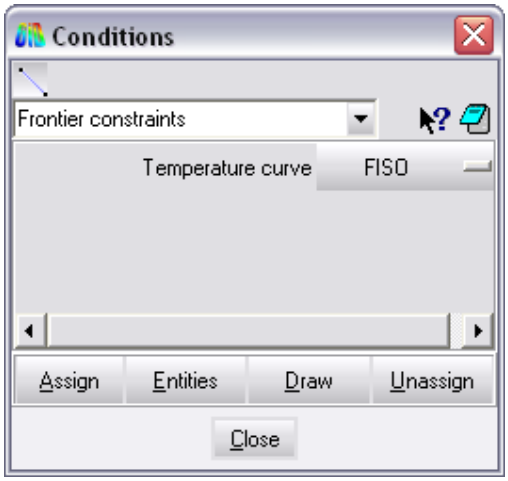
Press [Ctrl + v] and fill as below:



3. Assign a temperature curve

From the pull down menu select:

- ***Data->Conditions***

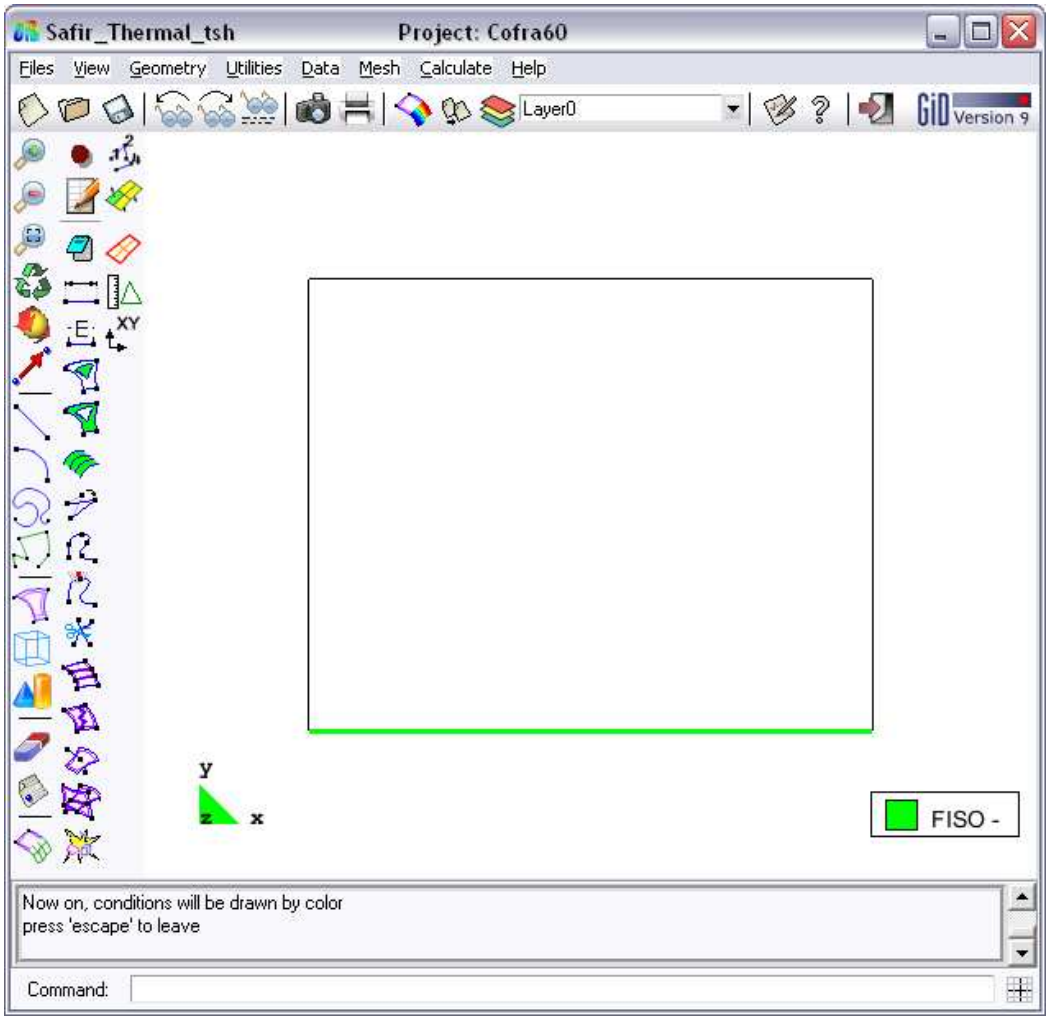


Select:

On the first pull down list: ***Frontier constraints***

On the Temperature curve pull down list ***FISO***

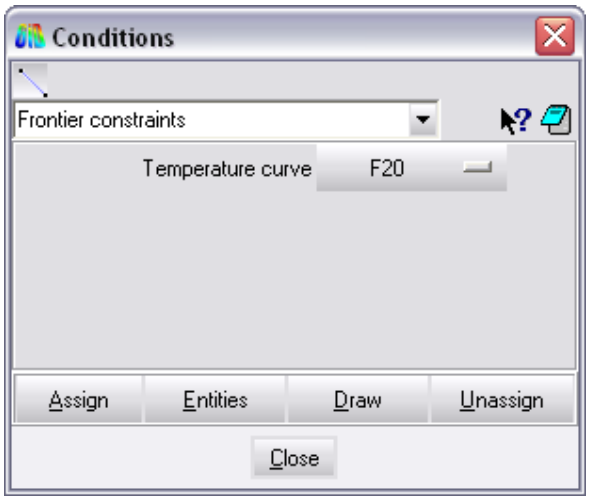
Click on the ***Assign*** button and assign it to the lower line as shown below



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

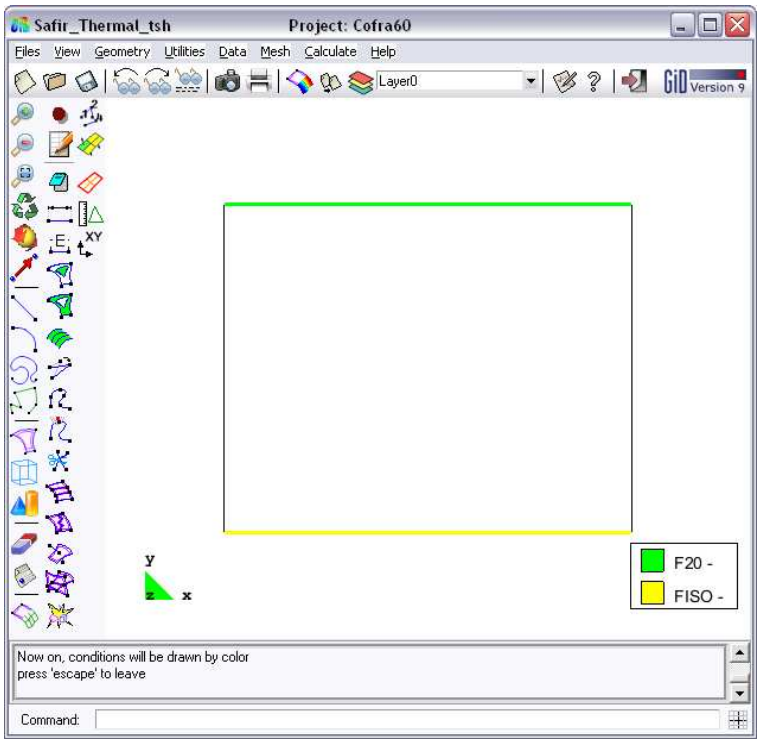
Select ***DRAW->Colors*** in the Conditions dialog box to display the frontier constraints

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



On the Temperature curve pull down list ***F20***

Click on the ***Assign*** button and assign it the upper line as shown below

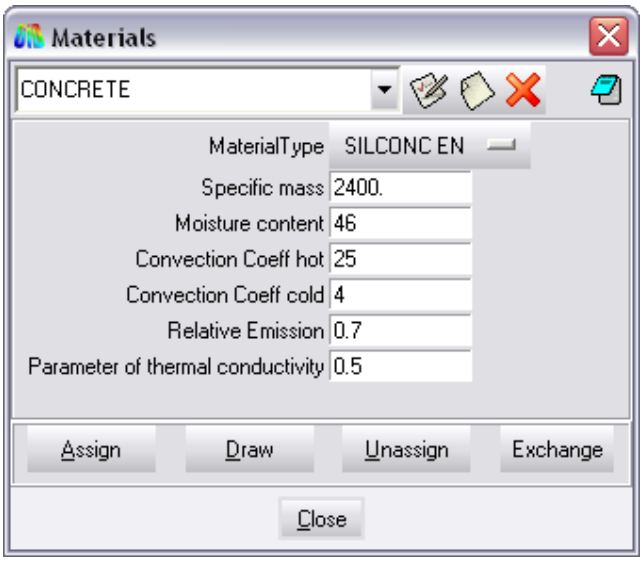


4. Assign Material

From the pull down menu select:

- ***Data->Materials***

Select ***CONCRETE*** from the dialog box pull down list



Then select:

SILICONC EN as Material Type

A specific mass of ***2400***

A moisture content of ***46***

A Convection Coeff hot of ***25***

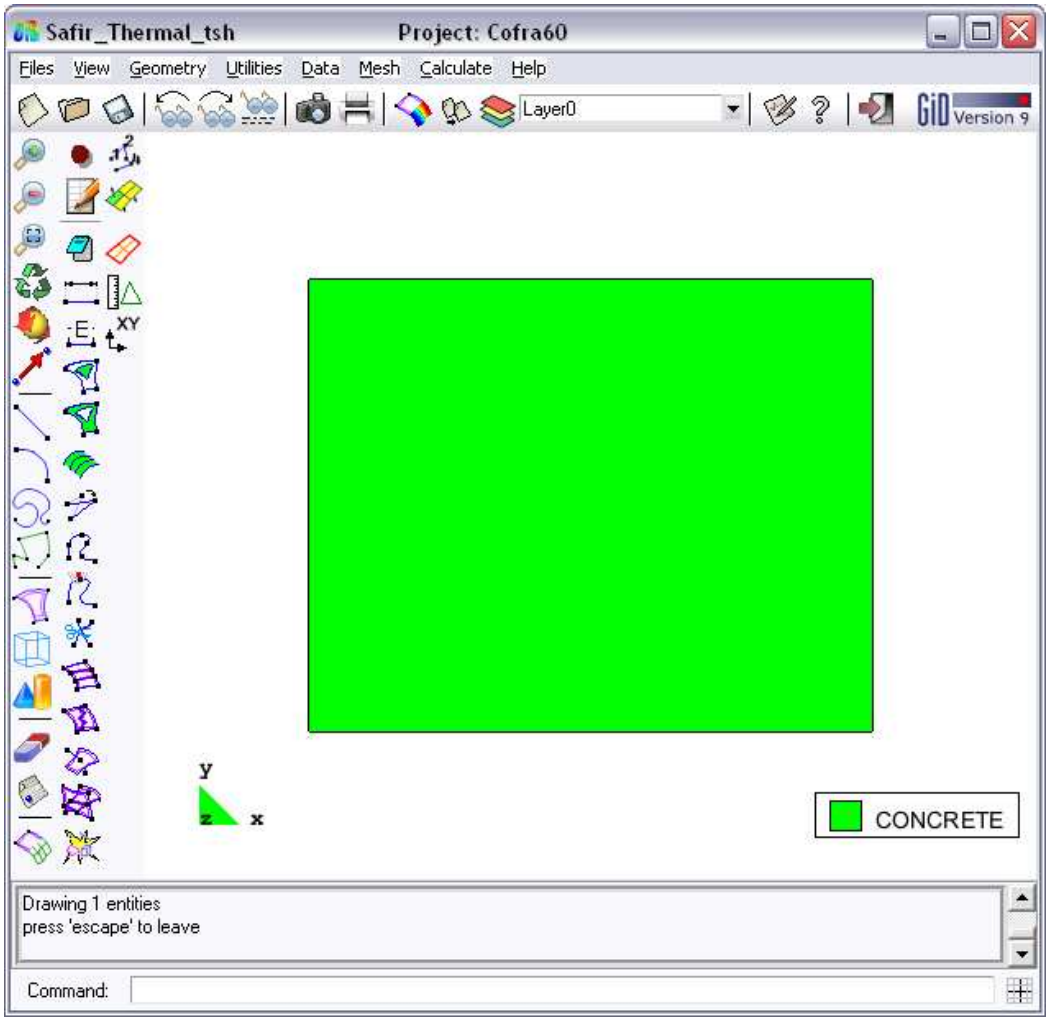
A Convection Coeff cold of ***4***

A Relative Emission of ***0.7***

A Parameter of thermal conductivity of ***0.5***

Click on ***Assign-> Surfaces*** and assign it to the surface

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



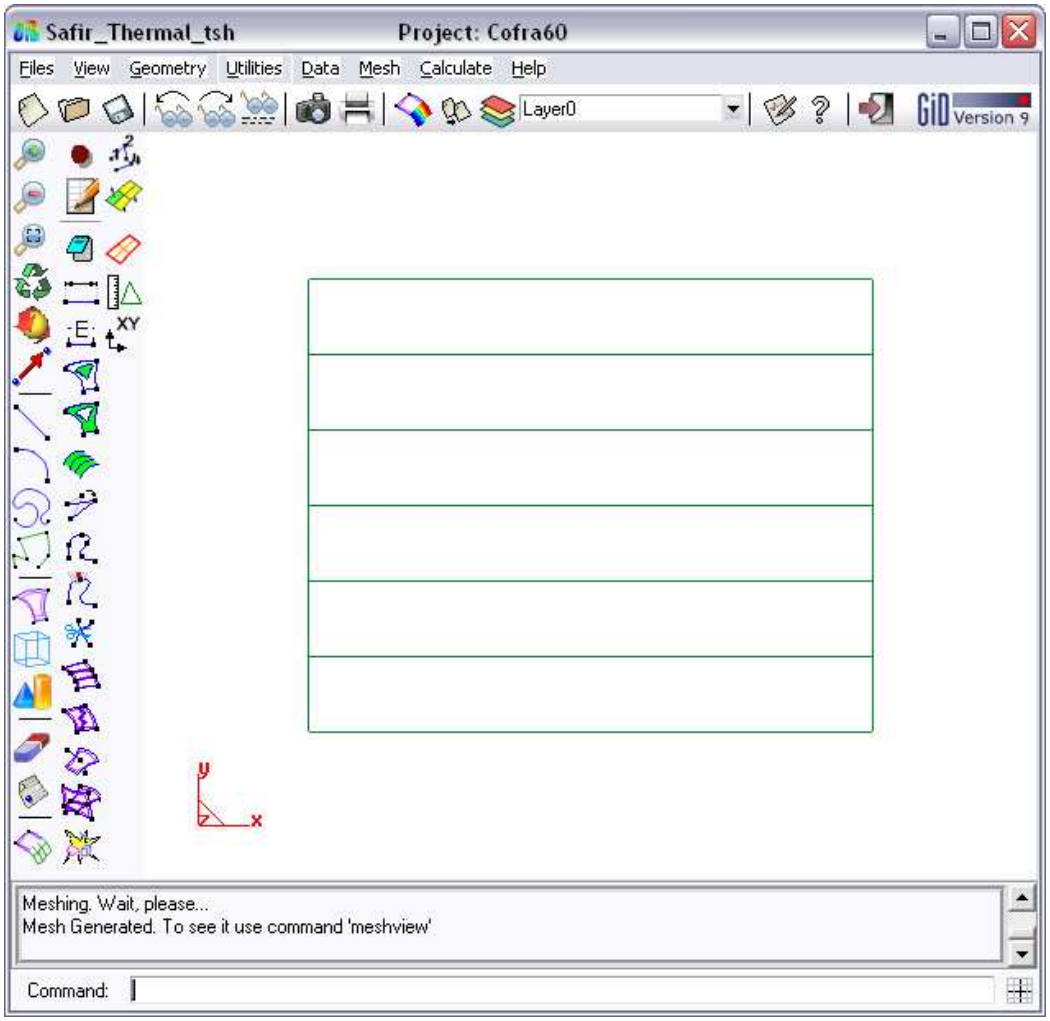
Select ***DRAW->all materials*** in the Material dialog box to display Materials

Press ***[Esc]*** or ***Finish*** to leave

5. Meshes

To create meshes select from the pull down menu:

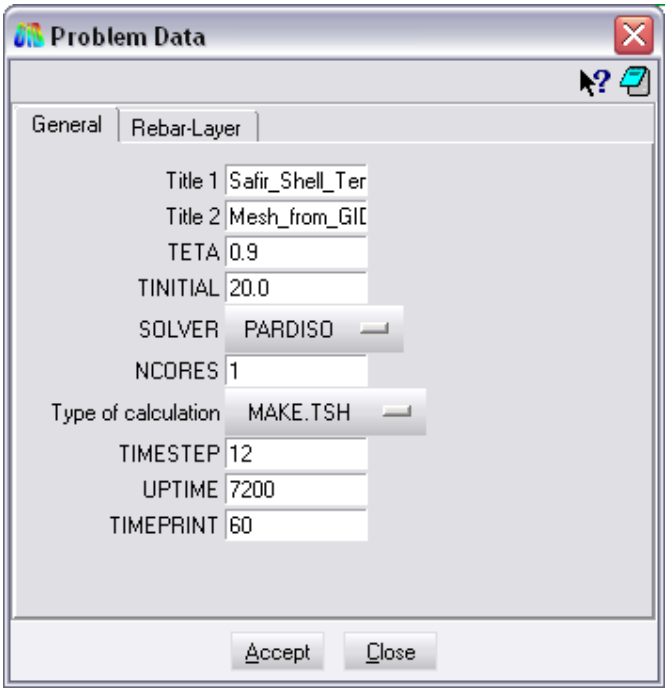
- ***Mesh->Generate mesh*** ***or use [Ctrl + g]***



6. Assign General Data

From the pull down menu select:

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



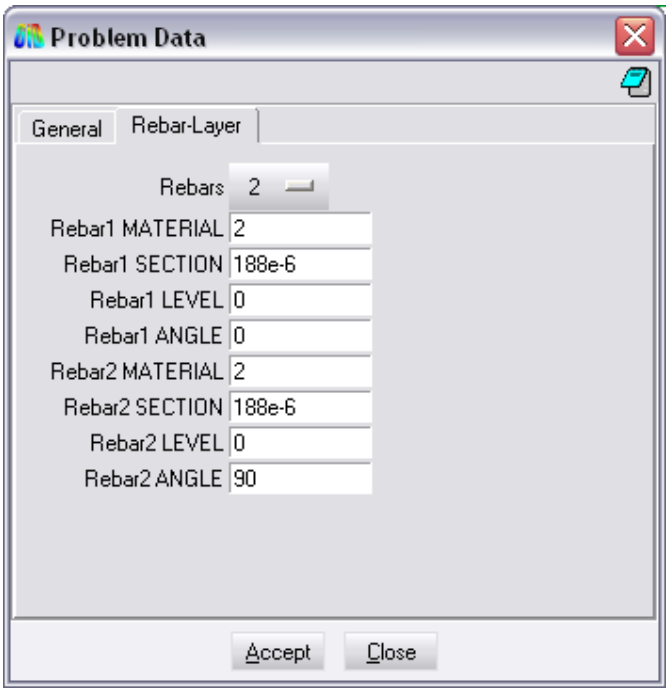
In the Problem Data dialog mask enter:

TIMESTEP, UPTIME, TIMEPRINT as needed

Click on the ***Accept*** data button

⚠ *When you click with the right button on one of the variables, GiD will display an online help message. The variables are also described in more detail in the SAFIR reference manual.*

Click on the ***Rebar-Layer*** tab and fill as shown below:



⚠ *Rebars is is the number of rebar layers in the section type*

⚠ *MATERIAL is the local number of this layer, in this case it is equal to 2. You will have to define the rebar material type for the mechanical calculation.*

⚠ *SECTION is the cross sectional area of rebar1 in this layer in [m^2/m]. In this case it represent a 150x150 gird with a rebar diameter of 6 mm:*

$$Section = \frac{\pi * 0.006^2 / 4}{0.150} = 188.5e-6\ m^2 / m$$

⚠ *LEVEL is the position of this layer in y axe with respect to the thickness (GiD-Safir always create new TSH section centered in Y)*

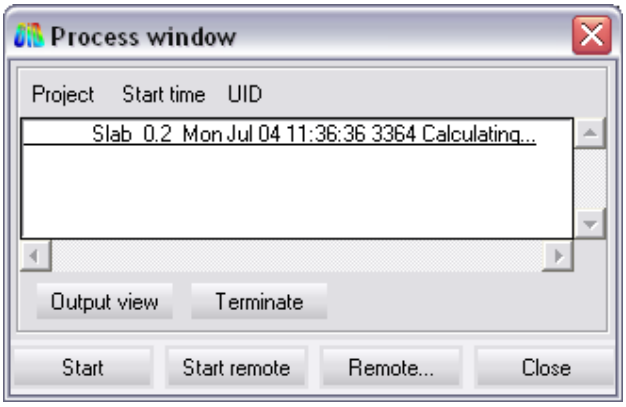
⚠ *Angle is the angle in degrees between the local axis x and the layer or rebars*

7. Start the calculation

From the pull down menu select:

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

Click the ***Start*** 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.

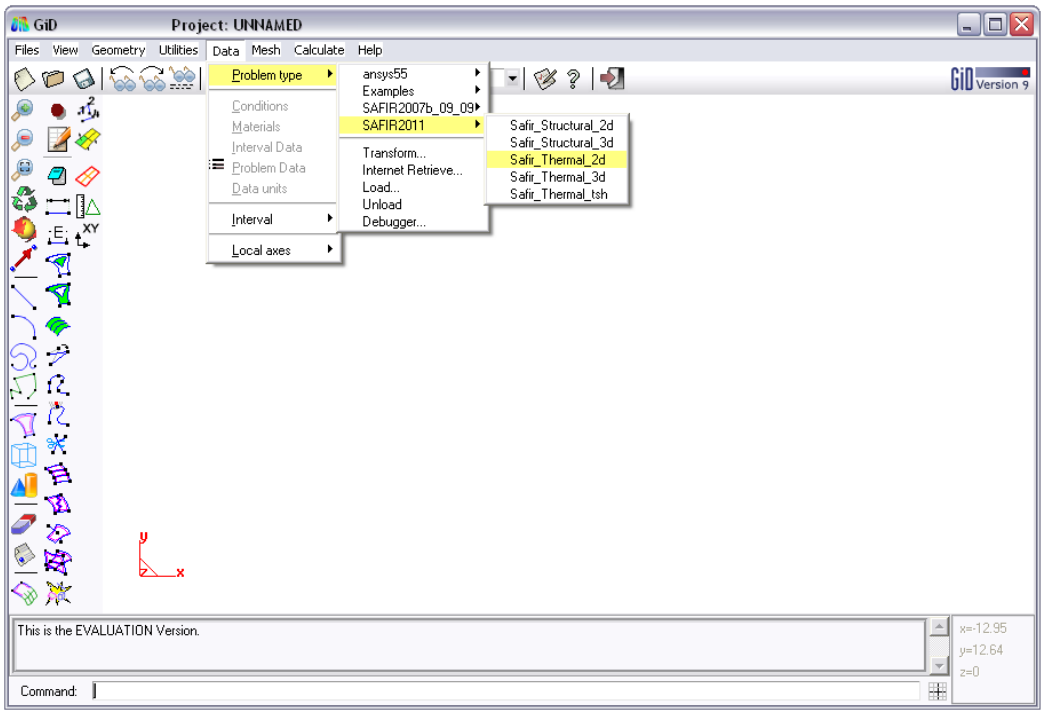
B. Create .TEM sections

The objective is now to create all .TEM files: one IPE300 cold section, one IPE300 hot section and one IPE400 cold section. All those sections have to be realizing as explained in exercise 3 with a slab 120 mm wide.

1. Create a project in 2D for Thermal Analysis

From the pull down menu select:

- *Data->Problem type->SAFIR2011->Safir_Thermal_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.: IPE300_H

GiD creates a directory with the name IPE300_H.gid

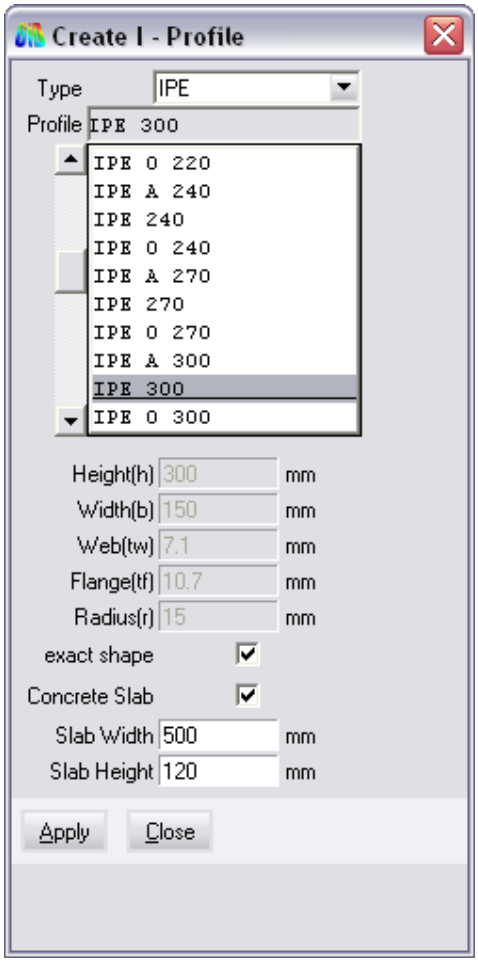
GiD creates a number of system files in this directory.

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:

- ***Cross-Section->I-Profile***



Select ***IPE*** as type, ***IPE 300*** as Profile, tick ***exact shape***, put ***500*** mm as slab width and a slab height of ***120*** mm

Click on ***Apply***

- *GiD-Safir will create an IPE300 profile. The center of this profile will be automatically centered on the 0,0 point of the xy-plan*

3. Assign a temperature curve

From the pull down menu select:

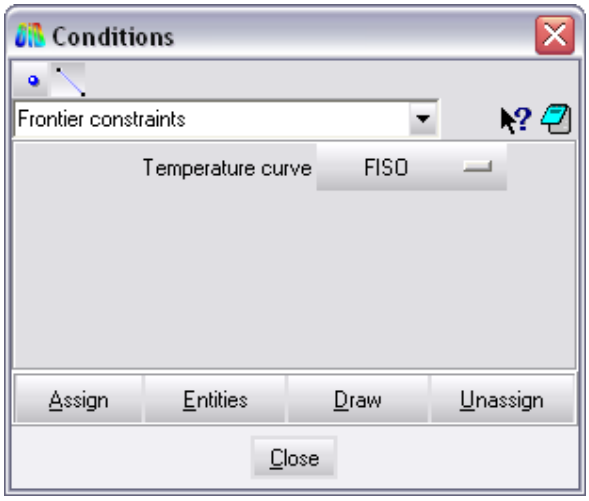
- ***Data->Conditions***

Select:

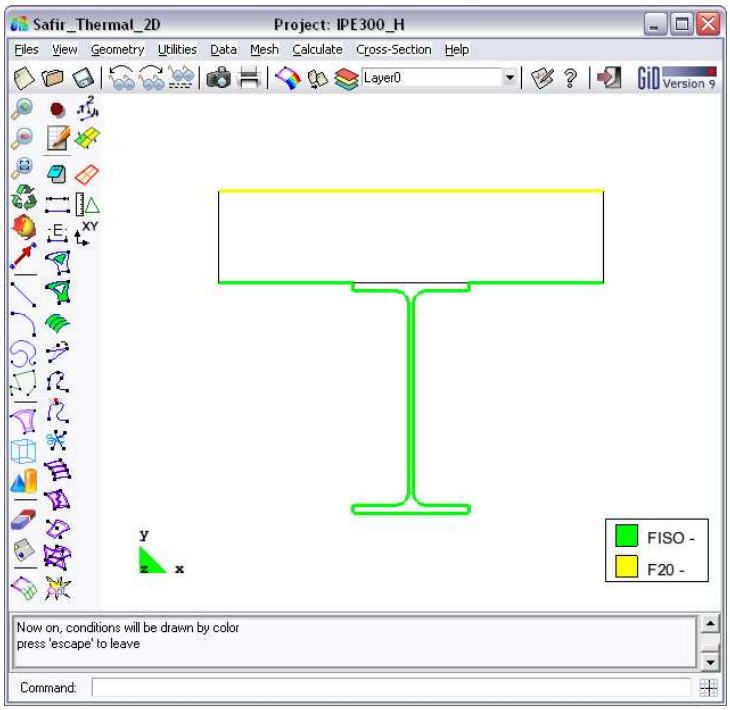
The  button

On the first pull down list: ***Frontier constraints***

On the Temperature curve pull down list ***FISO***



Click on the ***Assign*** button and assign it to profile and slab lines as shown below:

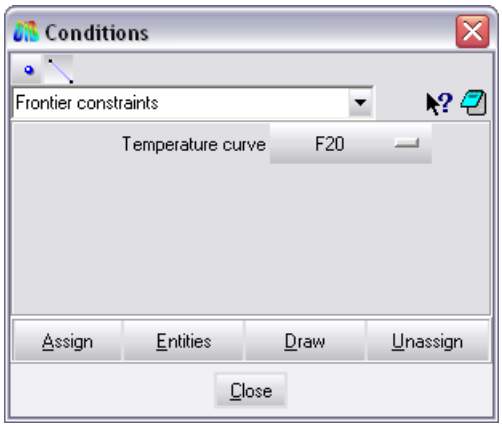


Press *[Esc]* or click on *Finish* to confirm

Select *DRAW->Colors* in the Conditions dialog box to display the frontier constraints

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

Then select *F20* as temperature curve



And assign it to the upper line of the slab, as shown previously

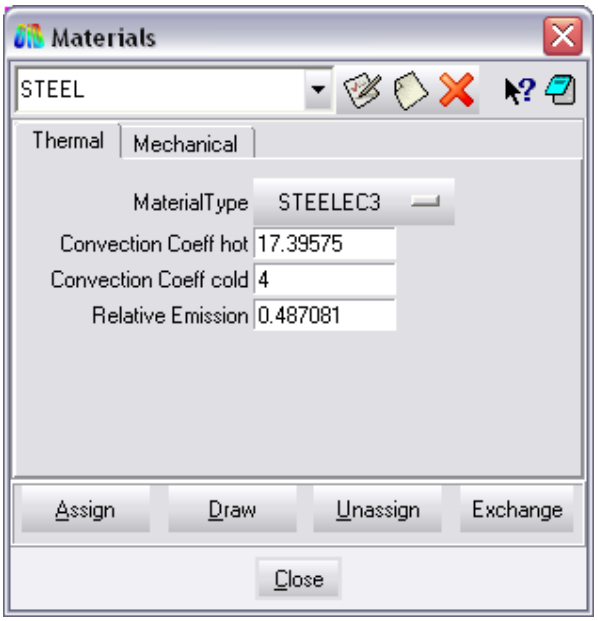
4. Assign Material

From the pull down menu select:

- ***Data->Materials***

Select ***STEEL*** from the dialog box pull down list

Then select fill as shown below:



As notified in the Eurocode 1993-1-2, formula (4.26a), for I-sections under nominal fire actions, the correction factor for the shadow effect may be determined from:

$$k_{sh} = 0.9 \times \frac{[A_m / V]_b}{[A_m / V]}$$

$[A_m/V]_b$ is the box value of the section factor [1/m]

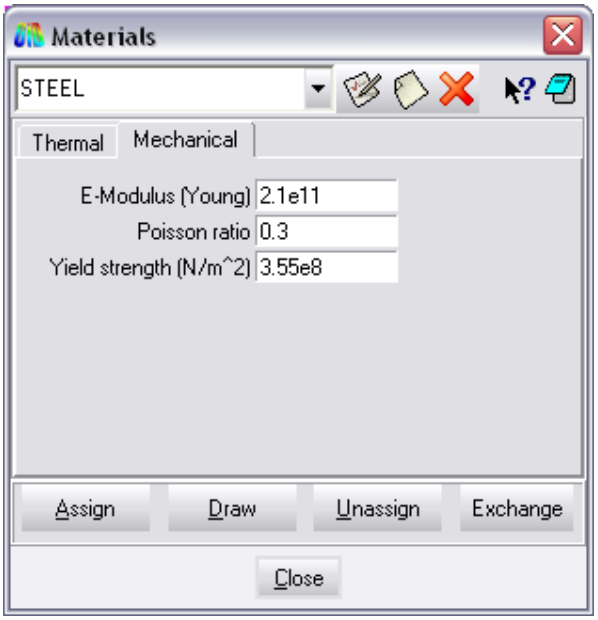
A_m/V is the section factor for unprotected steel members [1/m]

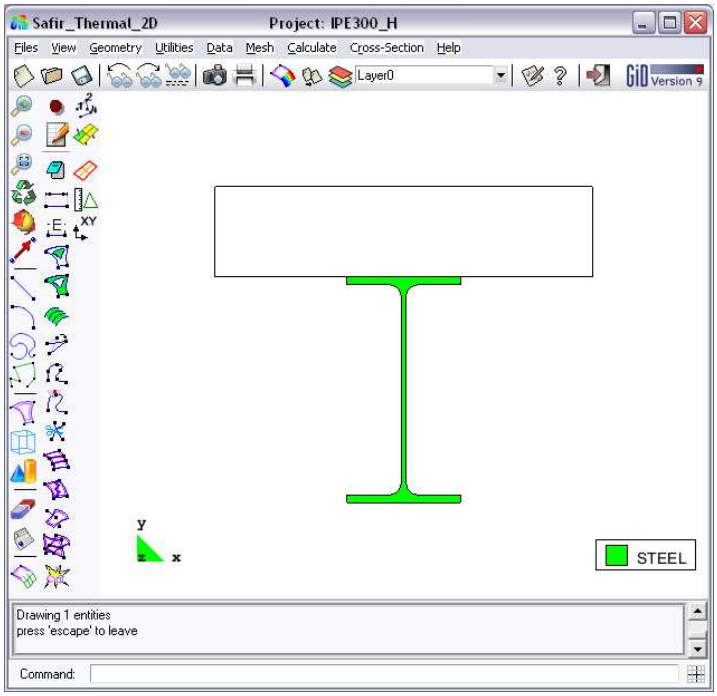
For an IPE300:

$$k_{sh} = 0.9 \times \frac{167}{216} = 0.69583$$

As explained in Eurocode 1993-1-2, formula (4.25), you can apply this coefficient to the GiD-Safir convection coefficient hot and relative emission

$$0.69583 * 25 = 17.395$$
$$0.69583 * 0.7 = 0.487$$





Click on ***Assign-> Surfaces*** and assign it to the IPE300 surface

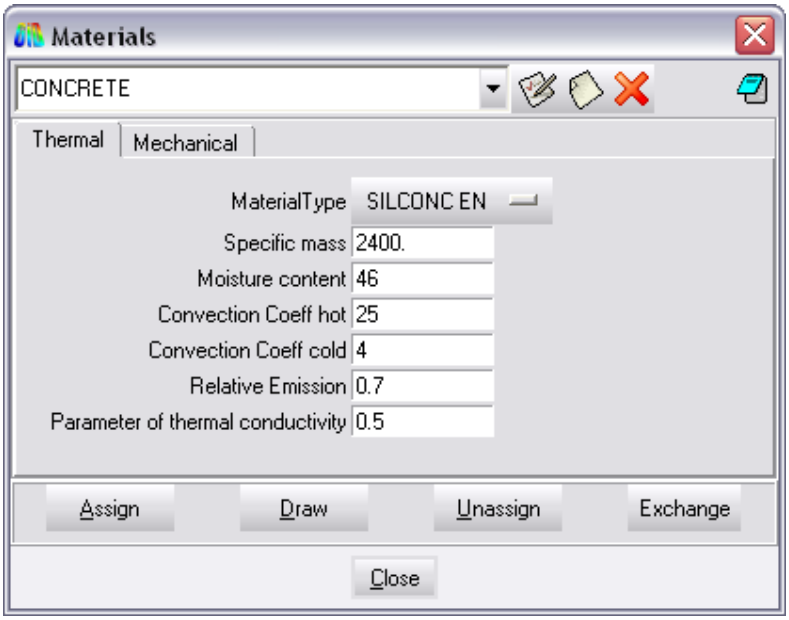
Press ***[Esc]*** or ***Finish*** to confirm

Select ***DRAW->all materials*** in the Material dialog box to display Materials

Press ***[Esc]*** or ***Finish*** to leave

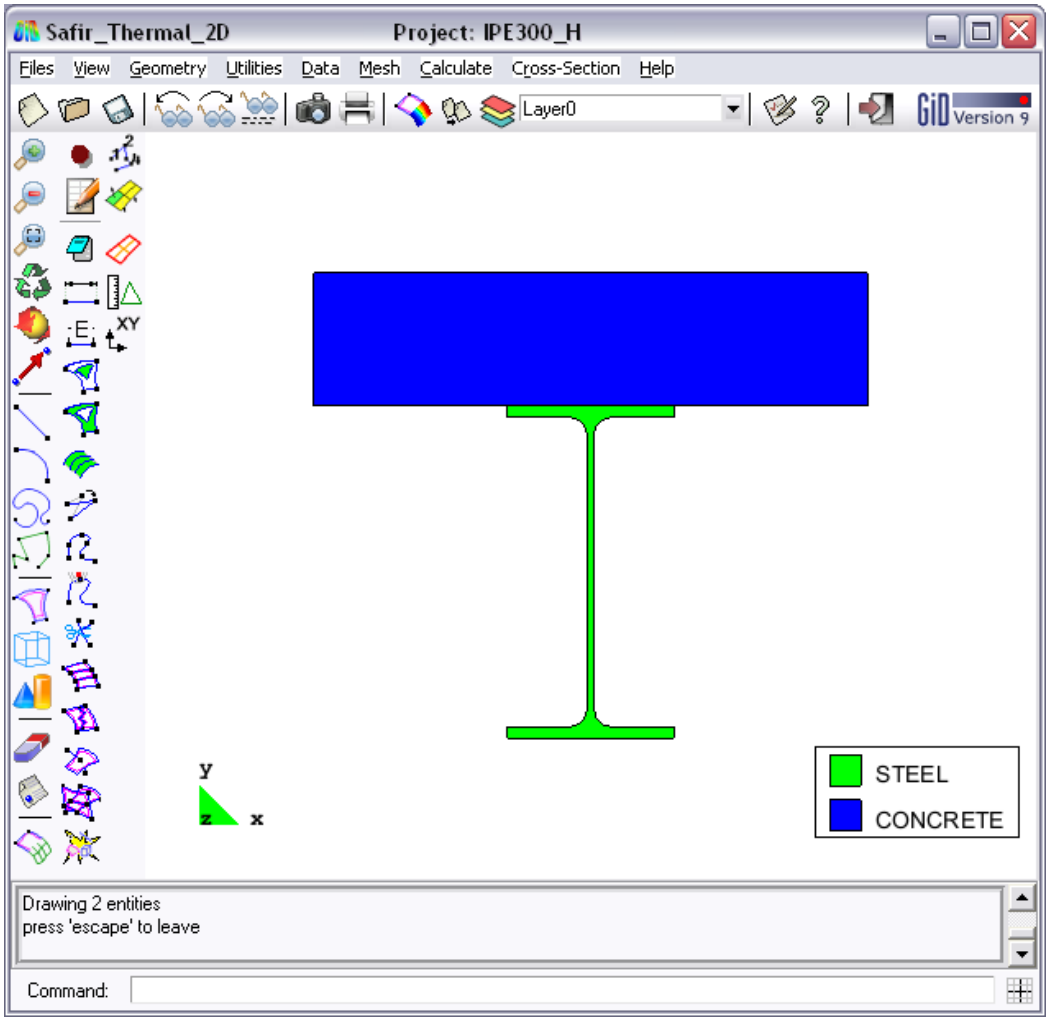
Select ***CONCRETE*** from the dialog box pull down list

Then fill as below:



Click on ***Assign-> Surfaces*** and assign it to the slab surface

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


Safir_Thermal_2D
Project: IPE300_H
Files View Geometry Utilities Data Mesh Calculate Cross-Section Help
Layer0
GiD Version 9
y
z x
STEEL
CONCRETE
Drawing 2 entities
press 'escape' to leave
Command:

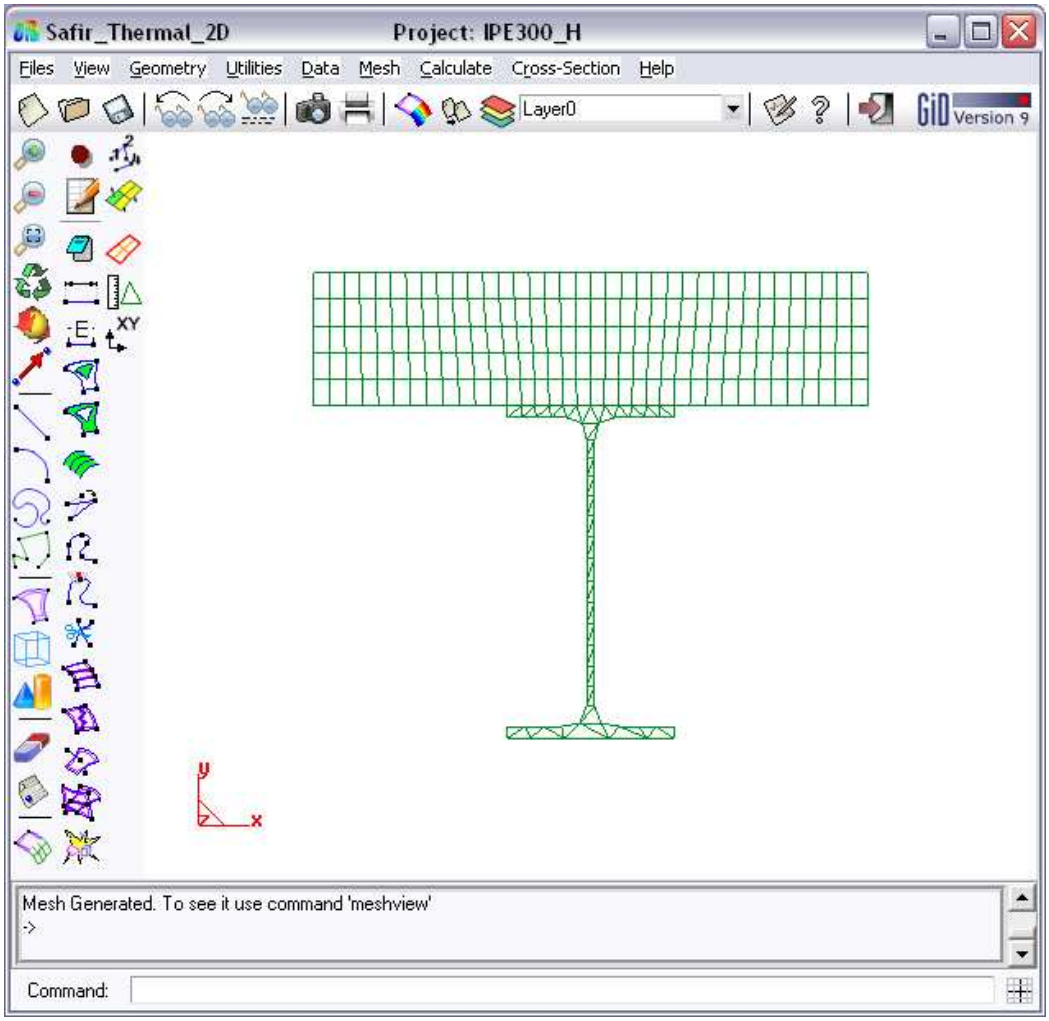
5. Create the mesh

Select the slab surface and press [Esc] to confirm

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

Enter 0.04 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, the maximum allowed is 1010 nodes*

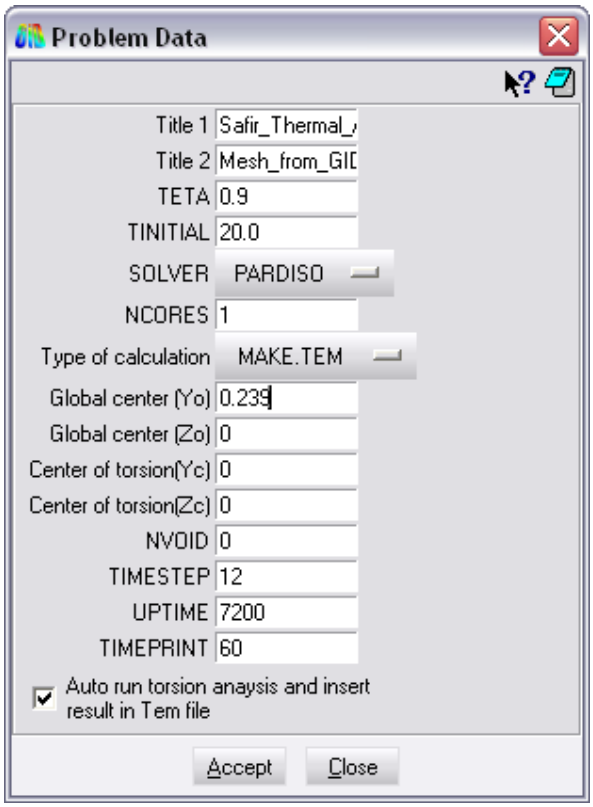


6. Assign General Data

From the pull down menu select:

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

Fill as shown below:



The global center is automatically centered in the center of the IPE profile. In order to position it correctly in comparison to the shell element already designed the global center (Yo) have to be shift:

ipe300 height/2 + (slab thickness – 0.31) = 0.3/2 + 0.89 = 0.239 m

In the Problem Data dialog mask enter:

TIMESTEP, UPTIME, TIMEPRINT as needed

Click on the ***Accept*** data button

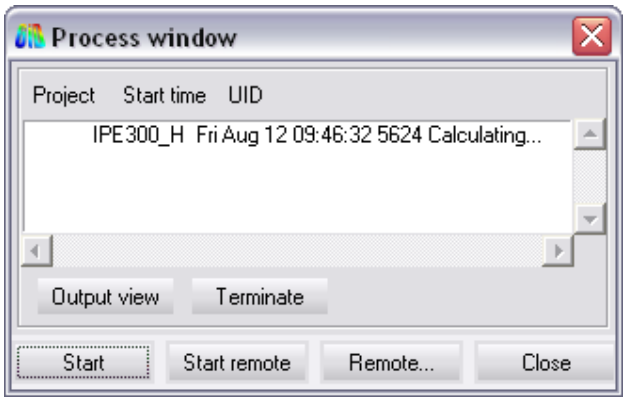
⚠ *When you click with the right button on one of the variables, GiD will display an online help message. The variables are also described in more detail in the SAFIR reference manual.*

7. Start the calculation

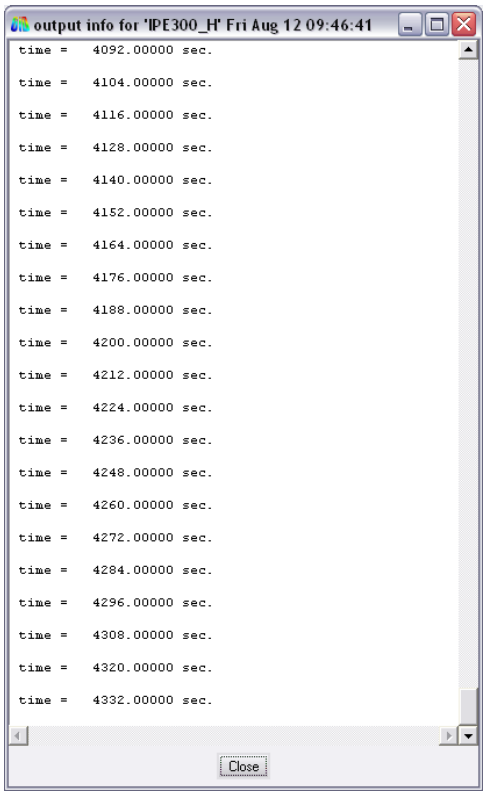
From the pull down menu select:

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

Click the ***Start*** button



Click the ***Output View*** button

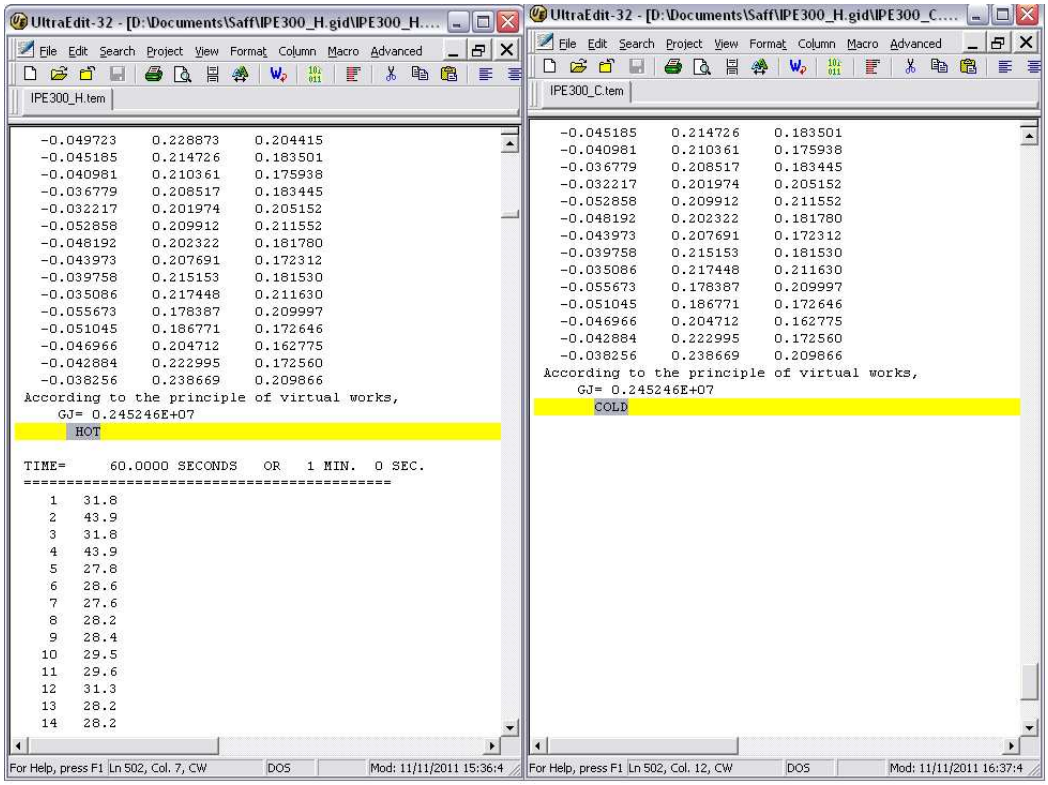


GiD creates a .IN file in the project directory and starts the calculation.

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. Don't forget to put the temperature curve fill into the GiD-Safir file.*

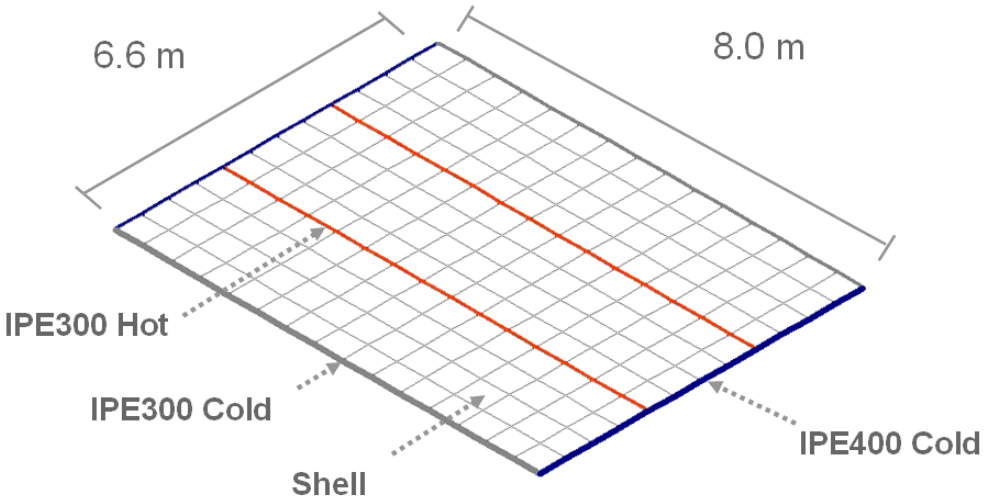
Then create an IPE300_C.tem file (the same file without frontiers constraints conditions). To create it, open the IPE300_H file, search the word "HOT" and replace it by "COLD" and delete all the following text.



Save this new file IPE300_C.tem

As for IPE300_H.gid file, create an IPE400_H and IPE400_C.gid file

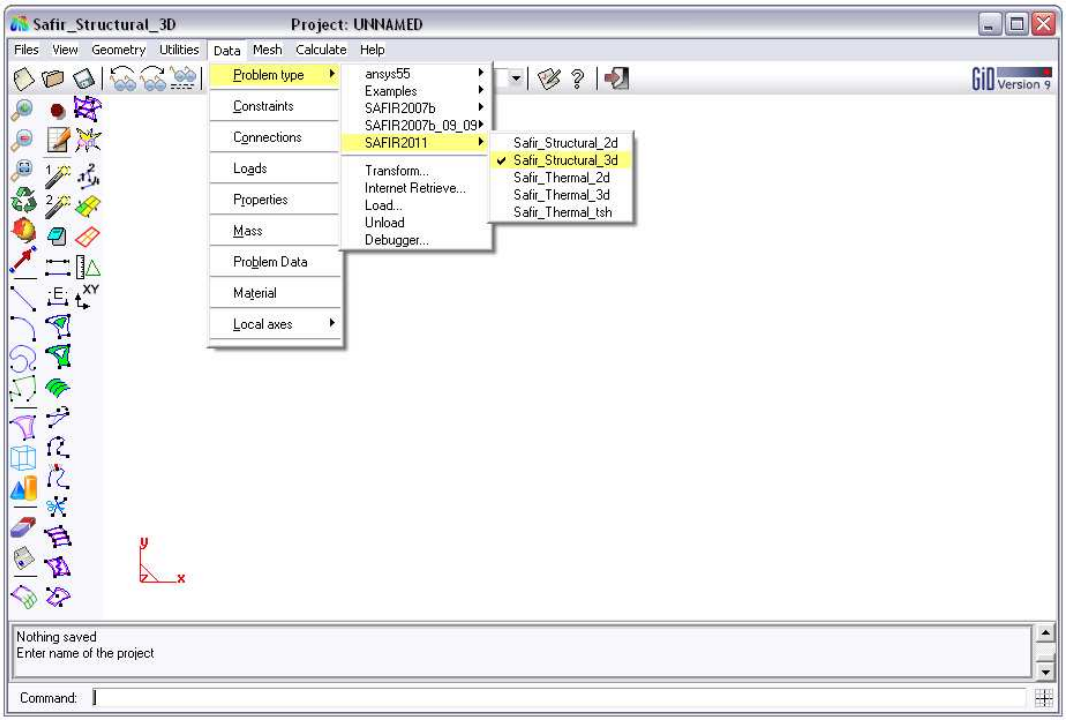
C. Create the 3D structure



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.: *Fracof*

GiD creates a directory with the name *Fracof.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.

2. 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:

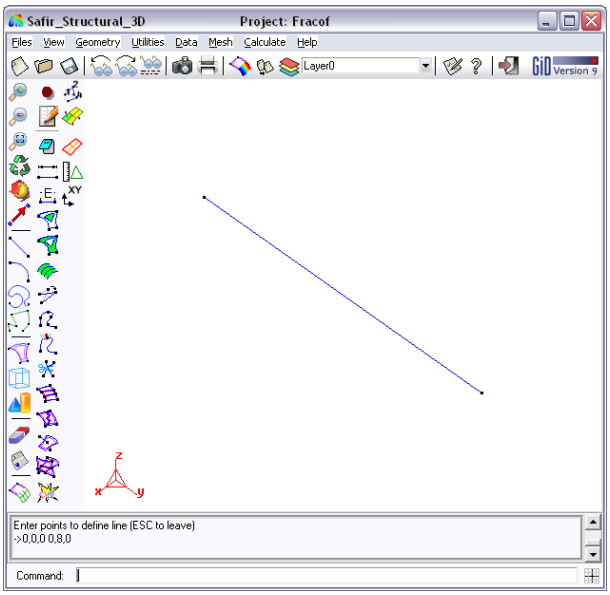
- *View->Rotate->Trackball* *or [F7] or* 

Create the system lines:

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

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

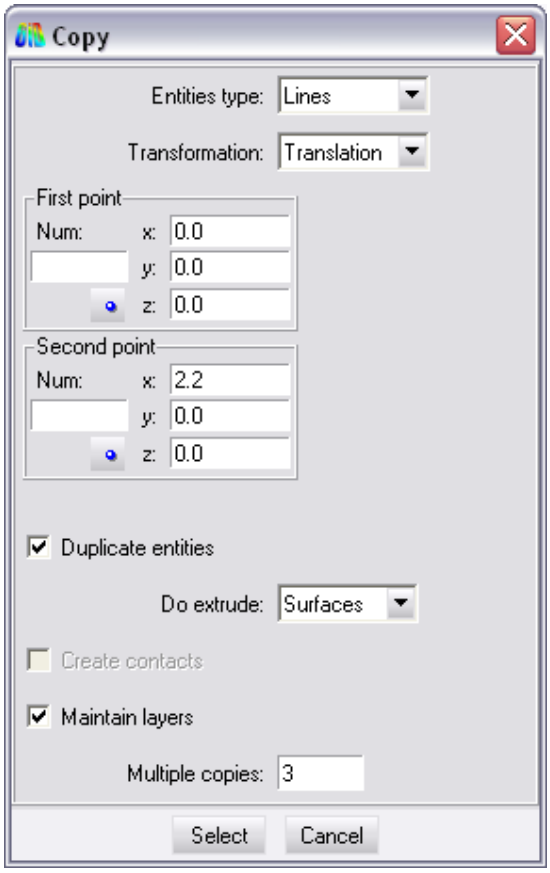
0,0,0 0,8,0 and press *[Enter]*



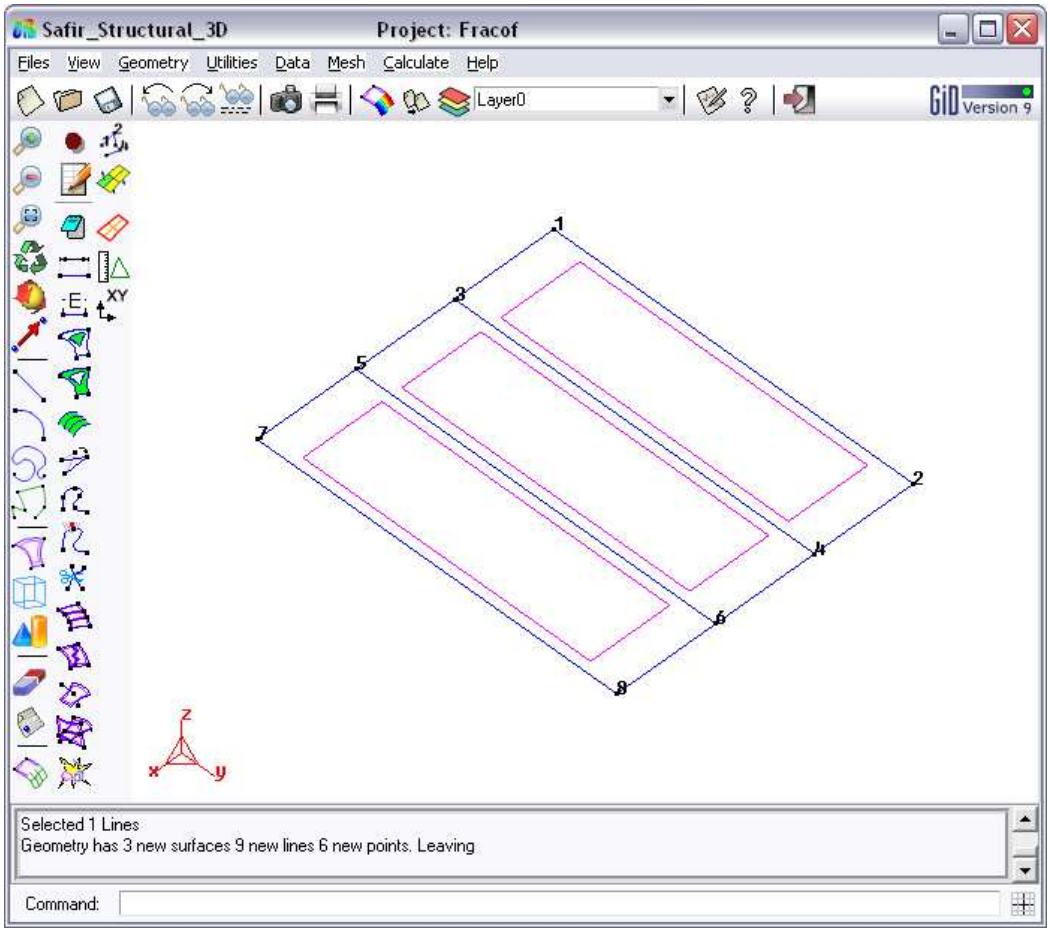
To create the 2 other lines:

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

And fill as shown below:



Then click on ***select*** and select the line and press ***[Esc]***



To see nodes and beams numbers select:

- ***View->Label->All in-> Nodes***

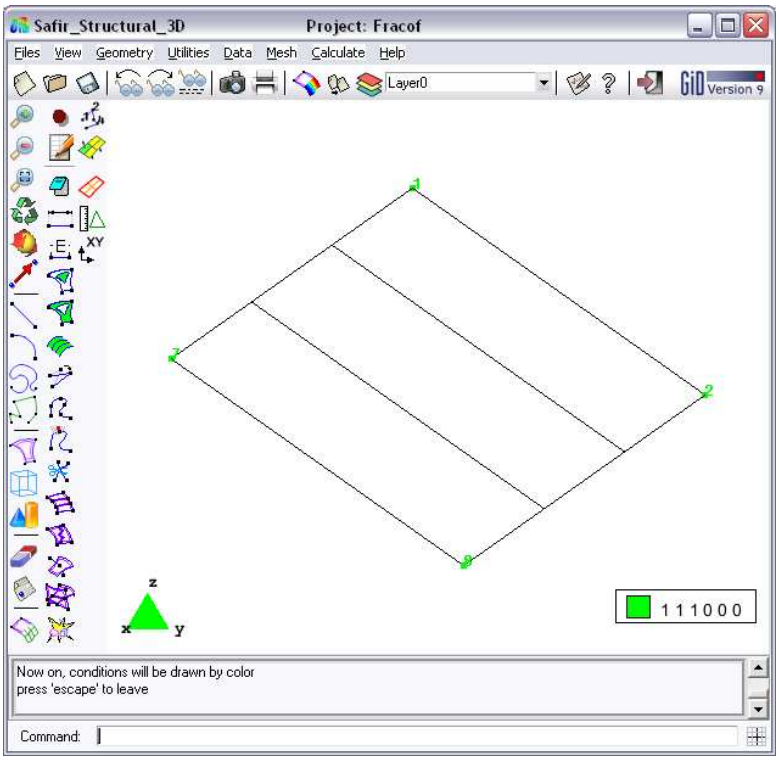
3. Define constraints for the supports:

From the pull down menu select

- ***Data->Constraints***

Select x,y and z constraints and assign them to POINT ***1, 2, 7*** and ***8*** and press ***[Esc].***

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

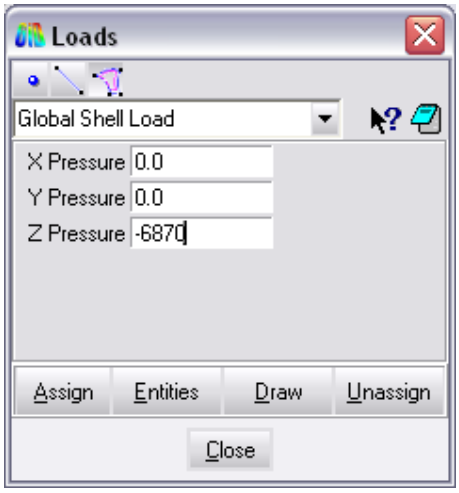


Press ***Finish*** or ***[Esc]*** to leave this view mode

4. Define loads

From the pull down menu select:

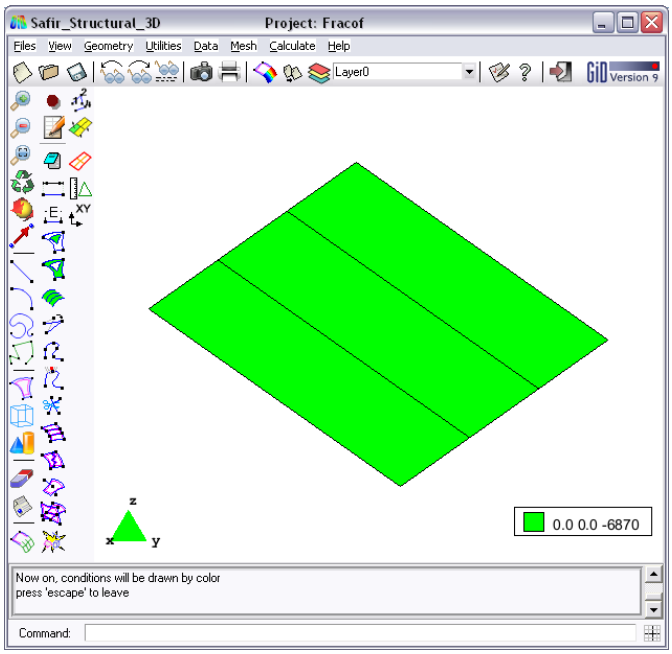
- *Data->Loads*



In the dial box select *Global Shell-Load* and enter a Z-Pressure of –6 870 N/m2 and *Assign* it to all surfaces.

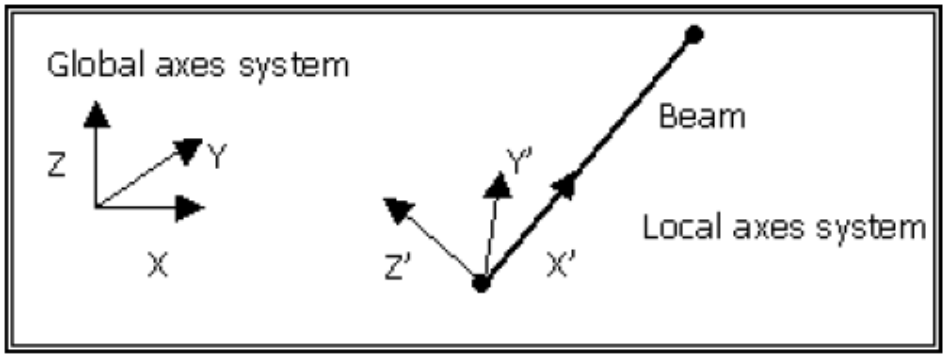
To display the loads select *Draw->Colors* in the dial box

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



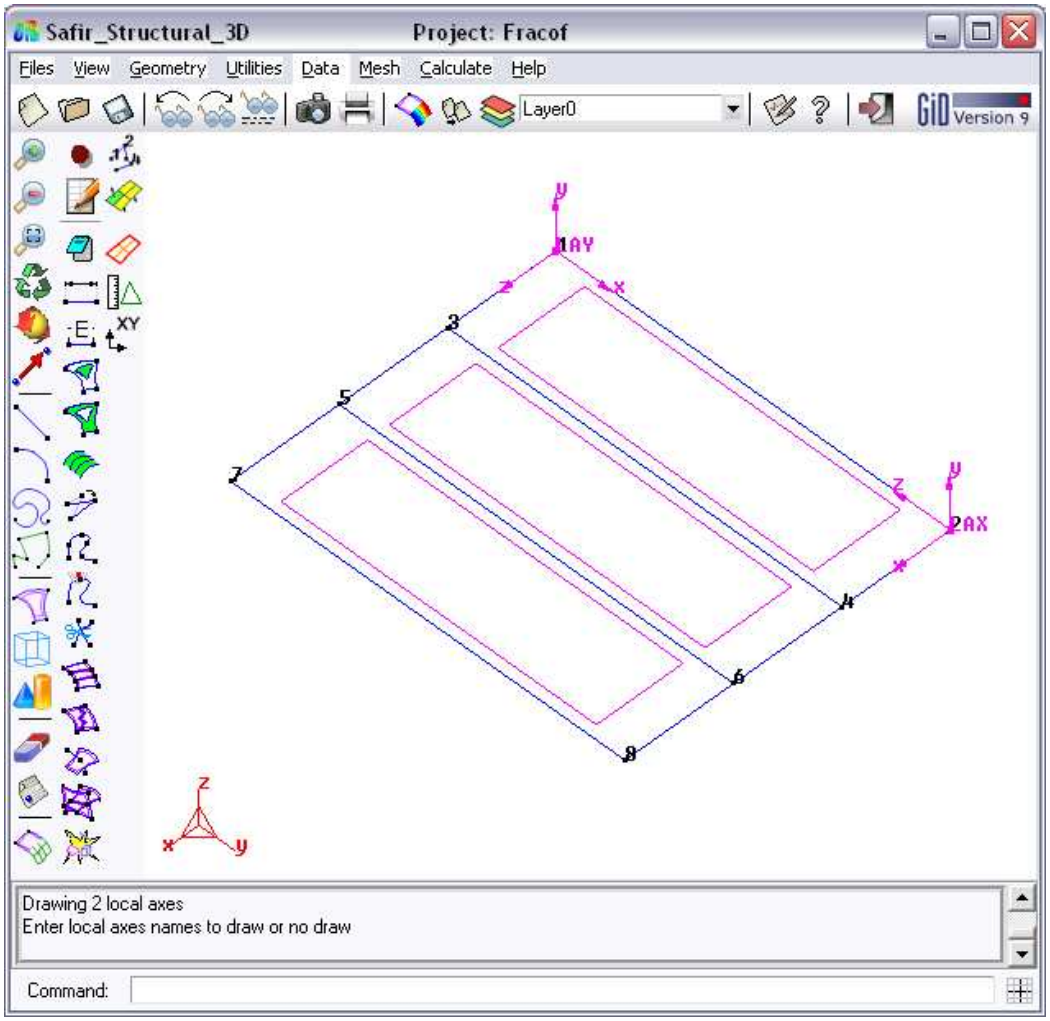
5. Create Local axes

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 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:

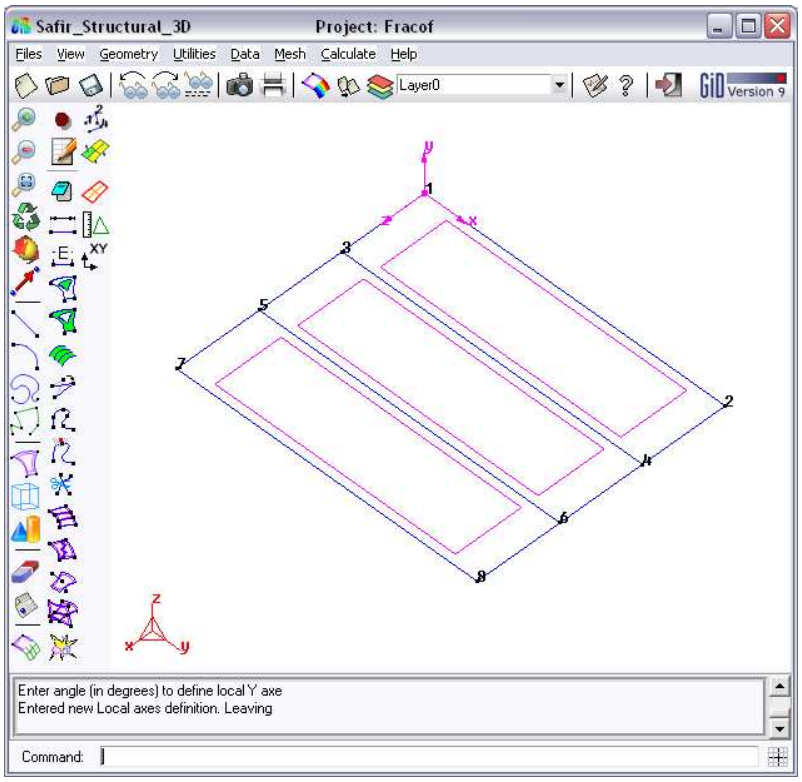
- ***Data->Local Axes->Define***

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

Select ***XandAngle***

Press ***[Ctrl+a]*** to select the local axe center. 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)



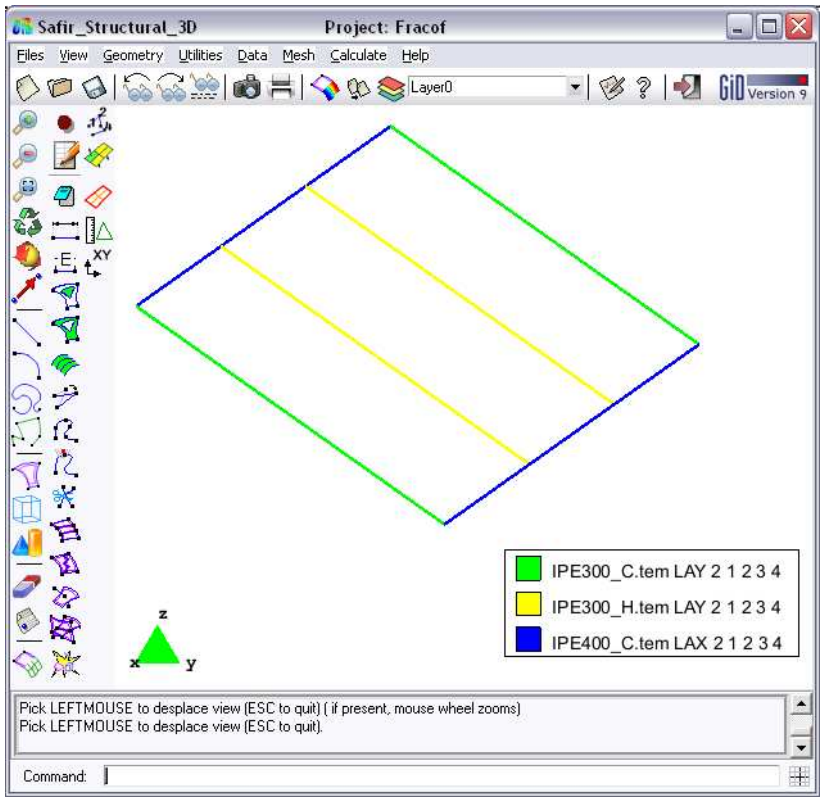
Do the same operations in order to create local axes LAX

To draw local axes select:

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

6. Assign temperature files (.TEM and .TSH files)

The objective is to assign the .tem file named (IPE300_C; IPE400_C and IPE300_H) to the system lines



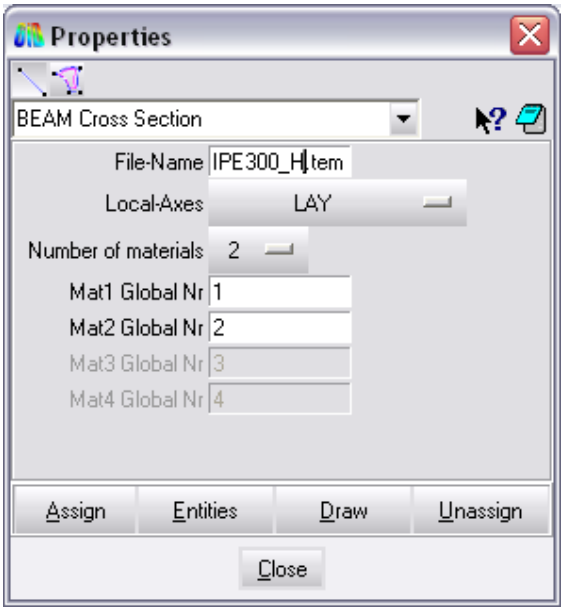
From the pull down menu select:

- ***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 ***IPE300_C.tem***.

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

In this case all cross-sections have two materials (Steel and concrete as defined during the .TEM creation)



Assign the ***IPE300_C.tem*** section to beams (in this case beams 1-2 and 7-8).

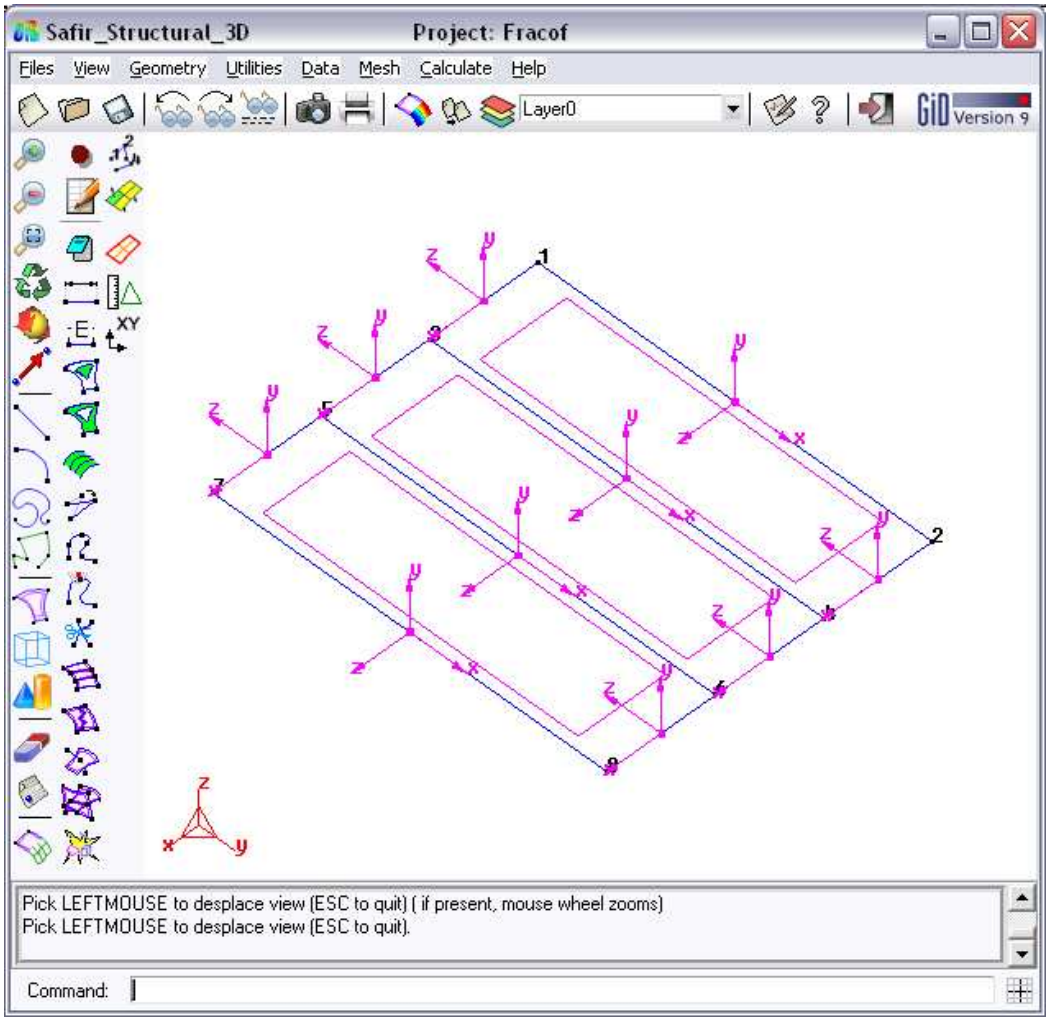
Do the same operation in order to assign a .tem file to all systems lines

⚠ *Don't forget to change the local axe and materials numbers*

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***

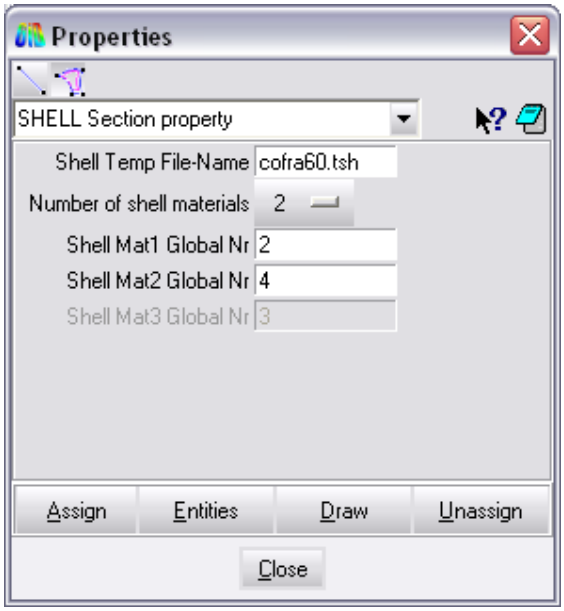
Press ***Finish*** or ***[Esc]*** to leave this view mode

From the pull down menu select:

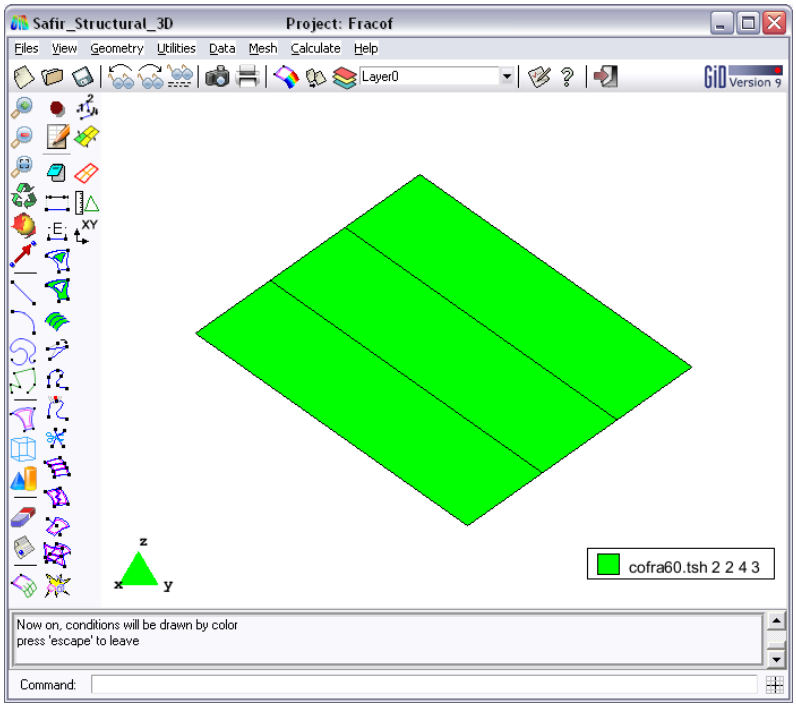
- ***Data->Properties***

Open the SHELL Section property tab and change the File-Name: ***test.tsh*** to the temperature file (.TSH file) of the slab section, in this case ***cofra60.tsh***.

In this case the slab has two materials (Mat 1 = concrete and Mat 2 = Steel S500)



Then assign this .TSH to the entire slab



To display Property select in the dialog box:

Draw->Colors

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

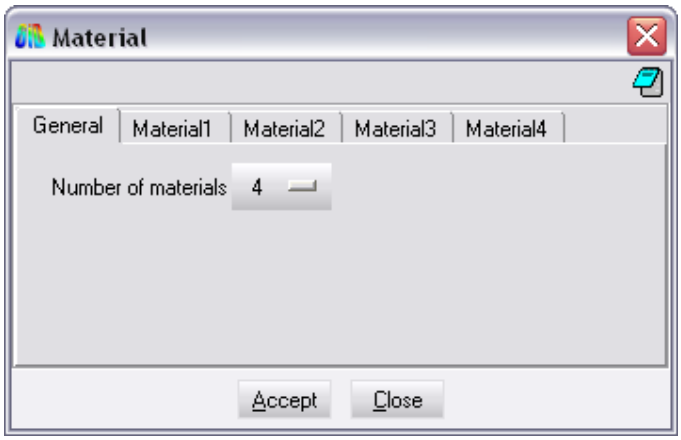
7. Define global materials:

I this case we are using 4 different materials: one steel for rebars, one steel for beams, insulation for .TEM slab and concrete for .TSH slab.

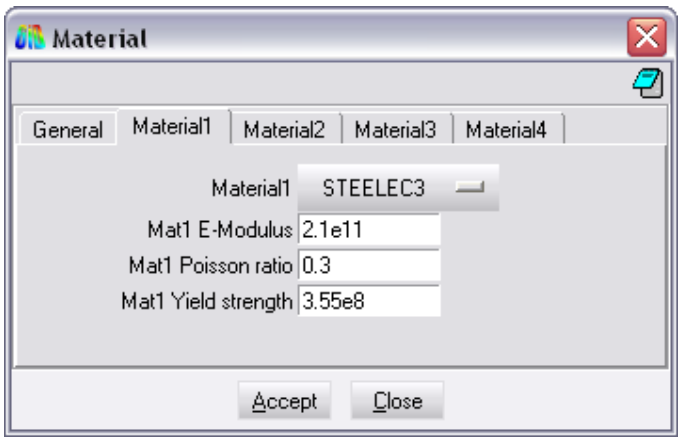
To define materials select from the pull down menu:

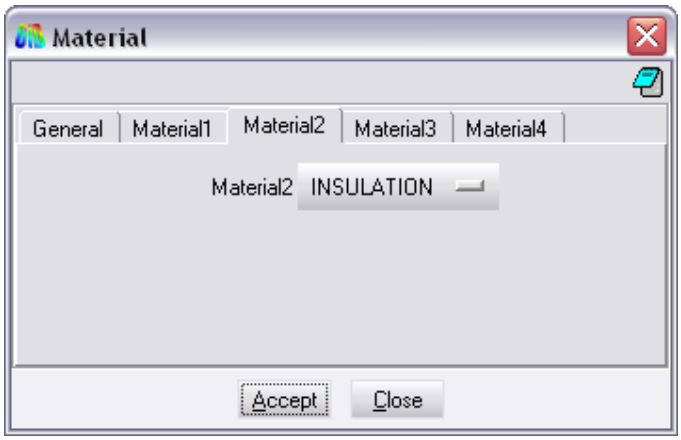
- ***Data->Material***

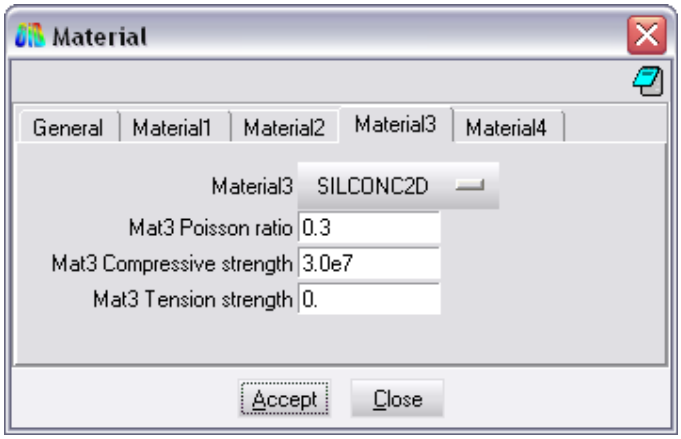
In the general tab, put 4 materials.

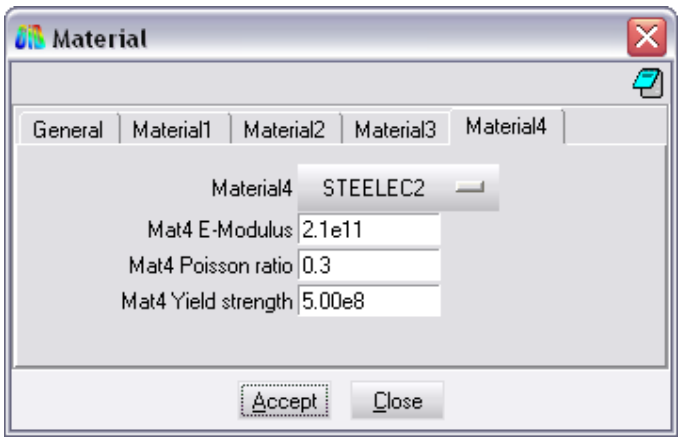


As defined in the property part, steel S355 is material 1, insulation is material 2, concrete is material 3 and steel S500 is material 4:¨









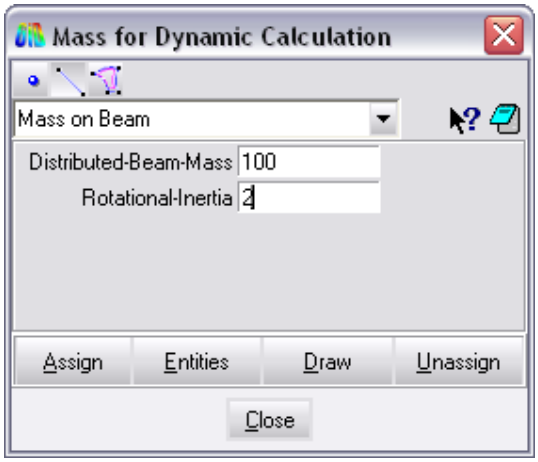
Click on ***Accept***

8. Define the Mass

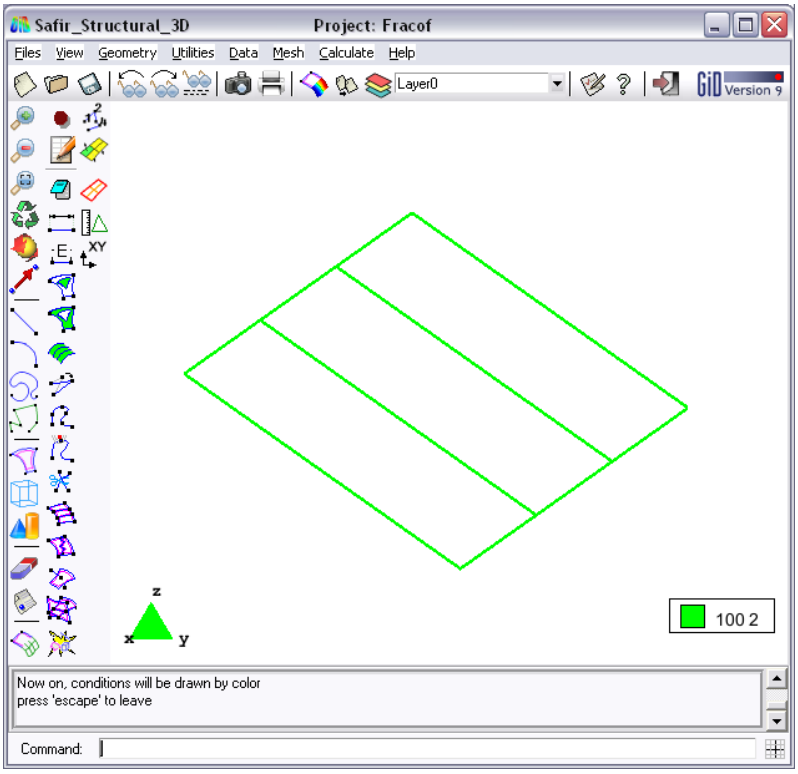
To define the mass for dynamic calculation, select from the pull down menu:

- *Data-> Mass*

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



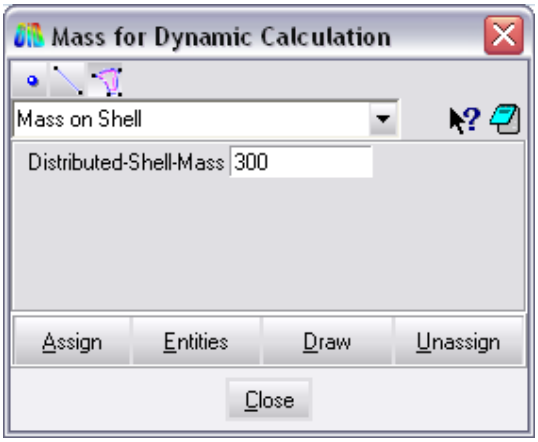
Assign the mass to all the beam elements



To display Property select in the dialog box:

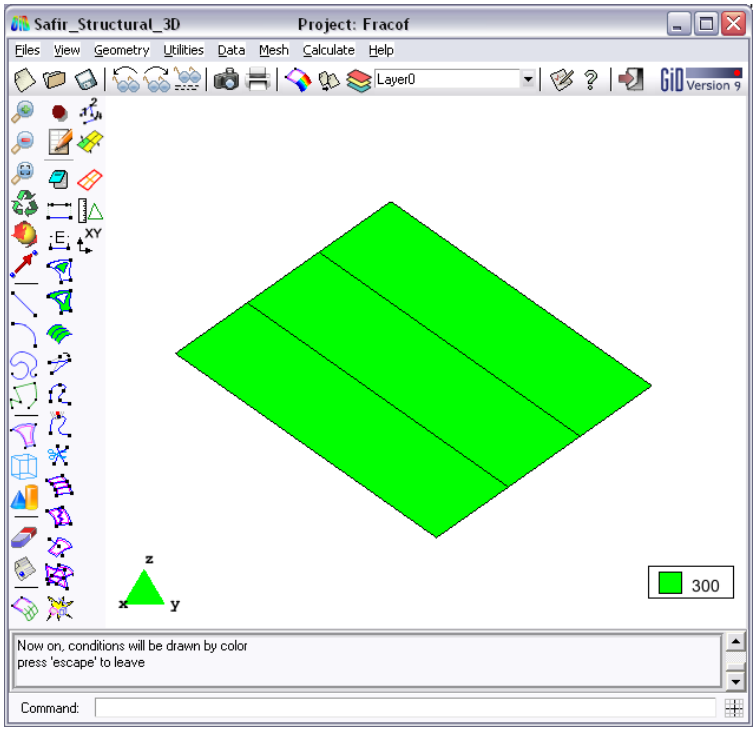
Draw->Colors

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



Select the shell tab, put 300 kg/m2 as Distributed-Shell-Mass

Assign the mass to all the shell elements



To display Property select in the dialog box:

Draw->Colors

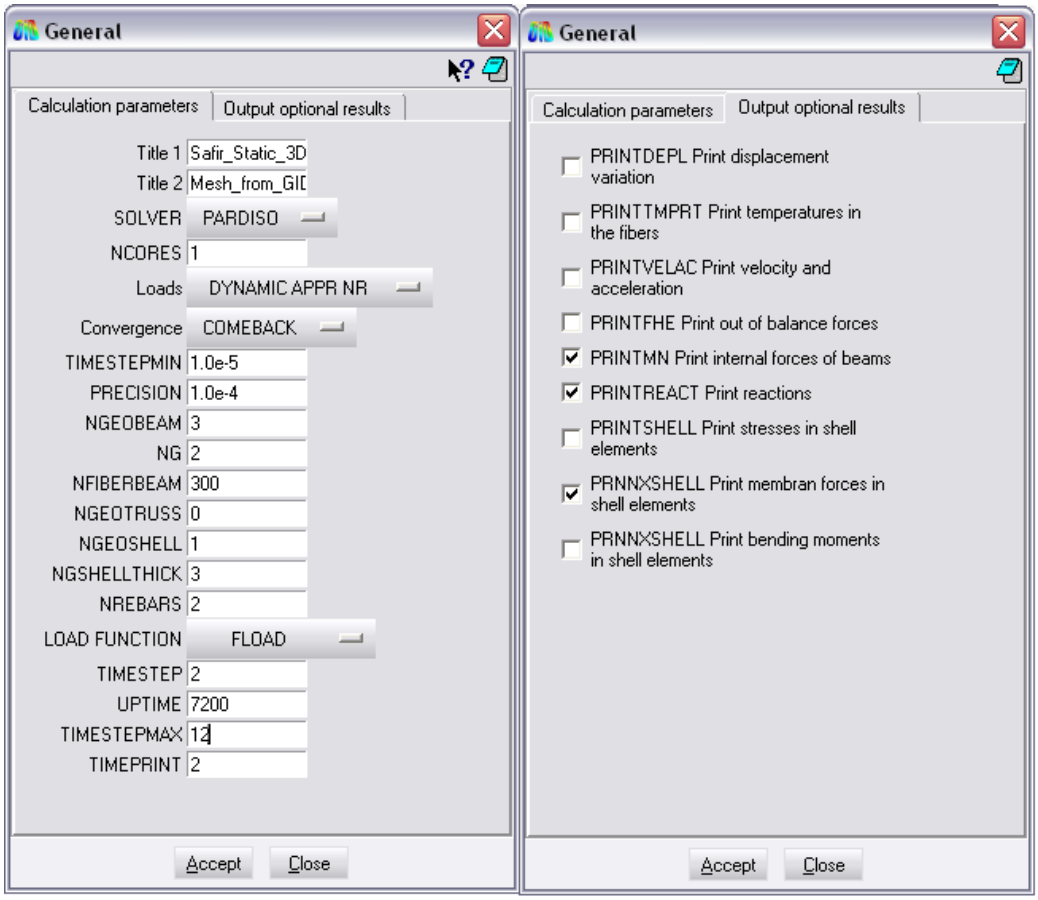
Press ***Finish*** or ***[Esc]*** to leave this view mode

9. Define general data

Select from the pull down menu:

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

And fill as shown below



Enter the following

NGEOBEAM = 6 (the number of .tem files)

NFIBERBEAM = 300 (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 300 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*

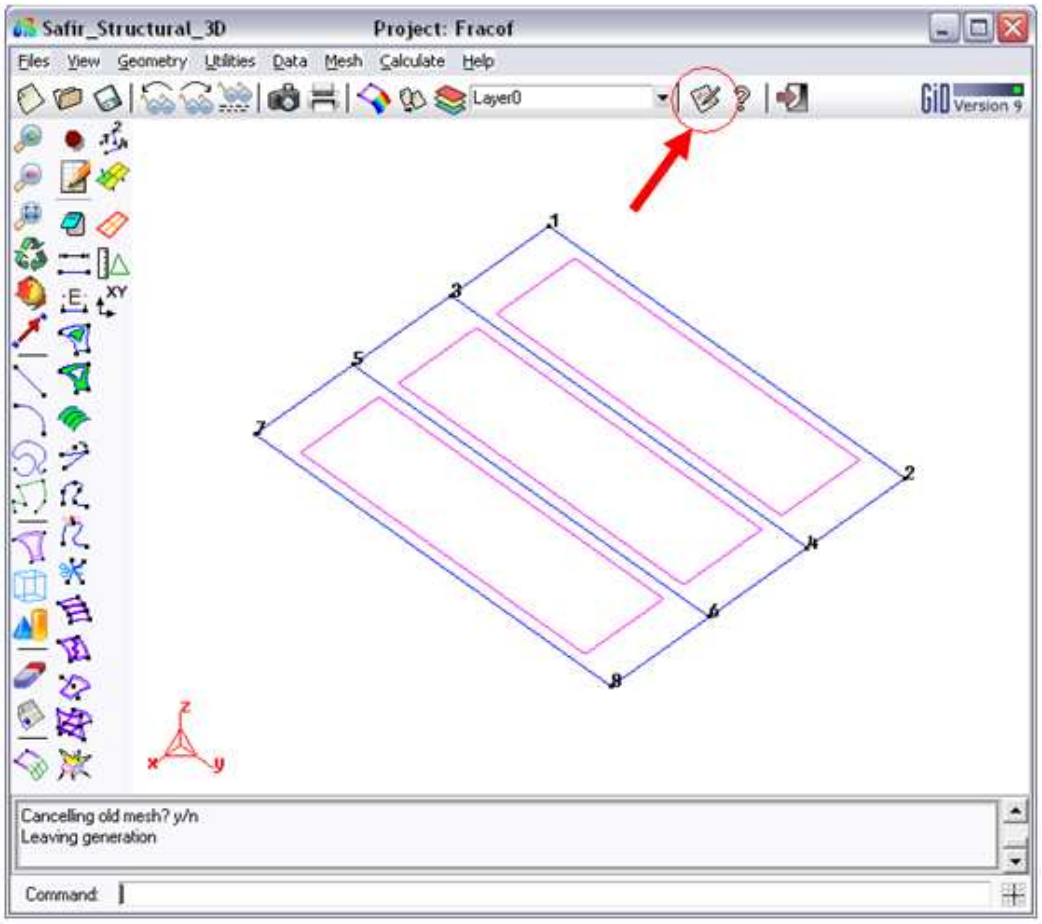
⚠ *You can change TimestepMin, Precision, Timestep, Ttimestepmax and Timeprint as needed but you have to be careful that your UPTIME is less or equal to the UPTIME used for thermal 2D calculation*

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

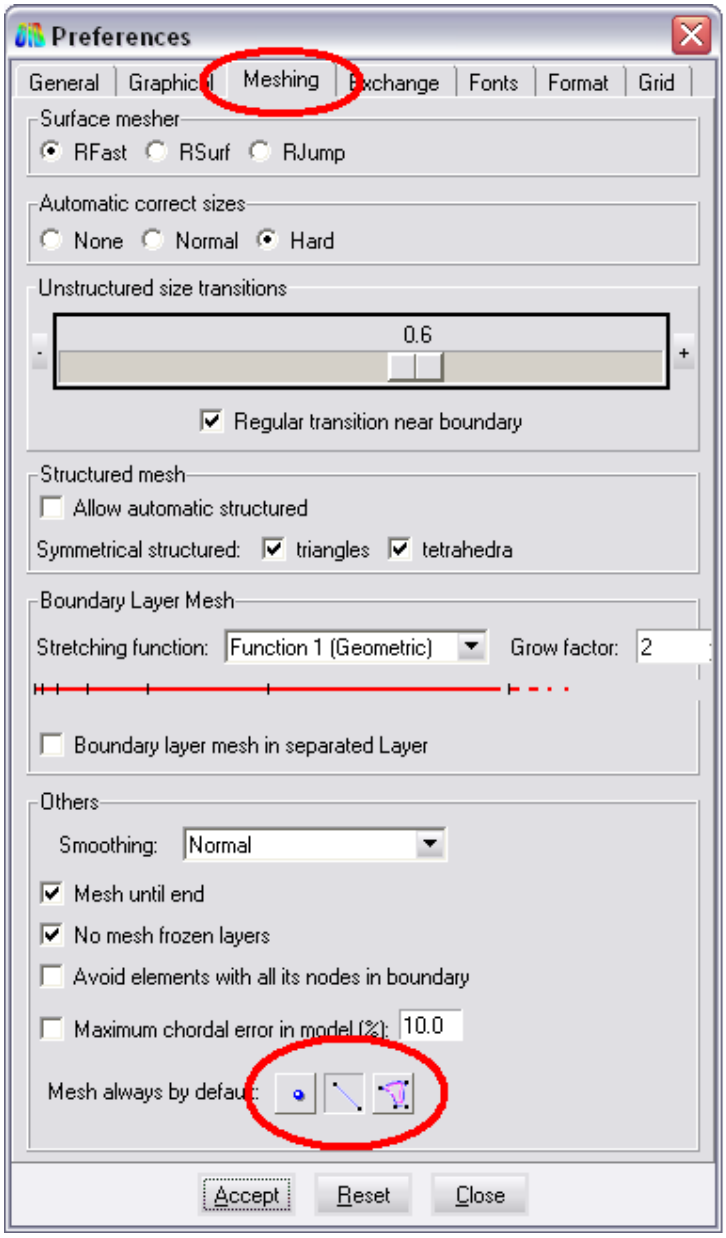
10. Generate the mesh:

Select from the pull down menu:

Preferences:



Then select the Meshing tab:



And select beam as “mesh always by default”

Click on ***Accept***

Then select:

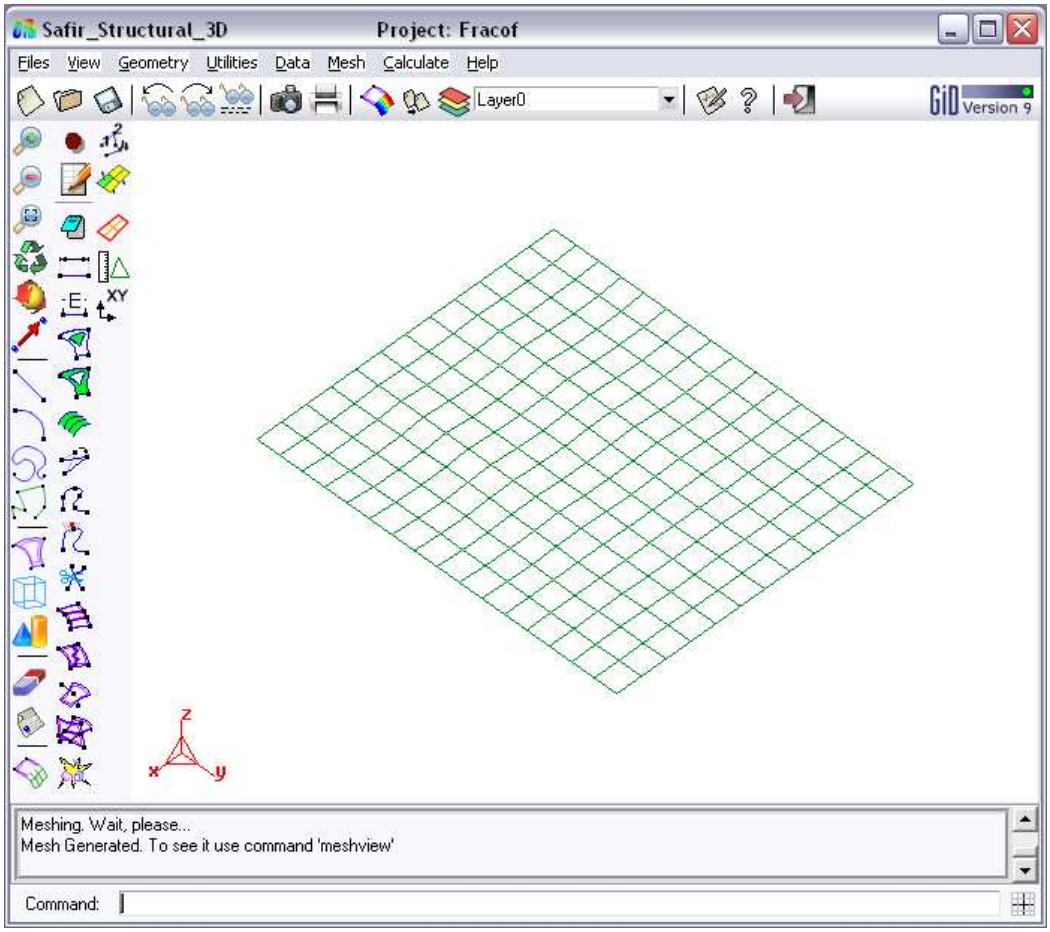
- ***Mesh->Element type->Quadrilateral***

Select all the structure

Then select

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

Enter the element size of 0.50 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

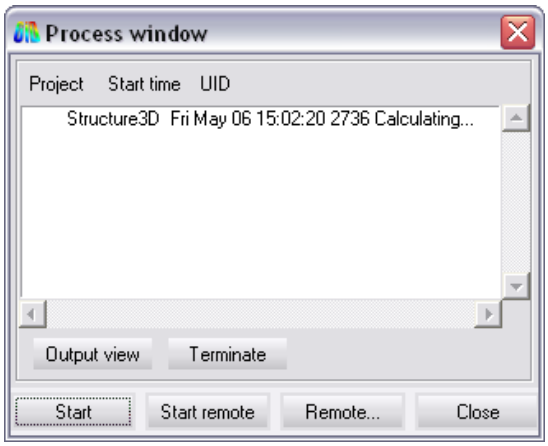
To quit the label mode use

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

11. Start the calculation:

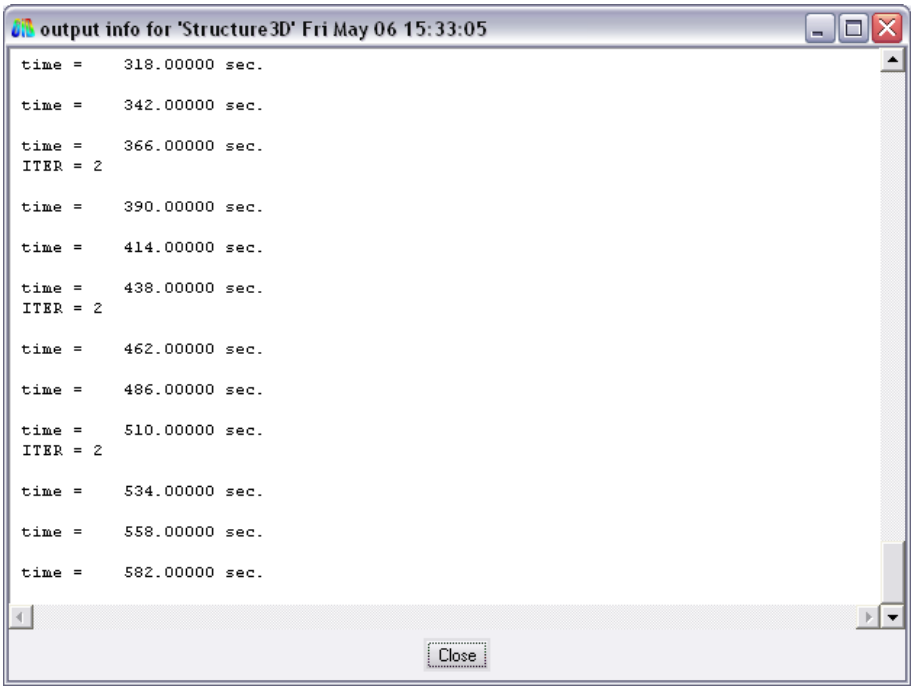
Select from the pull down menu:

- *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.