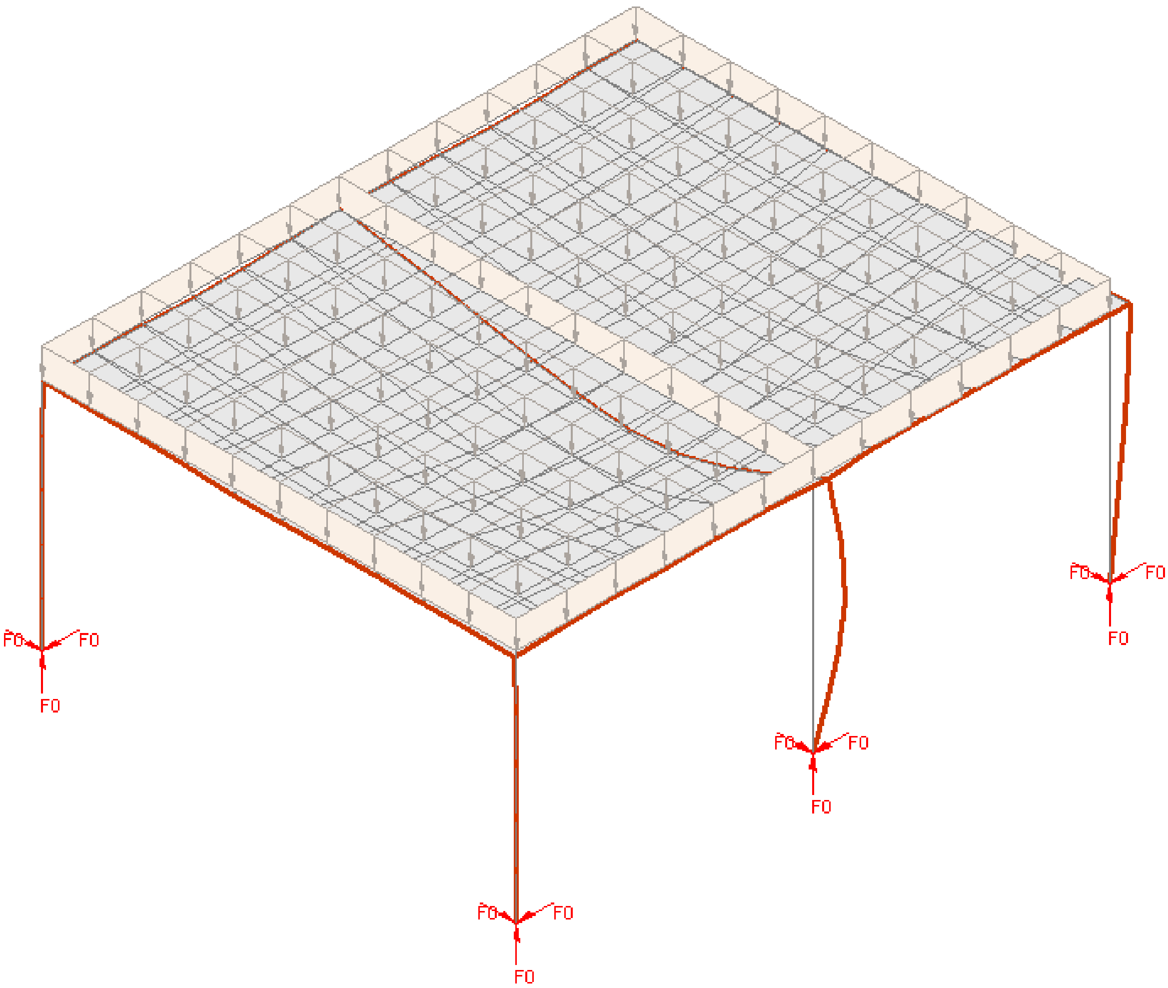
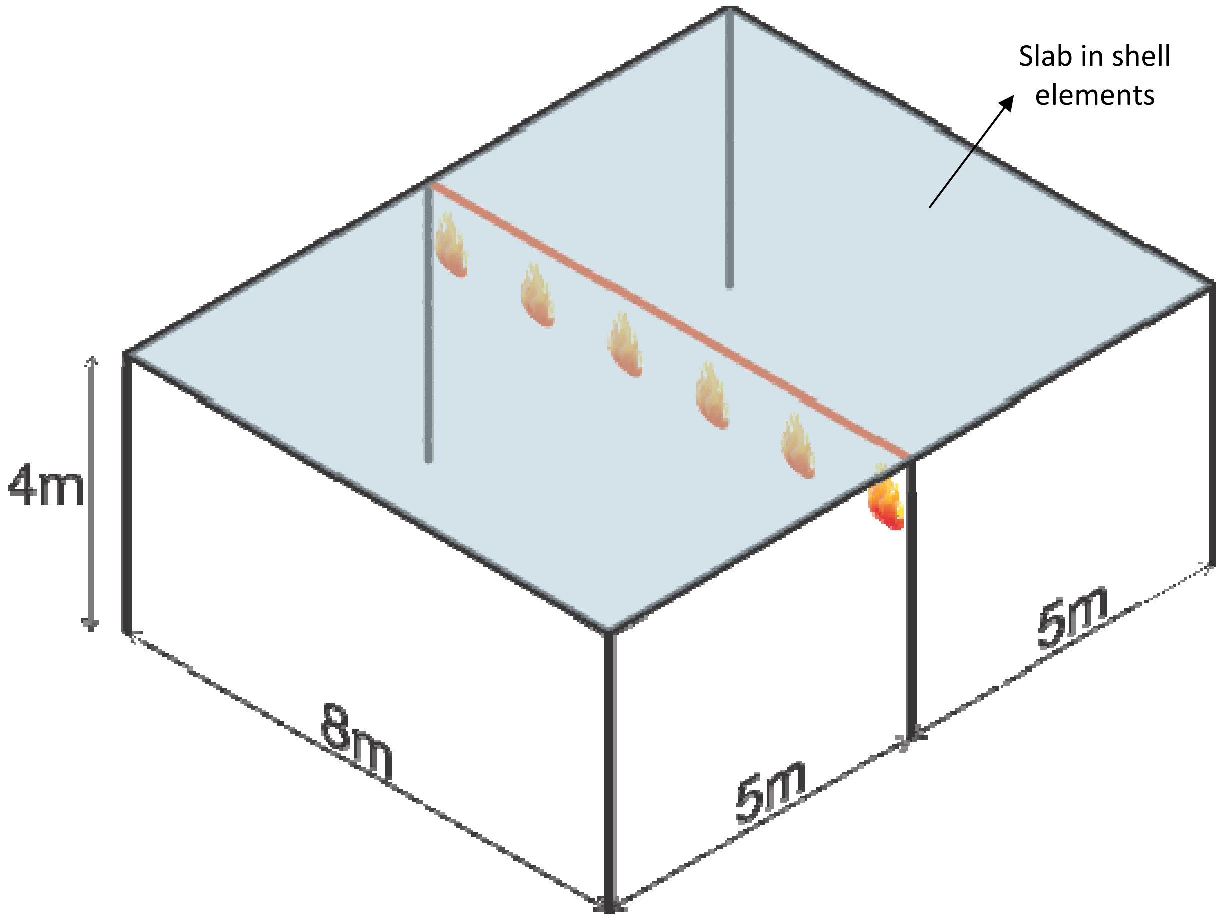
Example of GID-SAFIR

Structural Analysis

Exercise n°9 – 3D Hall with Slab in Shell elements



This example is a 3D frame similar to the frame done in the Exercise 8, except that the slab will be modelised using the shell elements method.



1. Create a new project

Open the GiD project Hall1_3D done in the Exercise 8, and save it in a new project :

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

Enter a file name, eg.: ***Hall2_3D***

2. Define the slab in the system geometry

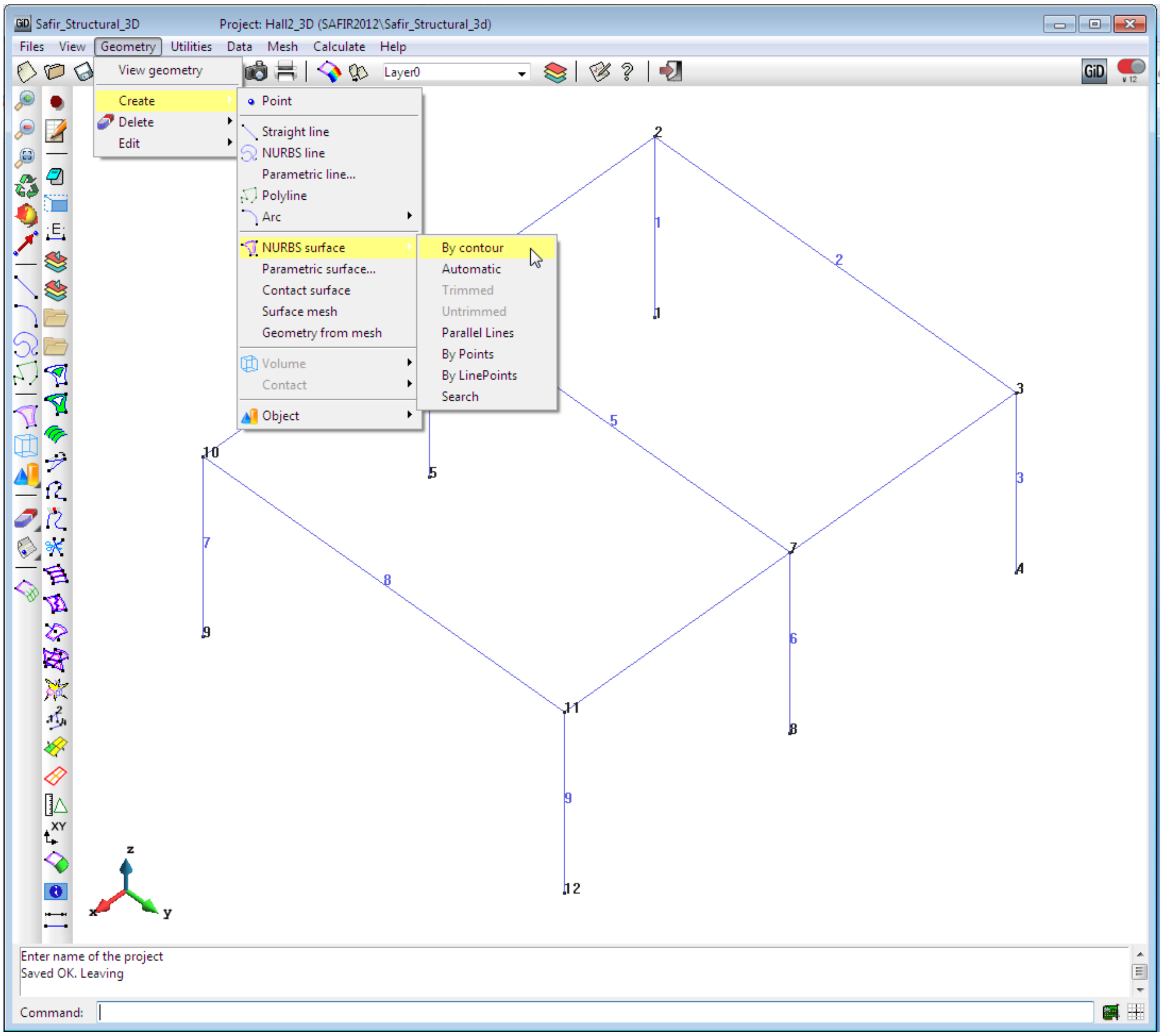
Change to the 3D isometric view : select from the pull down menu

- *View->Rotate->isometric*

In order to assign shell elements, you need to define the surface which will represent the concrete slab.

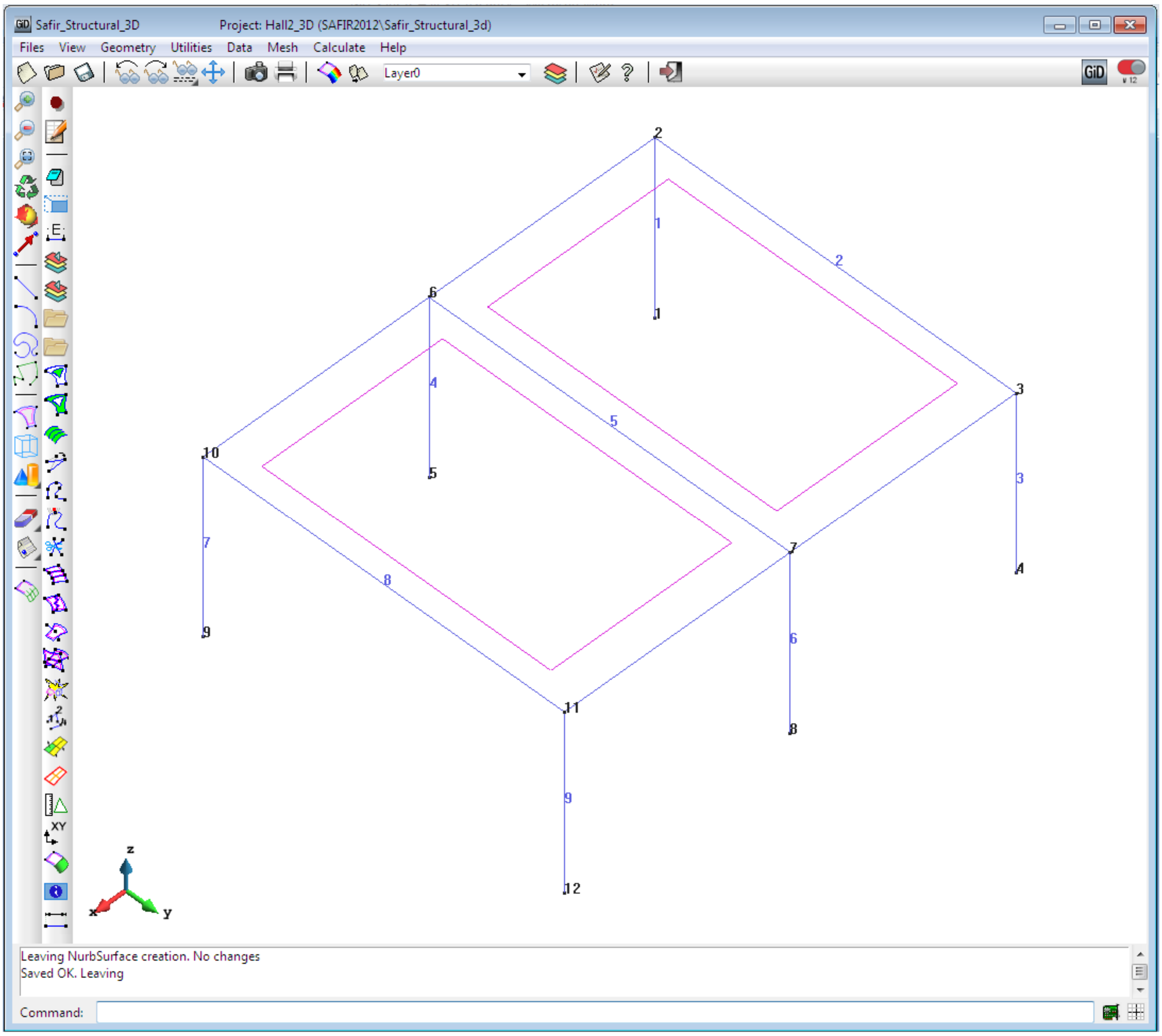
Select from the pull down menu:

- *Geometry-> Create-> NURBS surface-> By contour*



Select the contour lines to create the surfaces as shown in the following figure.

When you have selected the 4 corners of the surface, press *[Esc]* to validate the surface. Then select again 4 corners to define another surface, then press *[Esc]* twice to leave the NURBS surface mode.



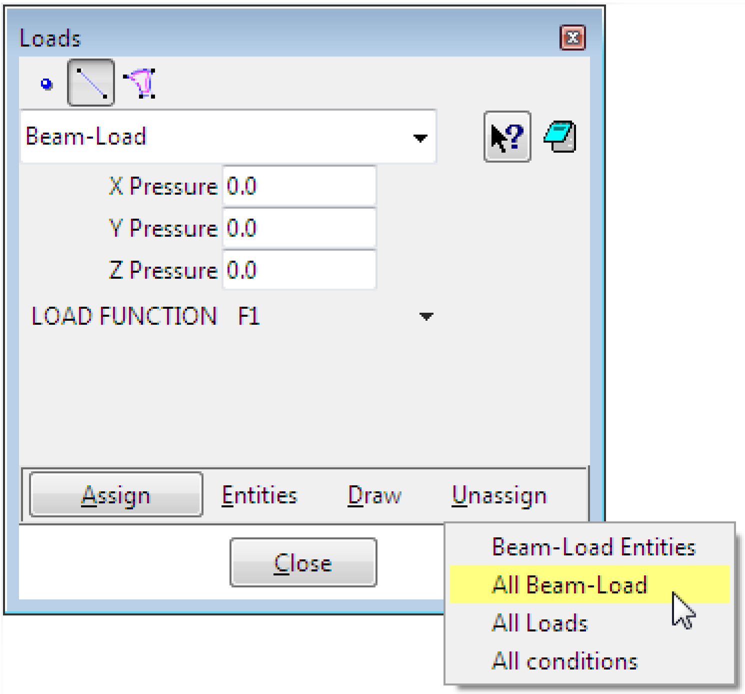
3. Modify the loads

In the exercise 8 the load where applied on the central beam.

As we will put the load only on the slab elements, you have first to remove all the loads from the beam elements.

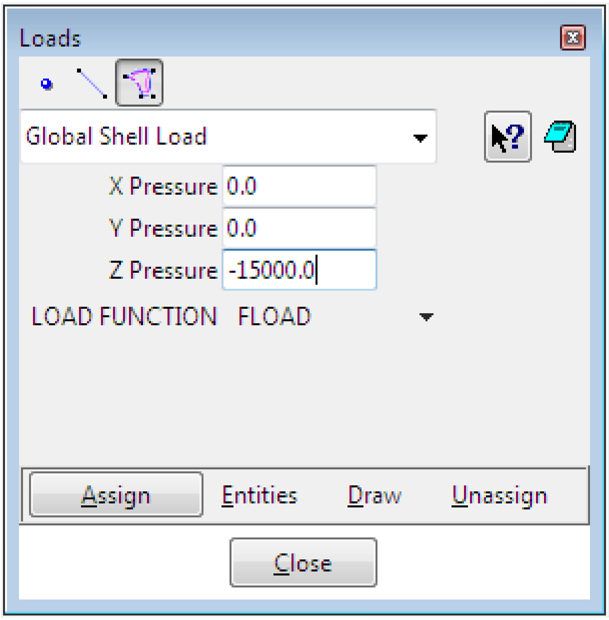
Select from the pull down menu:

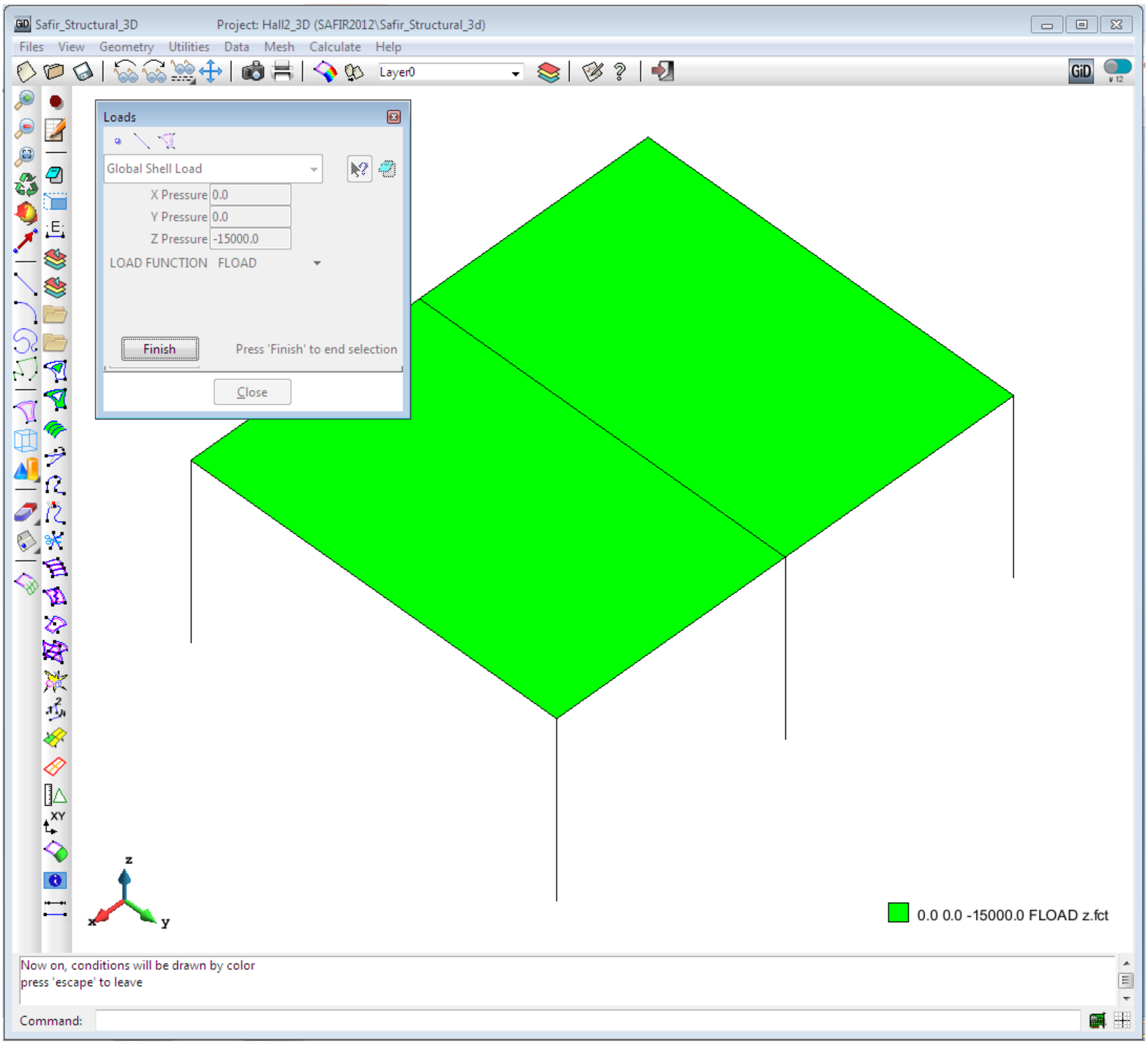
- ***Data-> Loads***



Select the *Beam-Load* tab, click on ***Unassign*** and select *All Beam-Load.*

Then to assign the load to the shell elements, select the *Global Shell Load* tab and assign -15000N/m² as Z Pressure to all the surfaces. Select FLOAD in the LOAD FUNCTION dialog box.





To display loads select in the dialog box:

- ***Draw->Colors***

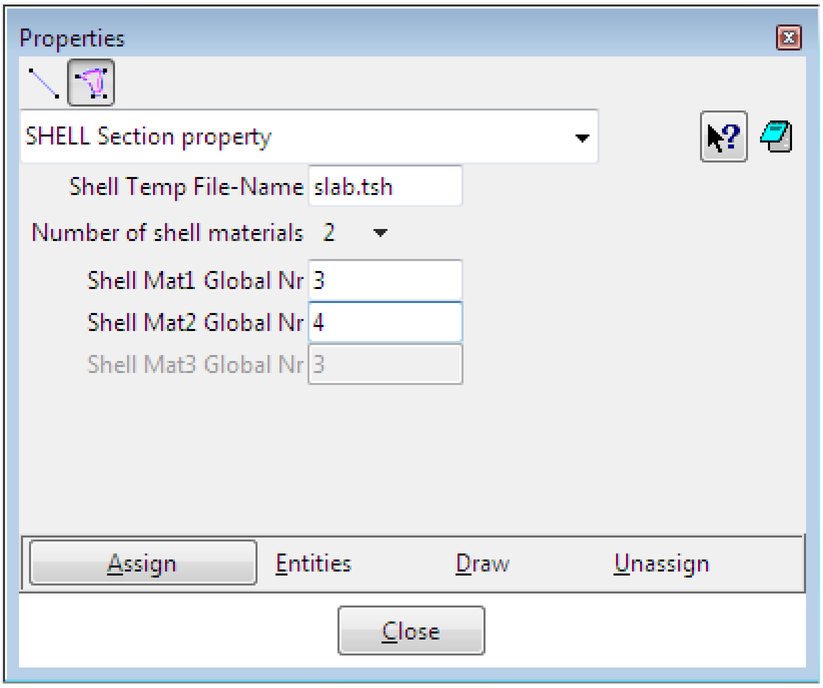
Press *Finish* or *[Esc]* to leave this view mode

4. Assign .TSH file to the slab elements

Select from the pull down menu:

- ***Data-> Properties***

Select the surface properties tab .

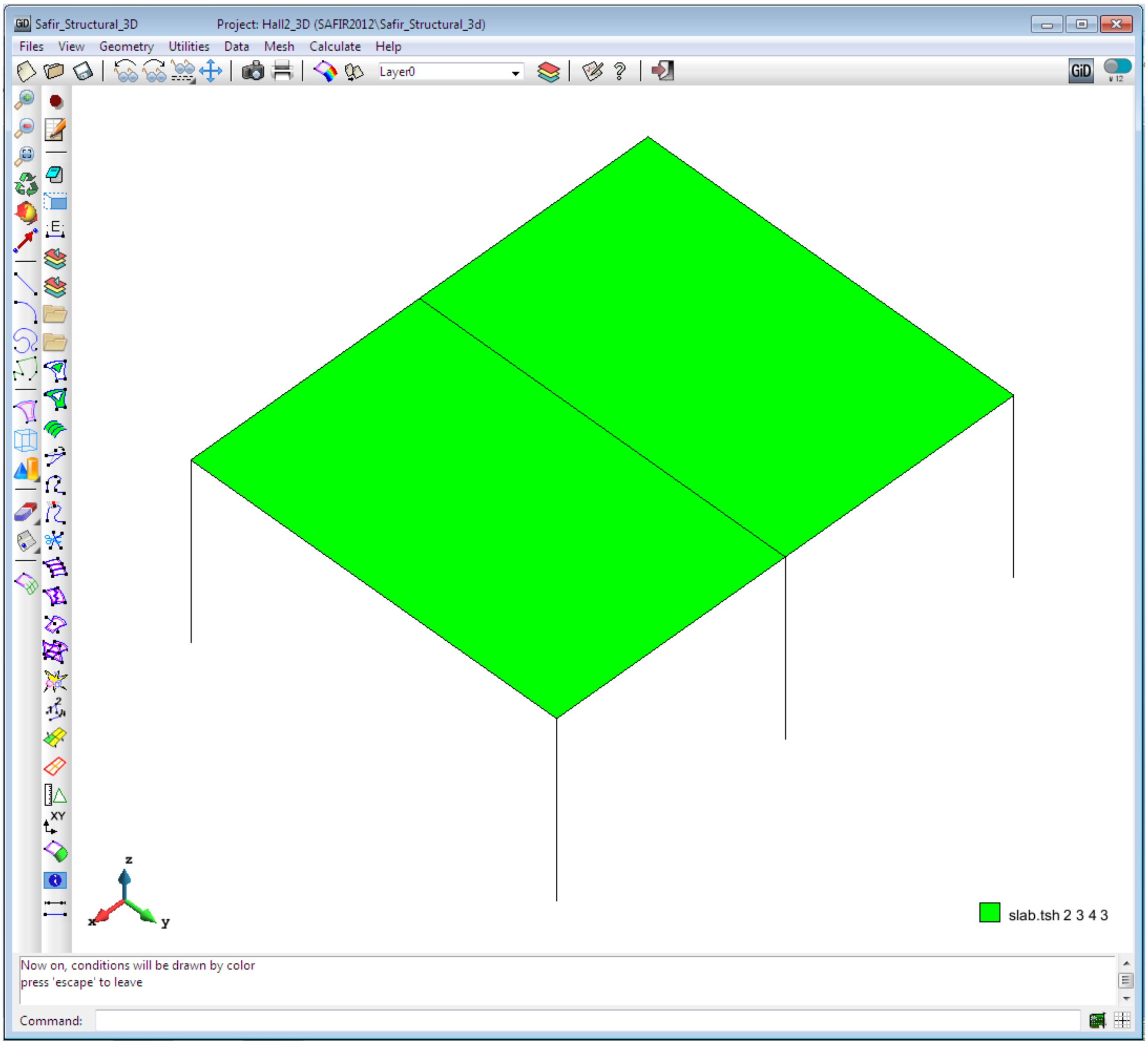


⚠ *Number of shell materials*	*The number of shell material should be the same as defined for the thermal analysis of the considered section (.tsh file).*
*Shell Mat**x** Global Nr **y***	*This is the rank **y** in the "structure.in" file (Global Nr **y**) of the Material **x** (Mat**x**) defined in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

Then fill the dialog box as shown in the figure hereabove.

- As *Shell Temp File-Name*, put ***Slab.tsh*** (= ***Slab130.tsh*** file done in Exercice 4)
- As *Number of shell materials*, put ***2*** (one for the concrete, one for the steel of the rebars)
- As *Shell Mat1 Global Nr*, put ***3*** (mechanical properties of material 1 : concrete)
- As *Shell Mat2 Global Nr*, put ***4*** (mechanical properties of material 2 : rebars)

Assign it to the 2 surfaces as shown below:



To display Properties select in the dialog box:

- *Draw->Colors*

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

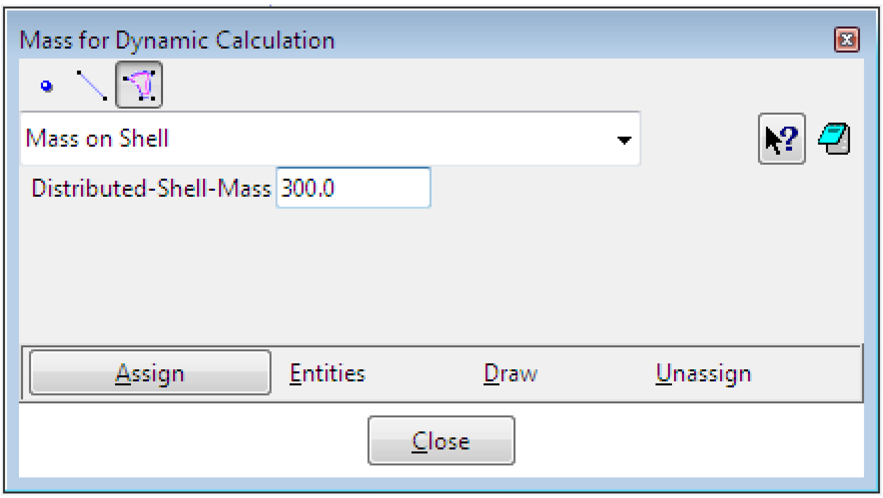
⚠ *Before launching the calculation, don't forget to copy the **slab.tsh** file in the calculation directory !*

5. Assign Mass

The mass have already been assigned to the beam elements in exercise 8. To assign mass on the shell elements defining the slab, select :

- *Data-> Mass*

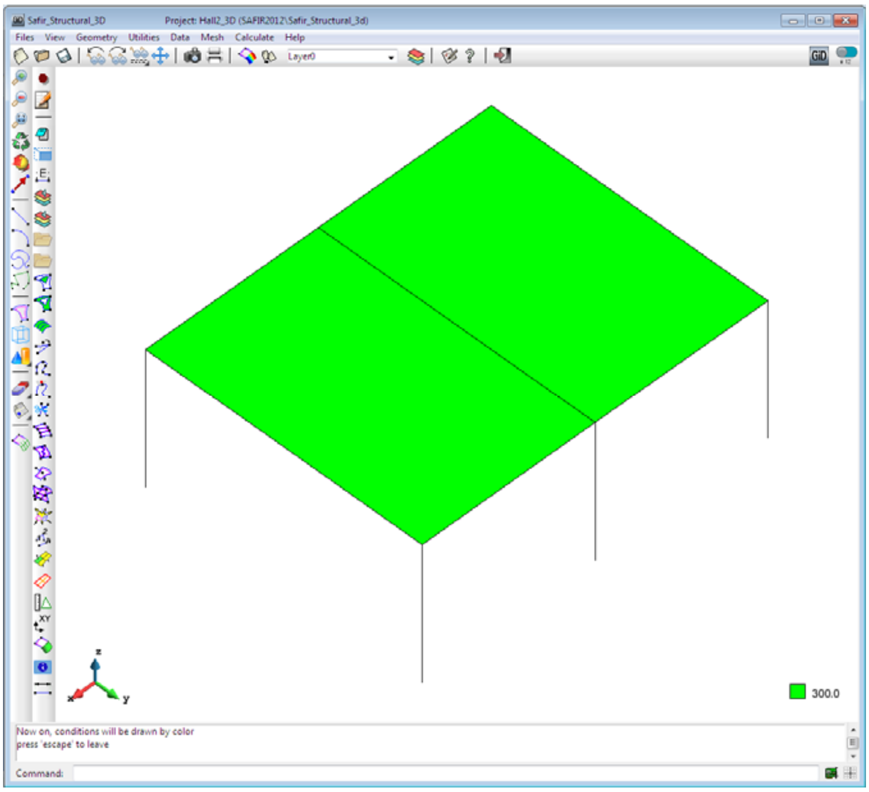
Select the Surface Mass tab and *Assign* 300kg/m² to all the surfaces.



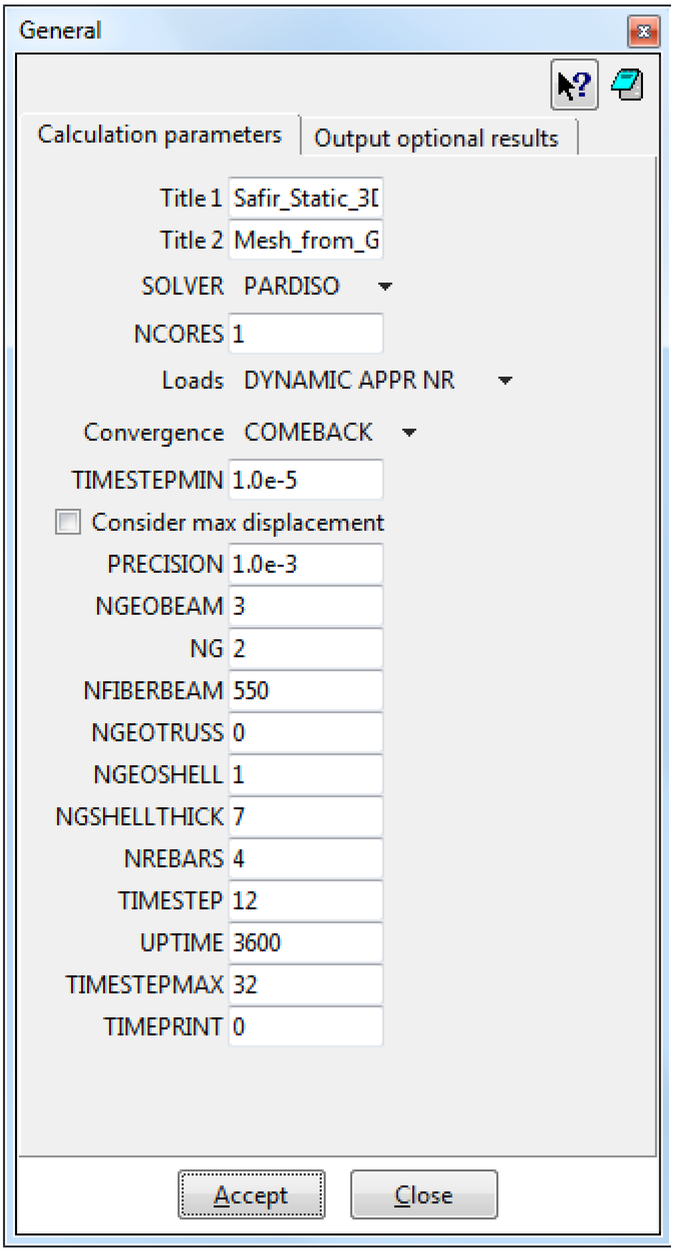
To display mass on shell, select in the dialog box:

- *Draw->Colors*

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



6. Modify Problem Data



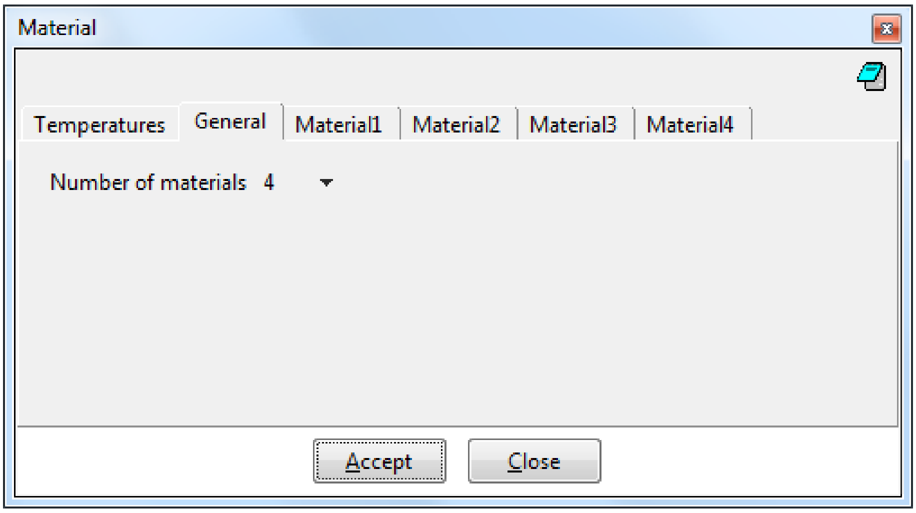
In the *Calculation parameters* tab, fill the cells as shown in the figure hereabove (NGEOBEAM = ***3*** ; NGEOSHELL = ***1*** ; NREBARS = ***4*** ...).

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

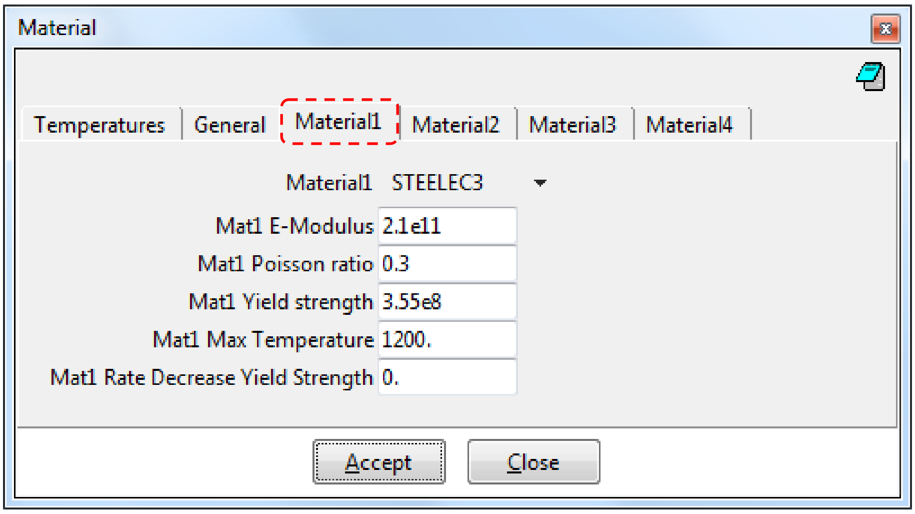
7. Modify the materials properties

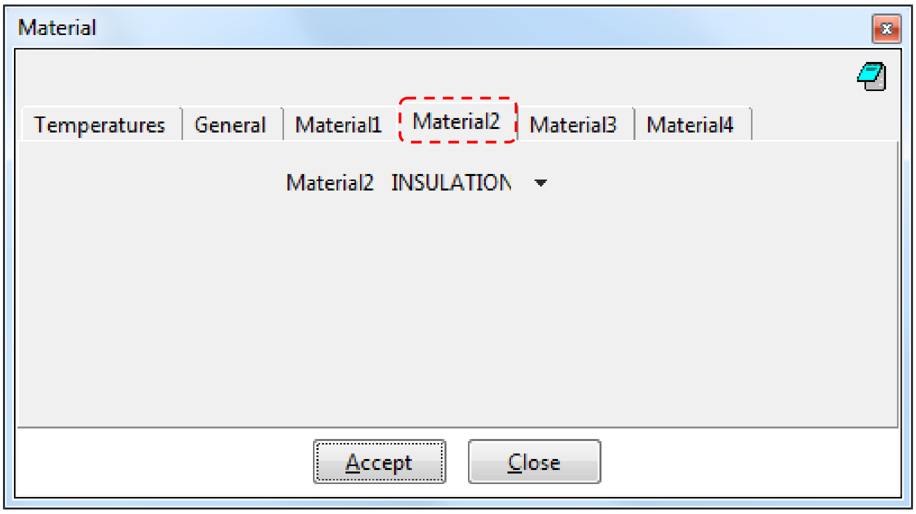
Select from the pull down menu

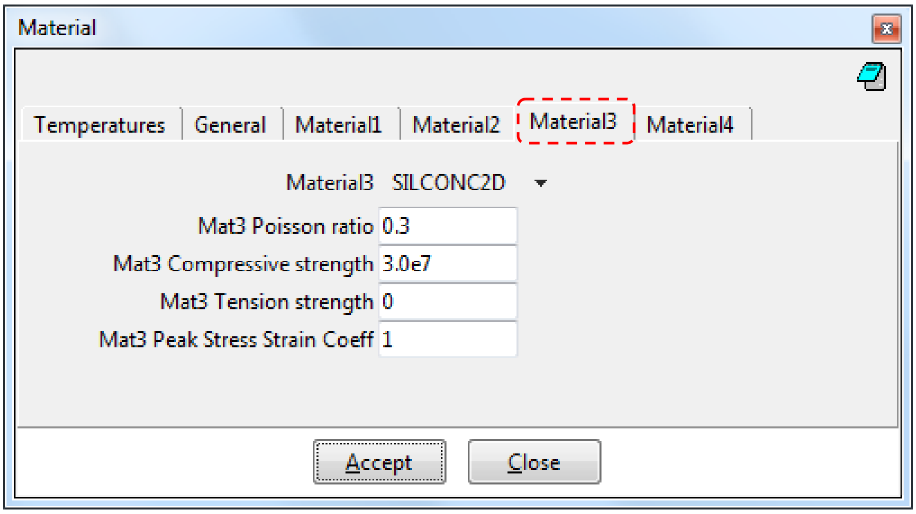
- *Data-> Material*

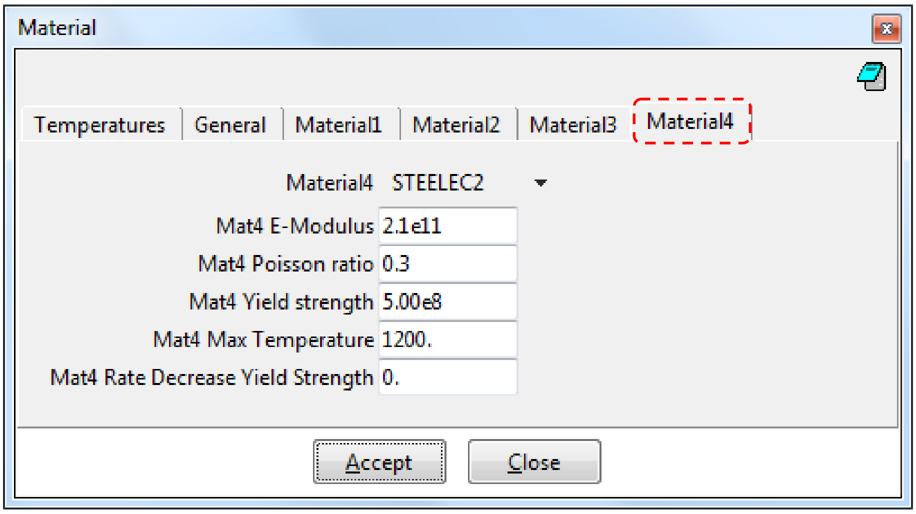


Put *4* as number of materials and complete the different *Material* tabs as shown below.









Click on ***Accept*** to confirm your modifications, then ***Close***.

8. Generate Mesh

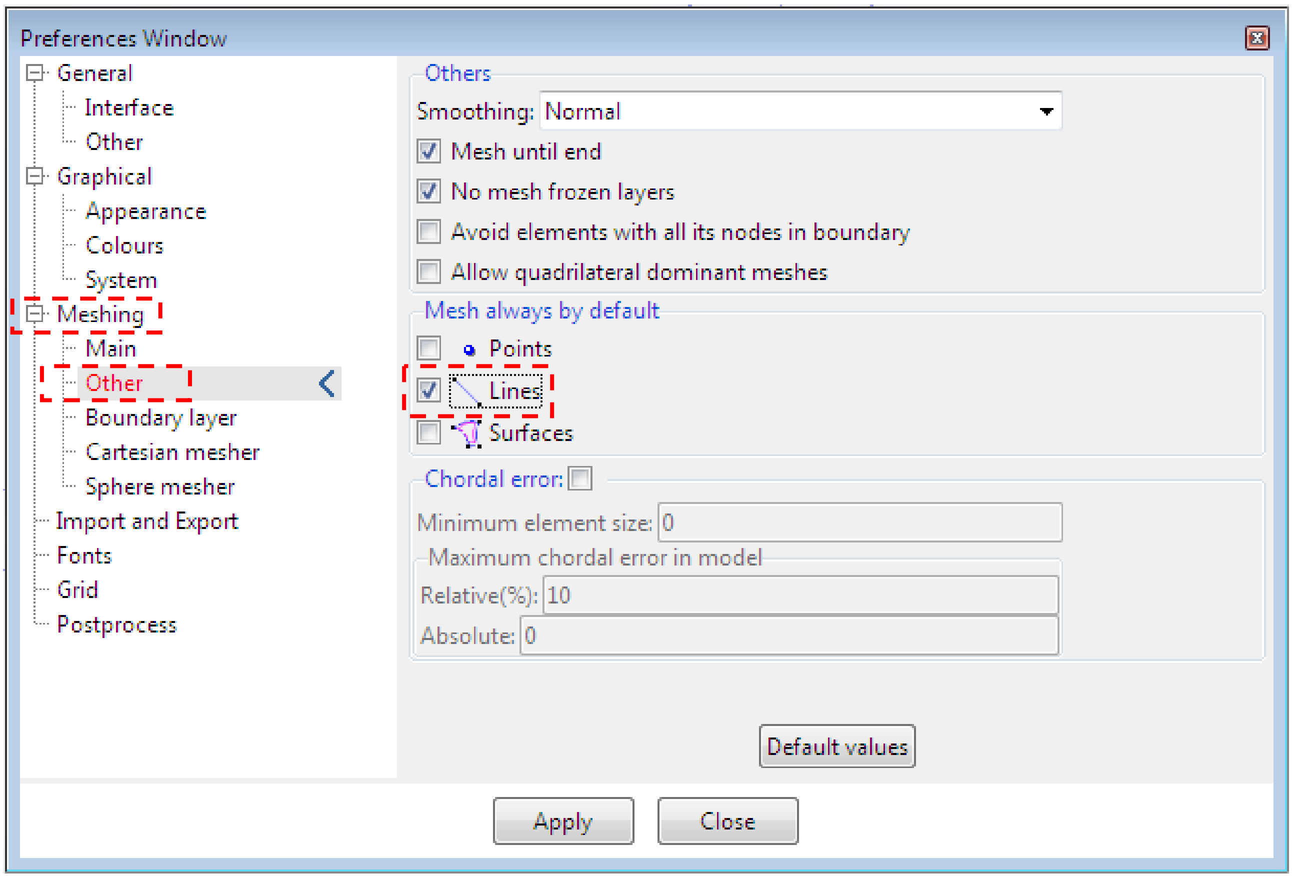
Before meshing, you need to specify the default meshing criteria which consists to start meshing by the beam elements.

Define the default values for the meshing

Select from the pull down menu

- *Utilities->Preferences*

Then go to *Meshing / Other / Mesh always by default* and click the *Lines* option as shown in the following figure.

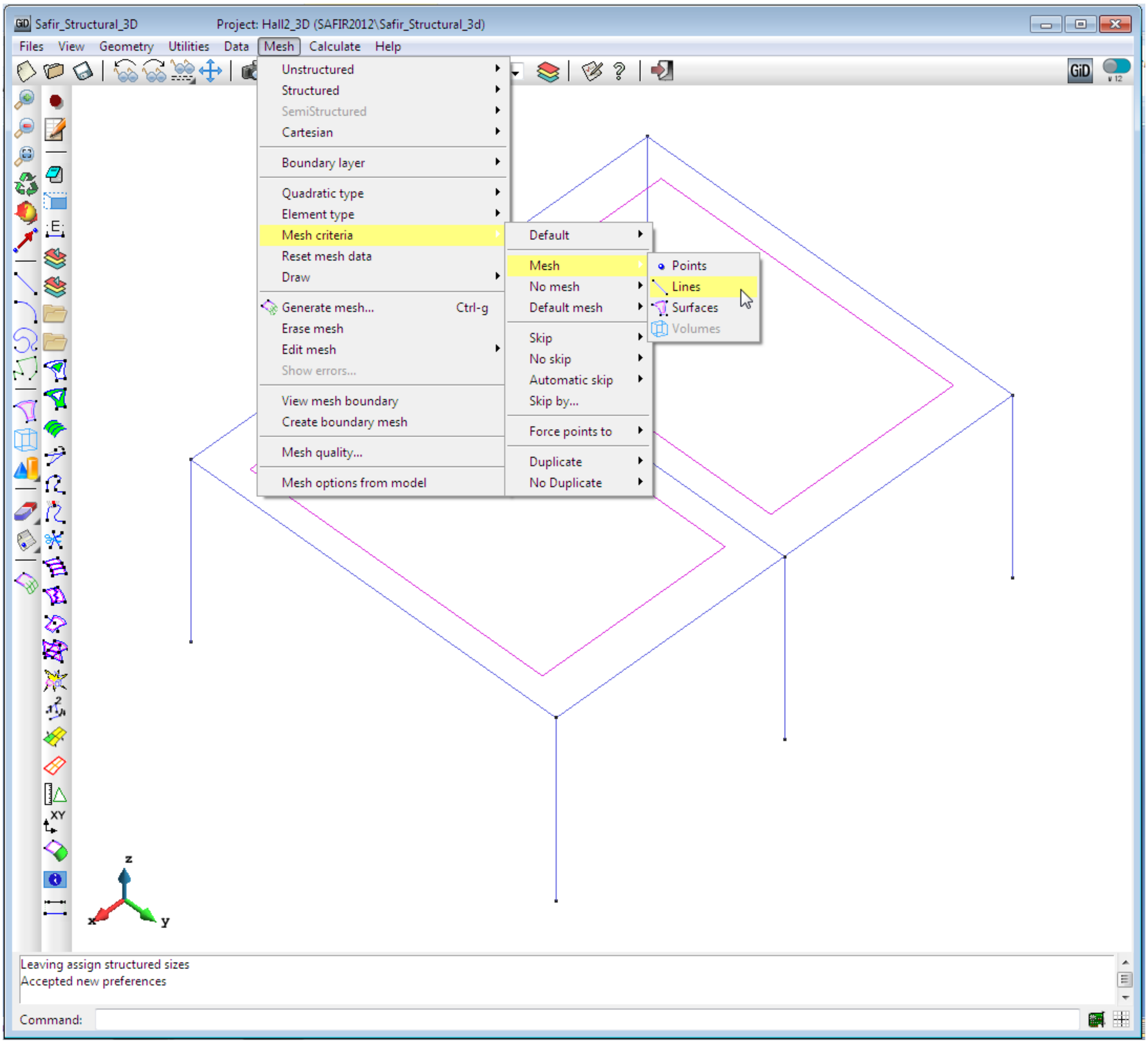


Define the mesh criteria

Select from the pull down menu

- *Mesh->Mesh criteria->Mesh->Lines*

Then select all the beam element lines, then press *[Esc]*.
esc.



Define structured elements to the surfaces.

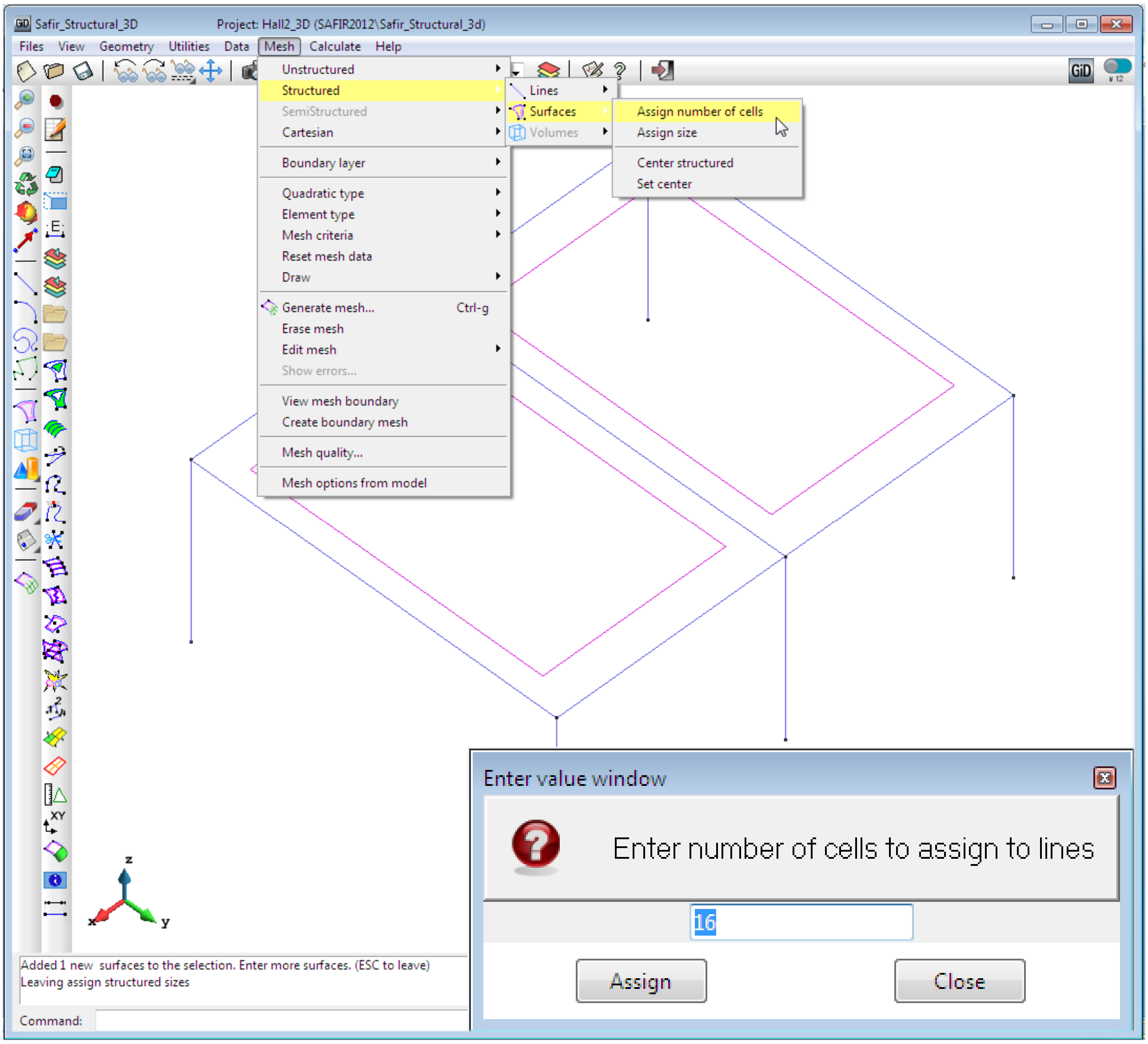
Select from the pull down menu:

- ***Mesh->Structured->Surfaces->Assign number of cells***

Select the surfaces to divide, then *Esc*.

In the dialog box, enter the number of cells (divisions) to assign to lines.

Put 10 divisions to the 5m side and 16 divisions to the 8m side.

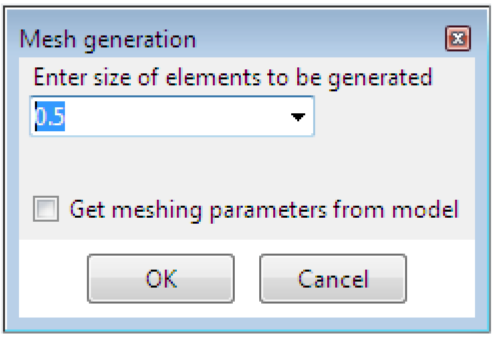


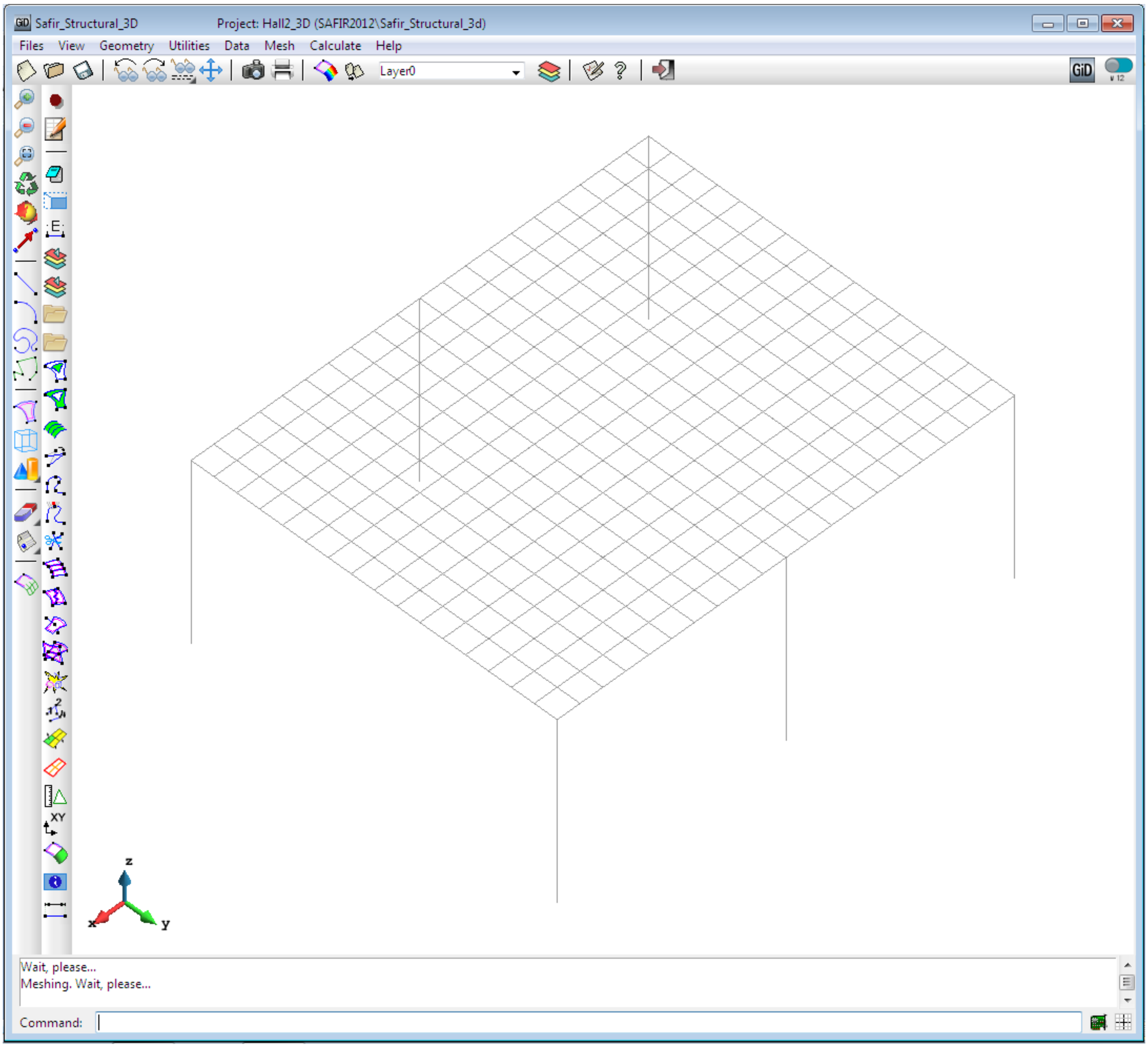
Press *[Esc]* then *Close*.

Then select from the pull down menu:

- ***Mesh->Generate mesh***

In the dialog box, enter 0.5m as size of elements to be generated.





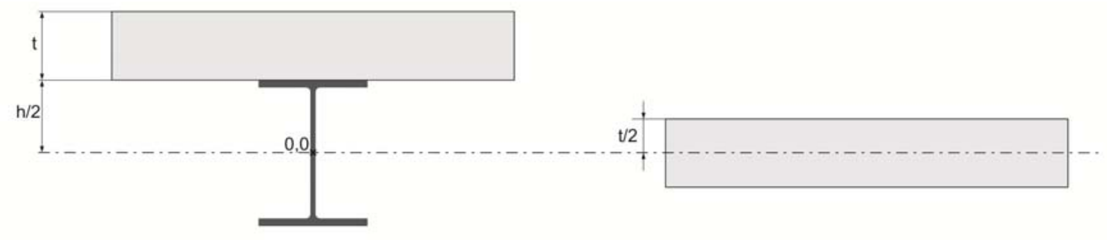
9. Create sections

In this exercise, you have to use exactly the same .TEM files than those used in Exercise 8 for beams and columns.

For the concrete slab, use the .TSH file realised in the Exercise 4.

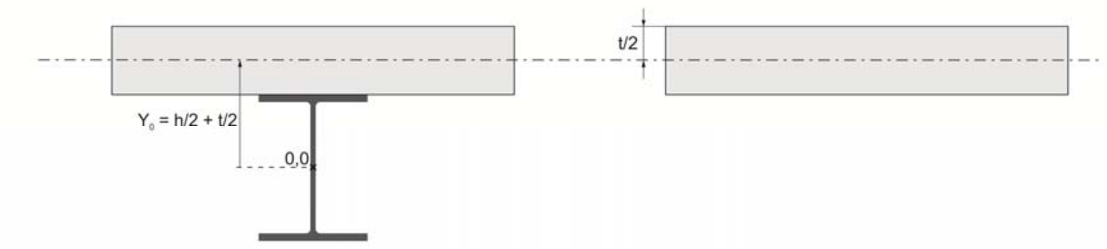
10. Modify the node line in .TEM files

By using the *Cross-Section* option to create the section for the BEAM elements (.tem files), GiD always put the center of the 0,0 plan coordinates to the center of the steel profile. Similarly, in the section for SHELL elements (.tsh file), the symmetry axis of the section is put at mid-thickness of the section (see figure below).



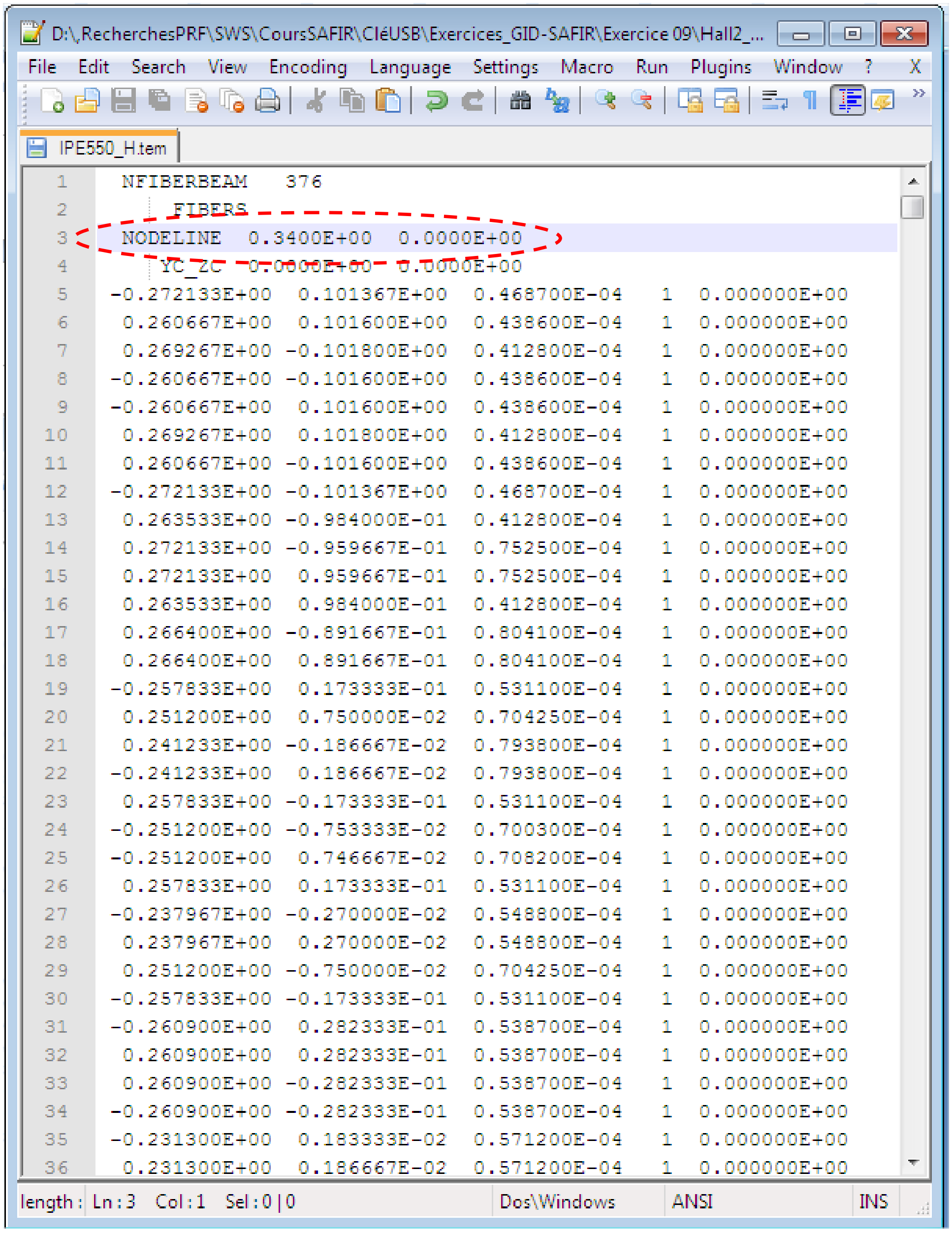
BEAM element cross section SHELL element cross section

So you need to align the slab of the beam element cross-section with the slab of the shell element cross-section. Then displace the local center of the composite cross-section as shown in the following figure.



In our exercise, the beam is an IPE550 with a height of 550mm and the slab is 130mm thick. Therefore, you need to displace the global center Y_0 = 550/2 + 130/2 = 340mm.

Open the file *.tem* corresponding to the beam element and modify the Y_0 value of the NODELINE as shown in the following figure (the two numbers are respectively the Y0 and the Z0 coordinate of the global center).



```
1   NFIBERBEAM    376
2       FIBERS
3   NODELINE   0.3400E+00   0.0000E+00
4     YC_ZC    0.0000E+00   0.0000E+00
5  -0.272133E+00  0.101367E+00  0.468700E-04   1  0.000000E+00
6   0.260667E+00  0.101600E+00  0.438600E-04   1  0.000000E+00
7   0.269267E+00 -0.101800E+00  0.412800E-04   1  0.000000E+00
8  -0.260667E+00 -0.101600E+00  0.438600E-04   1  0.000000E+00
9  -0.260667E+00  0.101600E+00  0.438600E-04   1  0.000000E+00
10  0.269267E+00  0.101800E+00  0.412800E-04   1  0.000000E+00
11  0.260667E+00 -0.101600E+00  0.438600E-04   1  0.000000E+00
12 -0.272133E+00 -0.101367E+00  0.468700E-04   1  0.000000E+00
13  0.263533E+00 -0.984000E-01  0.412800E-04   1  0.000000E+00
14  0.272133E+00 -0.959667E-01  0.752500E-04   1  0.000000E+00
15  0.272133E+00  0.959667E-01  0.752500E-04   1  0.000000E+00
16  0.263533E+00  0.984000E-01  0.412800E-04   1  0.000000E+00
17  0.266400E+00 -0.891667E-01  0.804100E-04   1  0.000000E+00
18  0.266400E+00  0.891667E-01  0.804100E-04   1  0.000000E+00
19 -0.257833E+00  0.173333E-01  0.531100E-04   1  0.000000E+00
20  0.251200E+00  0.750000E-02  0.704250E-04   1  0.000000E+00
21  0.241233E+00 -0.186667E-02  0.793800E-04   1  0.000000E+00
22 -0.241233E+00  0.186667E-02  0.793800E-04   1  0.000000E+00
23  0.257833E+00 -0.173333E-01  0.531100E-04   1  0.000000E+00
24 -0.251200E+00 -0.753333E-02  0.700300E-04   1  0.000000E+00
25 -0.251200E+00  0.746667E-02  0.708200E-04   1  0.000000E+00
26  0.257833E+00  0.173333E-01  0.531100E-04   1  0.000000E+00
27 -0.237967E+00 -0.270000E-02  0.548800E-04   1  0.000000E+00
28  0.237967E+00  0.270000E-02  0.548800E-04   1  0.000000E+00
29  0.251200E+00 -0.750000E-02  0.704250E-04   1  0.000000E+00
30 -0.257833E+00 -0.173333E-01  0.531100E-04   1  0.000000E+00
31 -0.260900E+00  0.282333E-01  0.538700E-04   1  0.000000E+00
32  0.260900E+00  0.282333E-01  0.538700E-04   1  0.000000E+00
33  0.260900E+00 -0.282333E-01  0.538700E-04   1  0.000000E+00
34 -0.260900E+00 -0.282333E-01  0.538700E-04   1  0.000000E+00
35 -0.231300E+00  0.183333E-02  0.571200E-04   1  0.000000E+00
36  0.231300E+00  0.186667E-02  0.571200E-04   1  0.000000E+00
```

On the 3th line you can see the line “NODELINE”. The two 0.0000E+00 numbers are respectively the Y0 and the Z0 coordinate in meter of the global center.

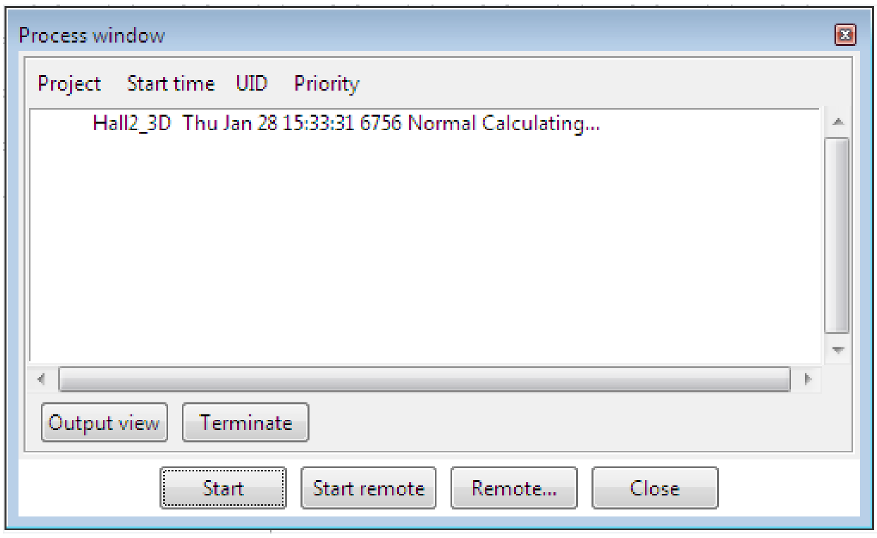
Modify the other beam section if necessary.

11. Start the calculation:

Before starting the calculation, be sure that all the *.tem* and *.tsh* files are in the directory “Hall2_3D.gid” (IPE550_C.tem ; IPE550_H.tem ; HEB220_C.tem ; Slab.tsh).

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

› ***Calculate->Calculate window***



Click on the ***Start*** button then on the Output view button to see the calculation progress.