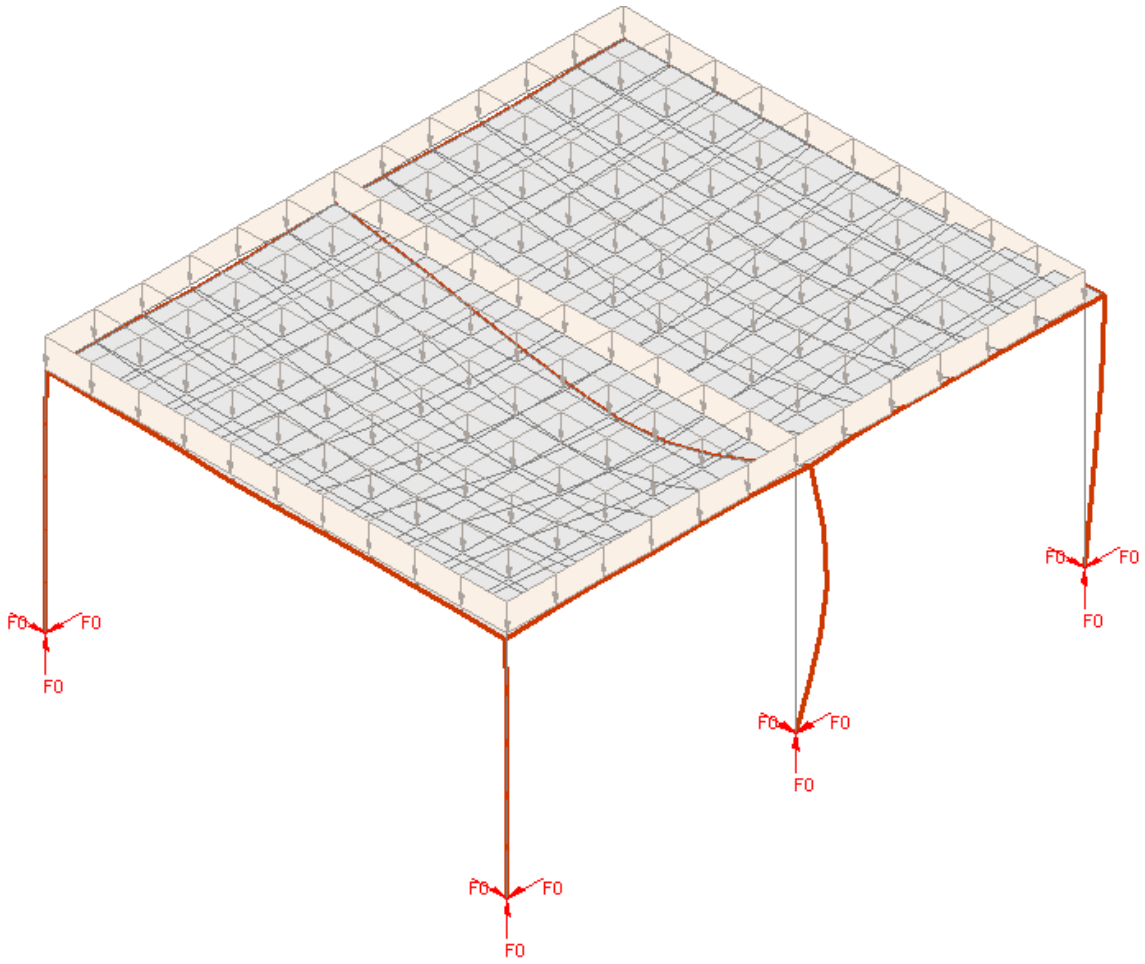
Tutorial for GID-SAFIR 3D Structural Analysis

Exercise n°9 Hall 2 3D



This example is a 3d frame with the following system geometry and beam profiles. There are 3 temperature zones, .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).

1. Create section

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

Furthermore you have to create 3 .TSH files: ***ShB1.tsh, ShB2.tsh*** and ***ShB0.tsh*** you have to realize them exactly as explain in exercise 4, with a shell thickness of 0.2 m. The only difference is that you will have to replace ISO fire condition with temperature curve B1 and B2, as explain in exercise 3.

2. Open the GiD-Safir project Hall1_3D.gid (Exercise n°8)

To save the project select (or use icon on the left):

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

⚠ *If Caps lock is active on your keyboard, shortcut don't work*

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

GiD creates a directory with the name ***Hall2_3D.gid***

GiD creates a number of system files in this directory.

When you start the SAFIR calculation the SAFIR ***.IN*** and ***.OUT*** file are placed here. Also before starting a calculation all ***.TEM*** files must be placed to this directory.

3. Modify the system geometry

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

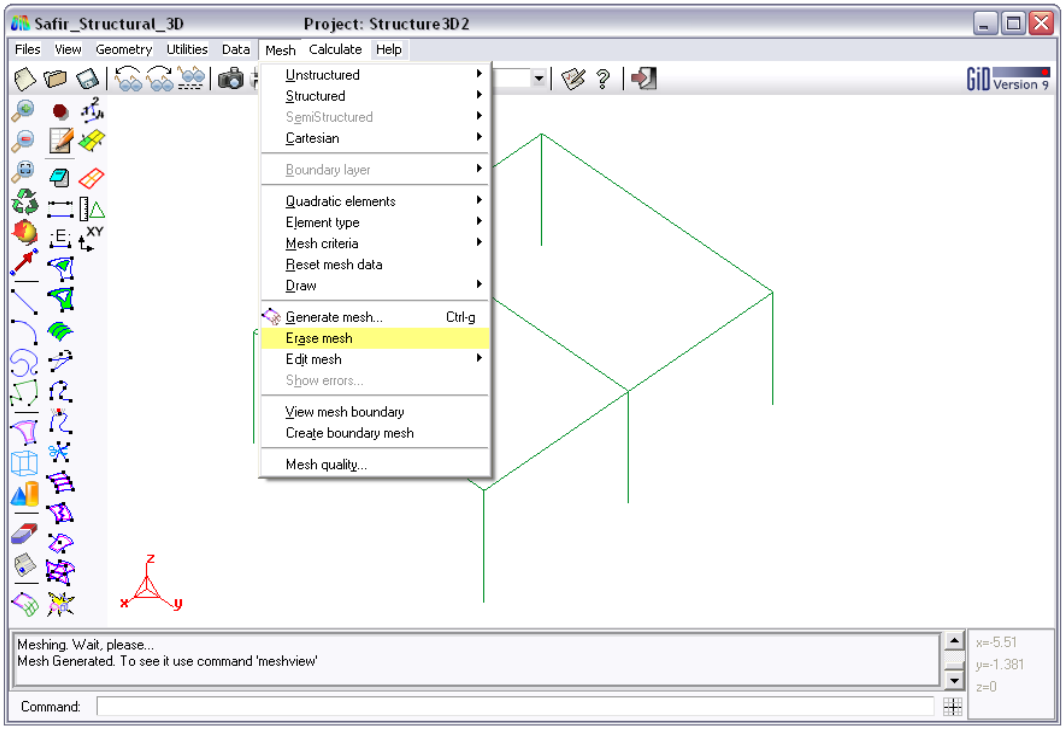
- ***View->Rotate->isometric***

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

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

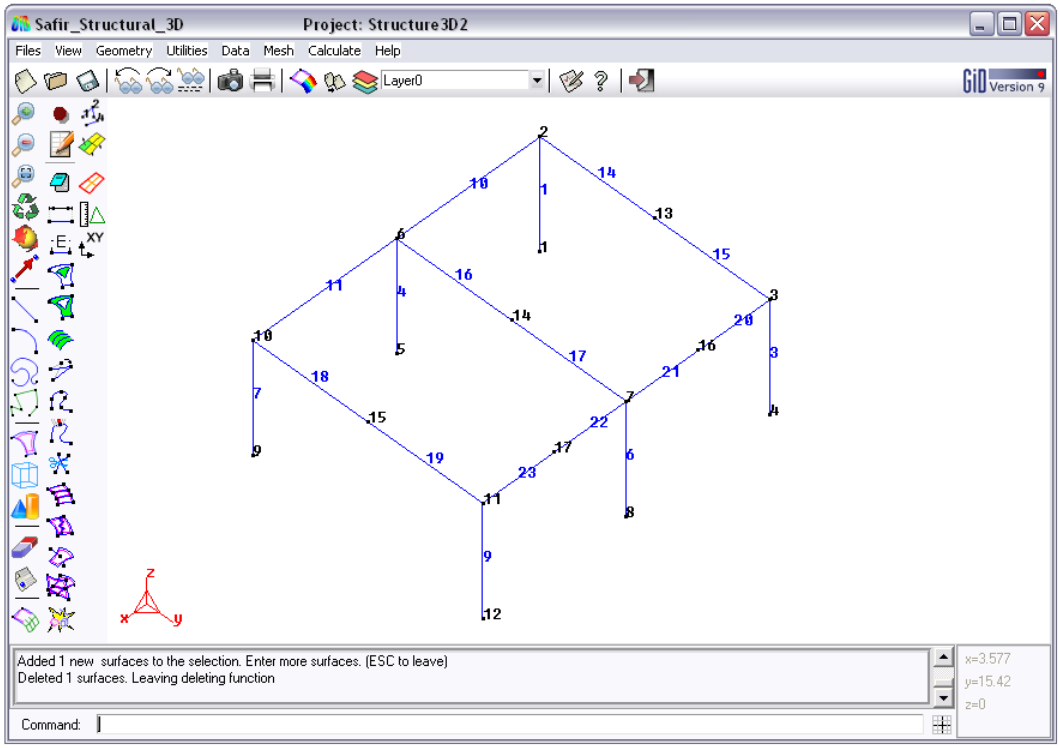
To modify am existent file, you have to first erase old mesh:

- ***Mesh->Erase Mesh***



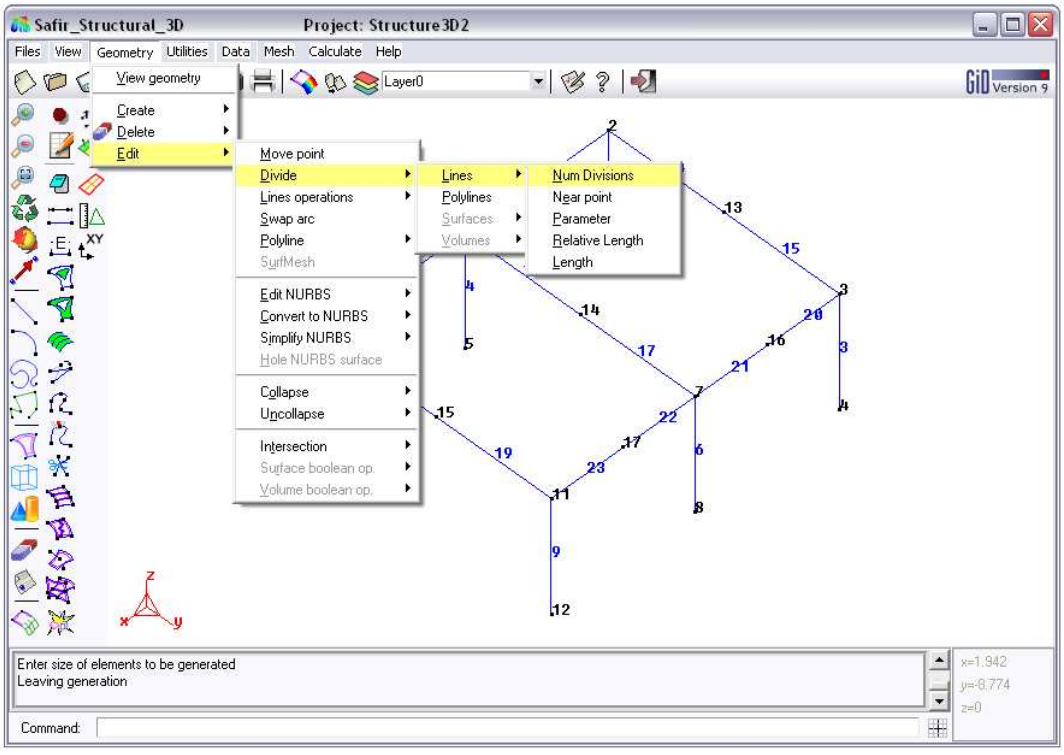
From the pull down menu select:

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



Select from the pull down menu:

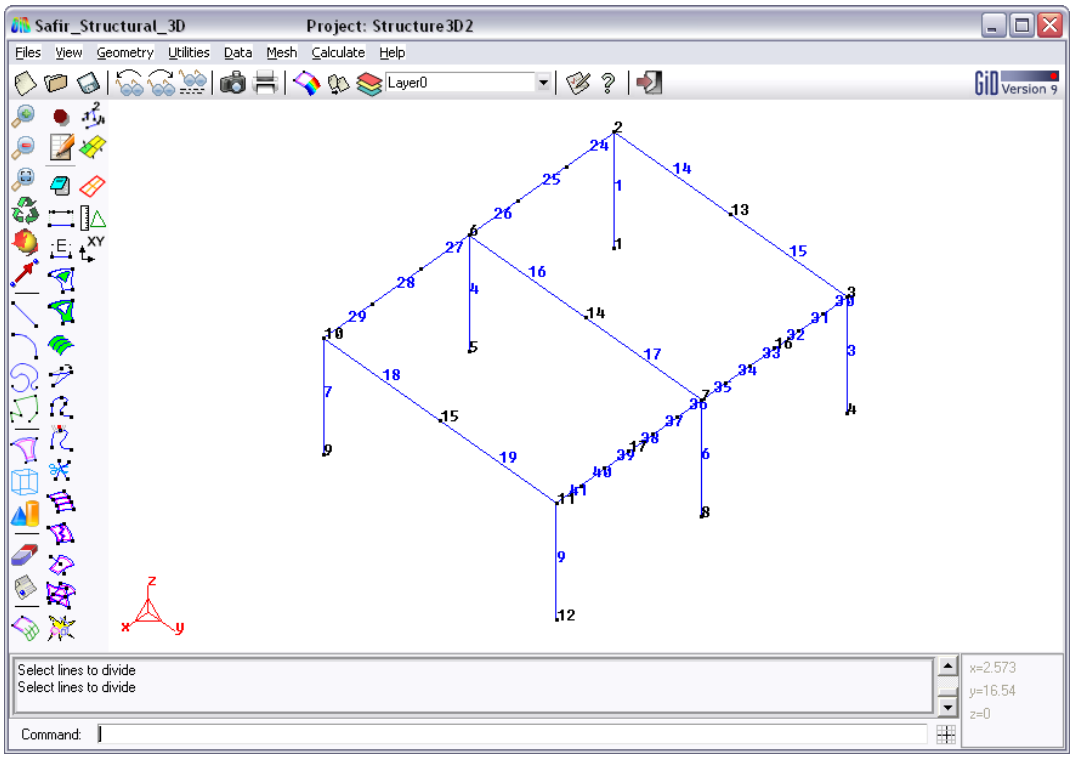
- ***Geometry-> Edit-> Divide-> Lines-> Num Divisions***



Enter 3 as number of divisions

Select lines 10; 11; 20; 21; 22 and 23 (line are in blue and nodes are in black)

Press *[Esc]*



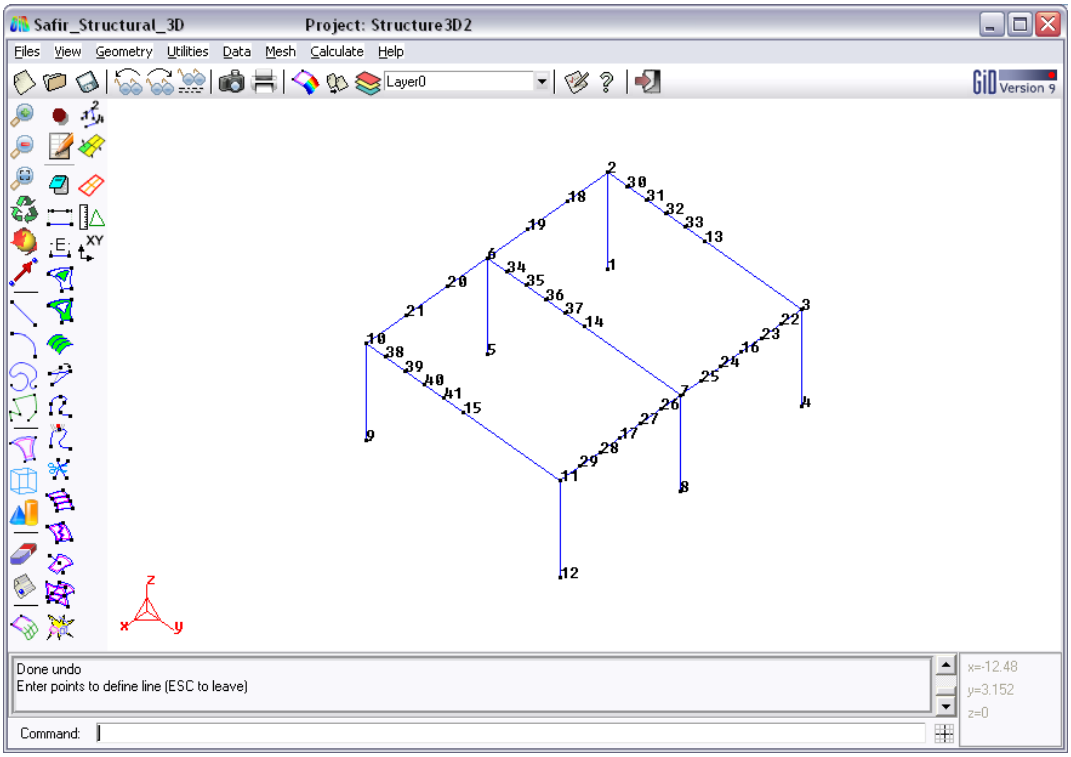
Do the same operation in order to divide lines 14; 16; and 18 in 5 parts

From the pull down menu select:

- ***View-> Label-> Off***

Then, select:

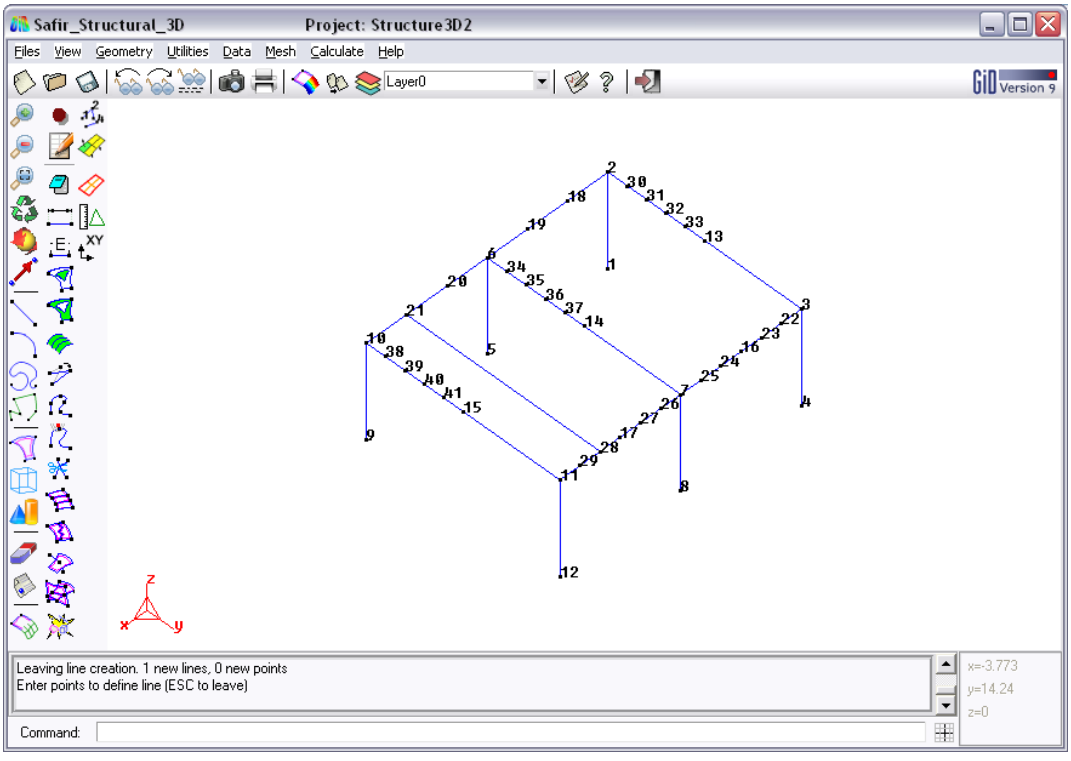
- ***View-> Label-> All in -> Points [ ]***



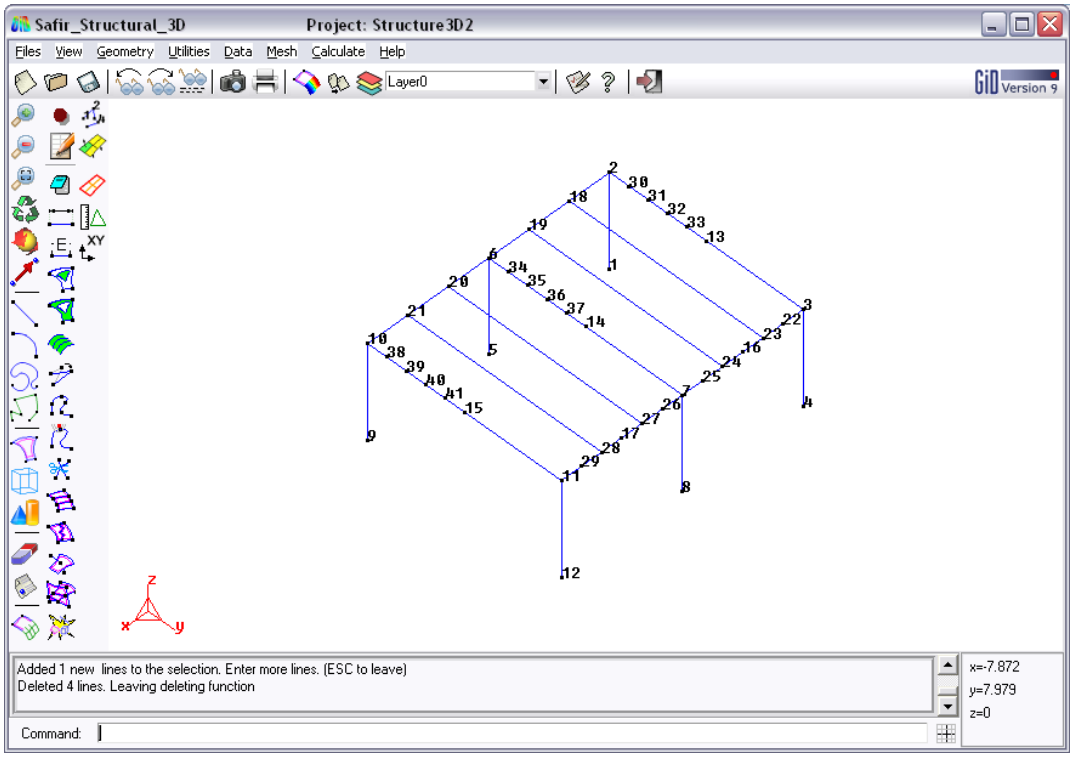
Select:

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

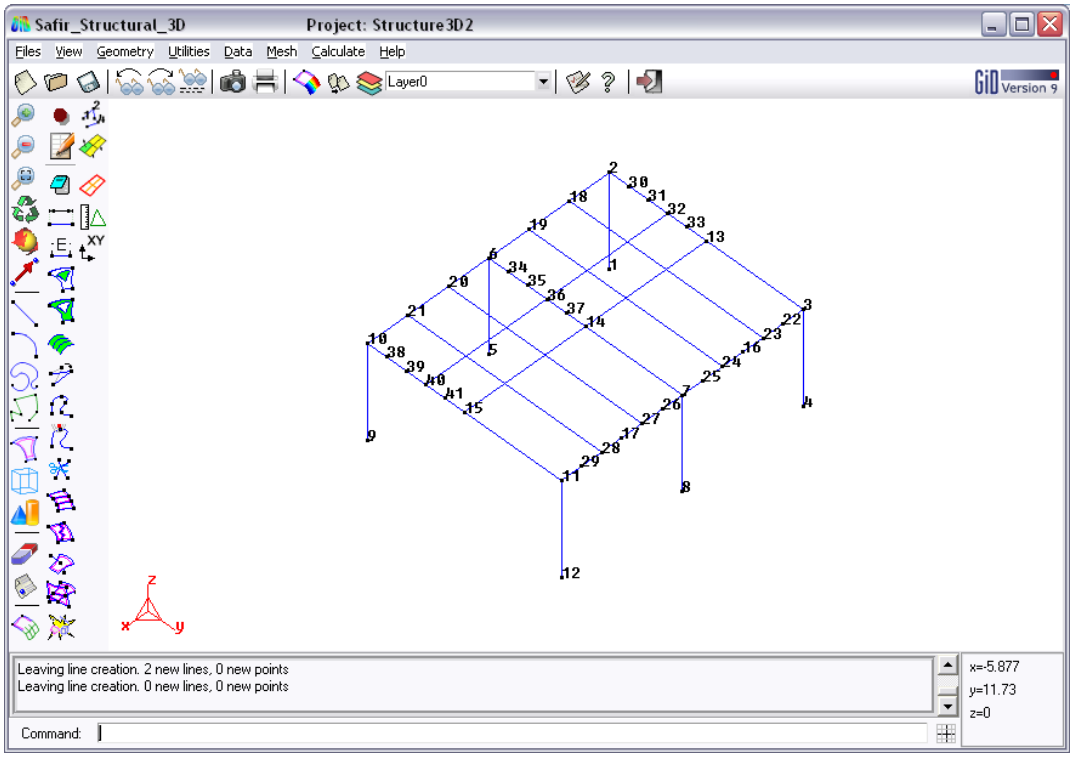
Press *[Ctrl + a]* and pick points 21 and 28 and complete with *[Esc]*



Do the same operation with points 20 and 27; 19 and 24; 18 and 23

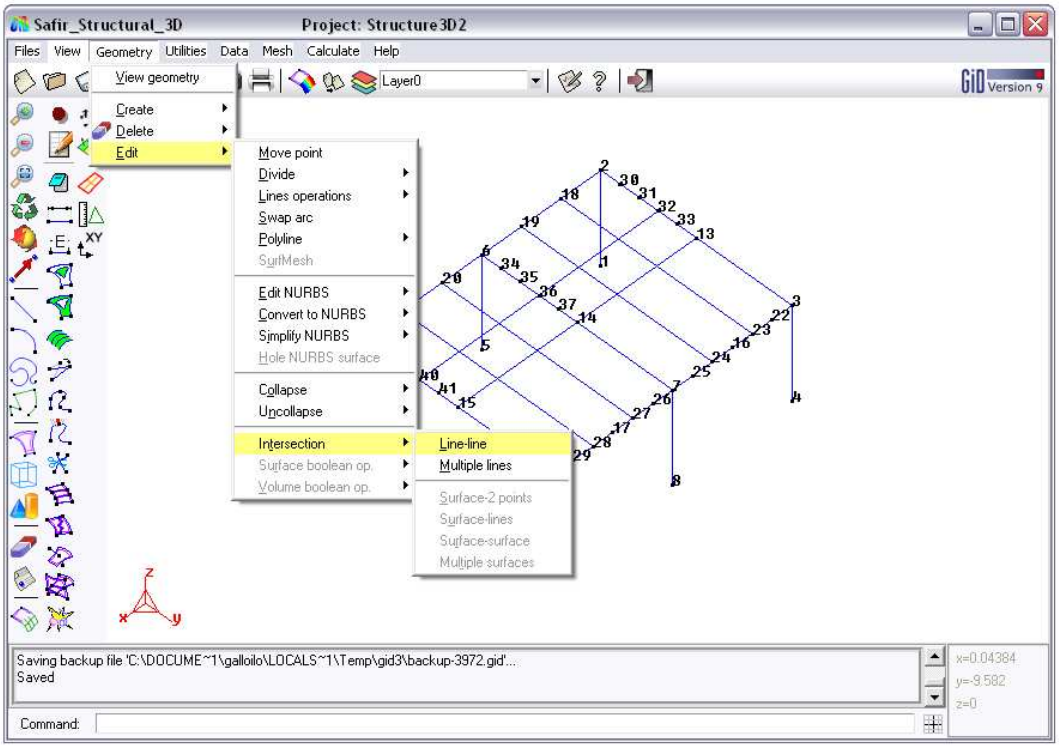


Continue with straight line select mode and select points 15 and 14; 14 and 13; 40 and 36; 36 and 32.

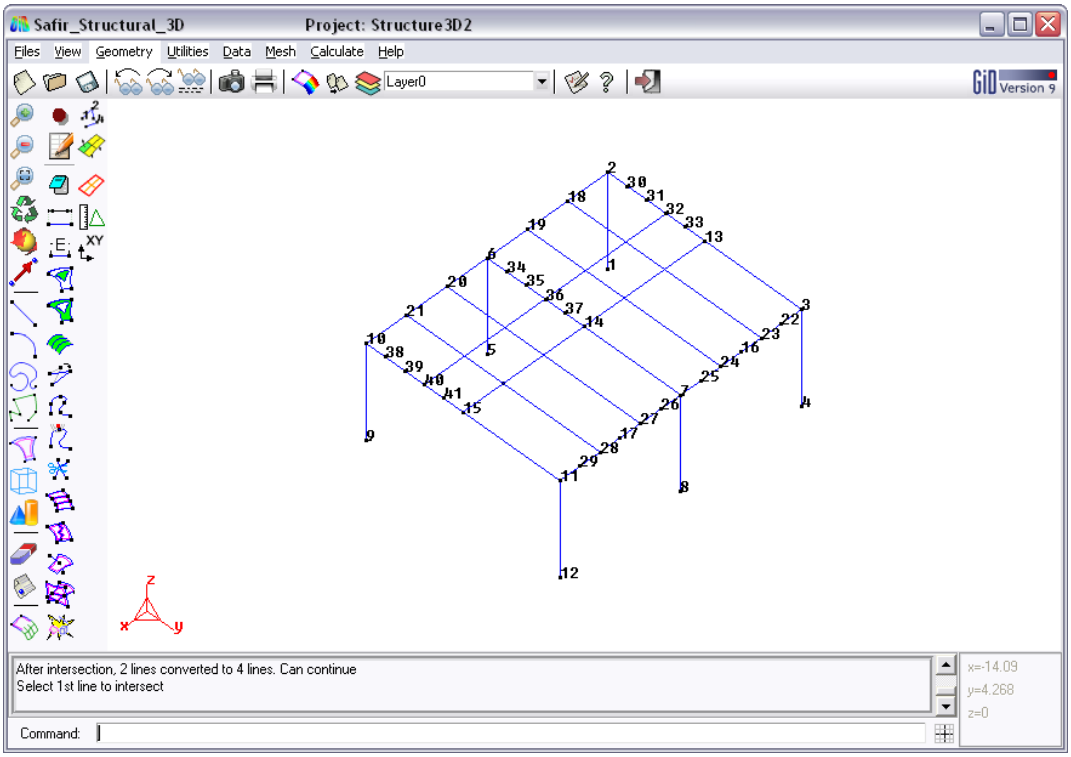


Select:

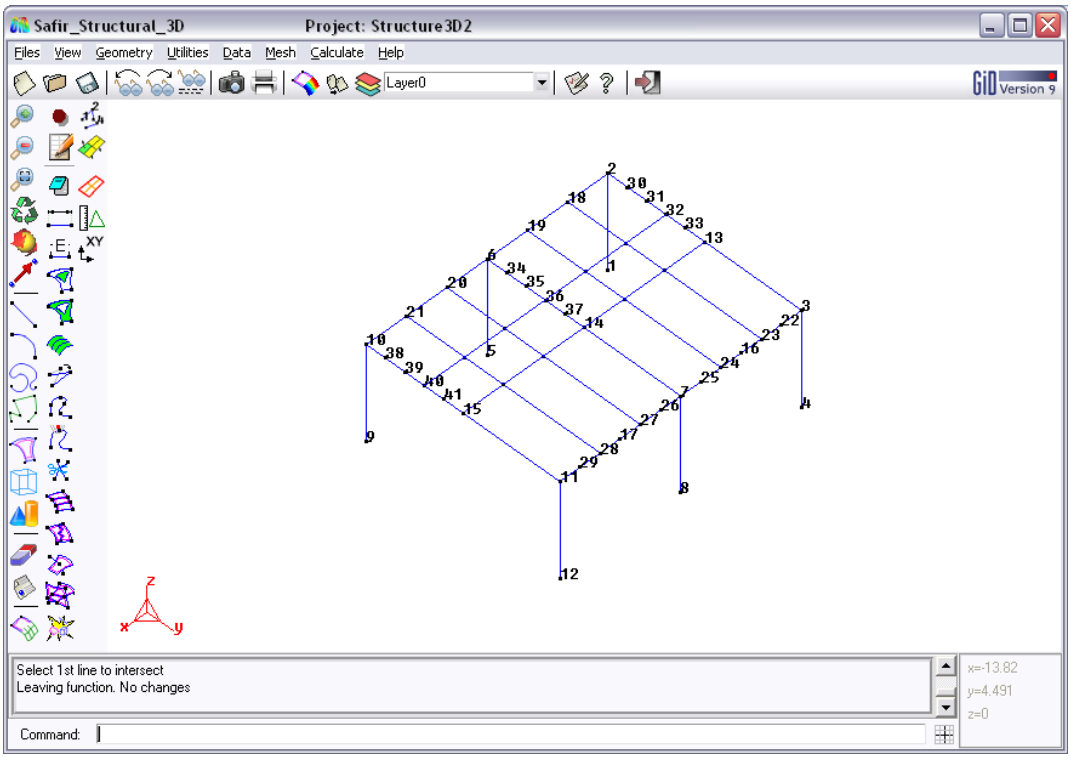
Geometry->Edit -> Intersection-> Line-line



Select line 21-28 and 15-14 and press *[Esc]*

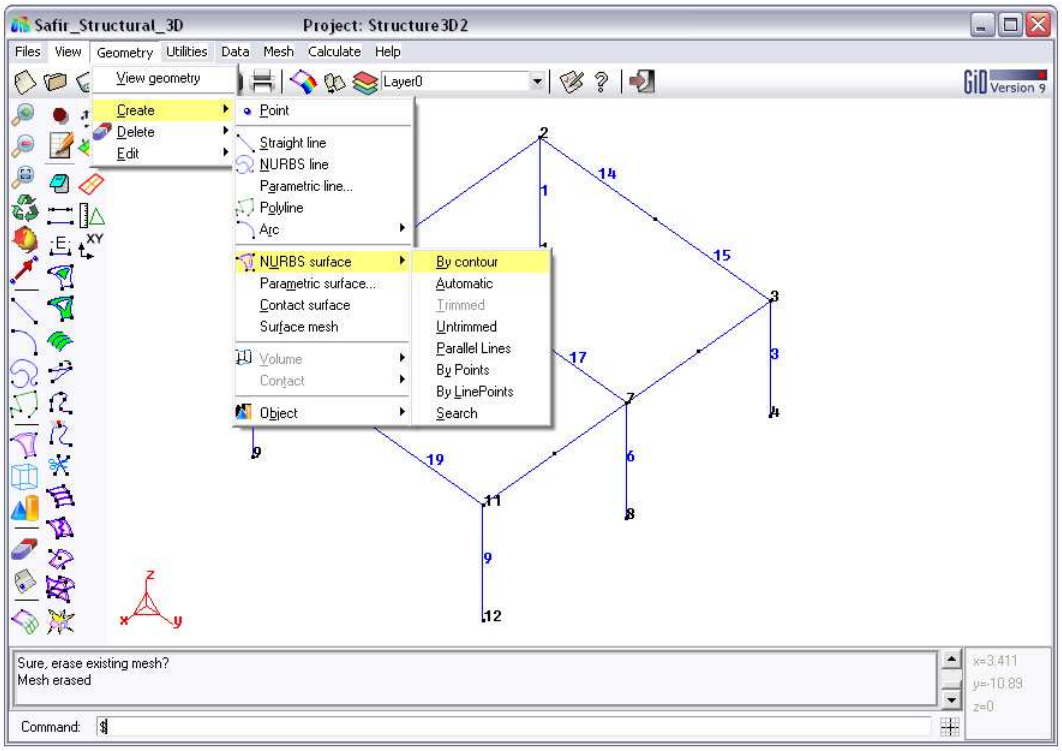


Do the same operation with line 15-14 and 20-27; 40-36 and 20-27 ; 40-36 and 21-28; 14-13 and 19-24; 14-13 and 18-23; 36-32 and 19-24; 36-62 and 18-23

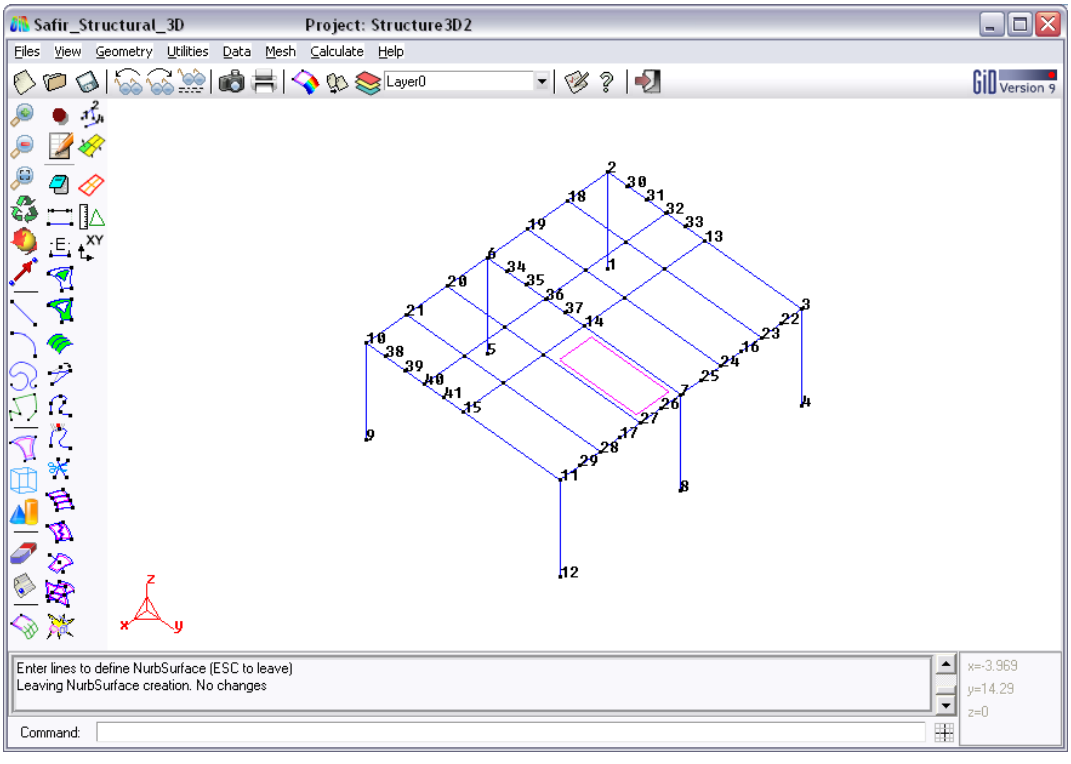


Select from the pull down menu:

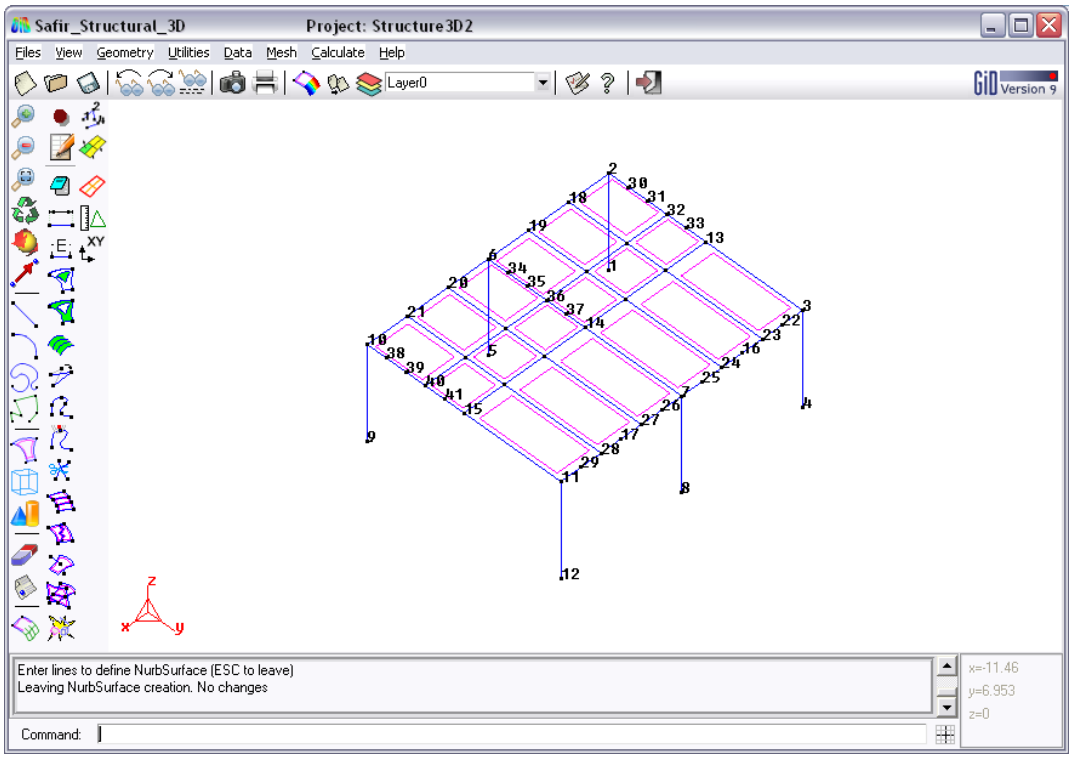
- ***Geometry-> Create-> NURBS surface-> By contour***



Select all contour line of the rectangle as shown below and press ***[Esc]***:



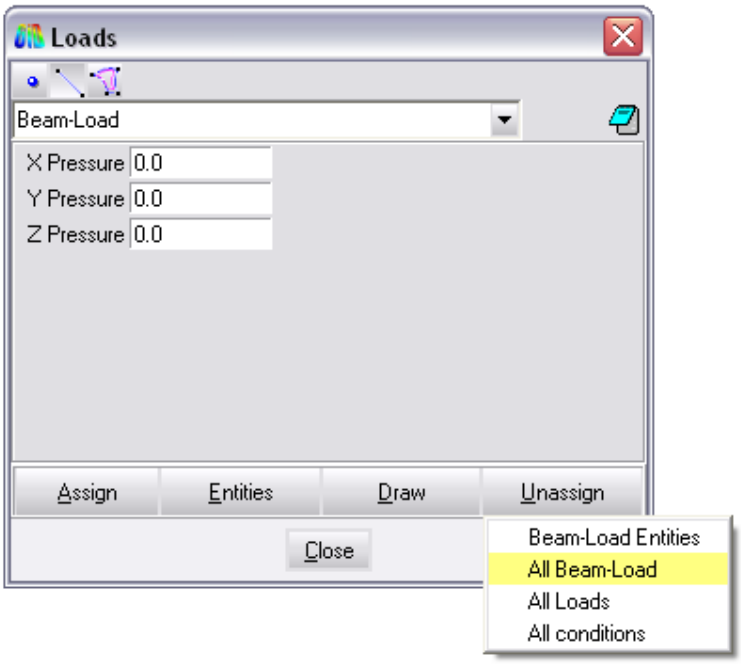
Do the same operation in order to create 9 surfaces per floor as shown in the figure below



4. Modify loads

Select from the pull down menu:

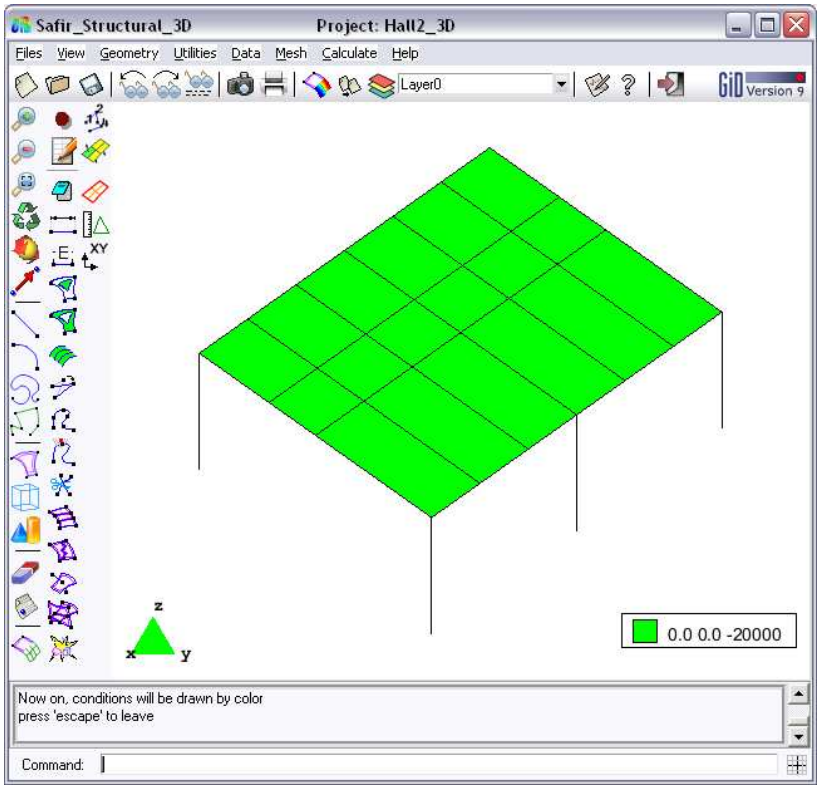
- ***Data-> Loads***



Select the beam tab on ***Unassign*** and select Point-Load ***Entities***

Select the surface mode [] and put -20000 N as Z Pressure

Assign this pressure to all surfaces



To display loads select in the dialog box:

- ***Draw->Colors***

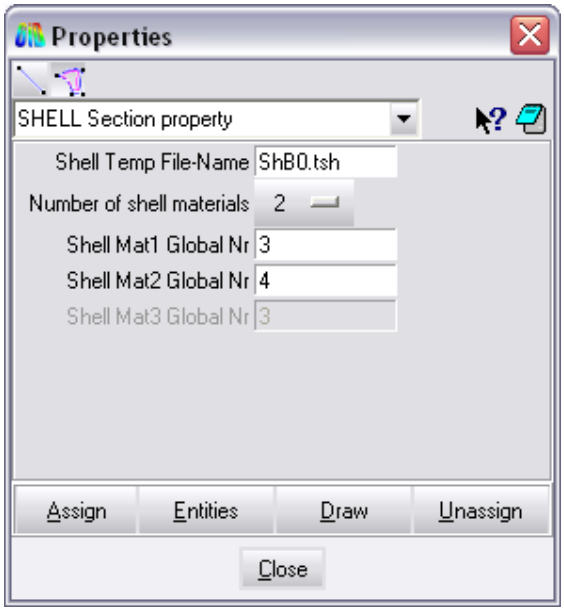
Press ***Finish*** or ***[Esc]*** to leave this view mode

5. Assign .TSH files

Select from the pull down menu:

- ***Data-> Properties***

Select Surface properties []



Put

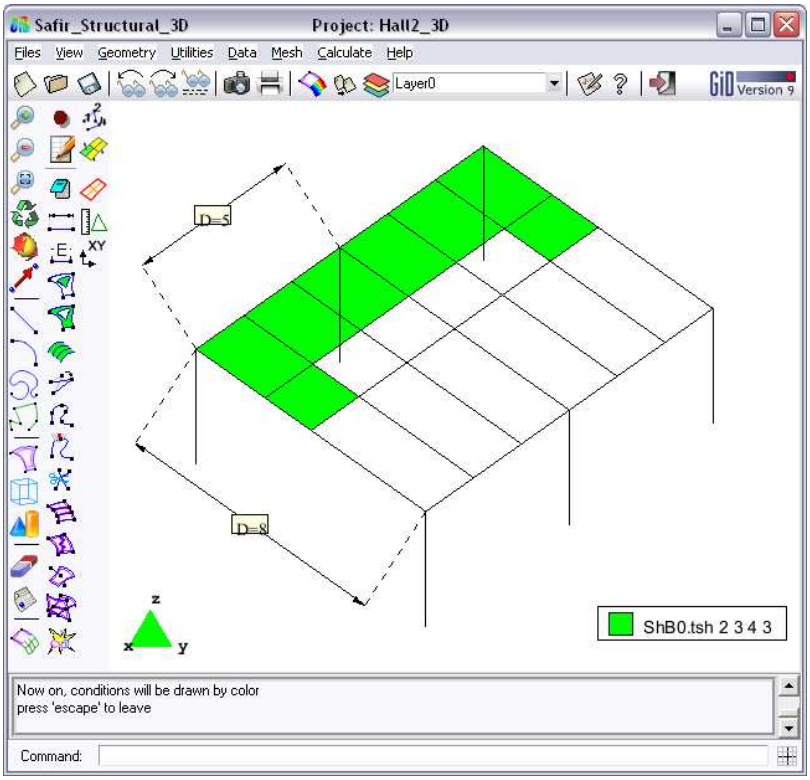
As File-Name, put ***ShB0.tsh***

A number of materials of ***2***

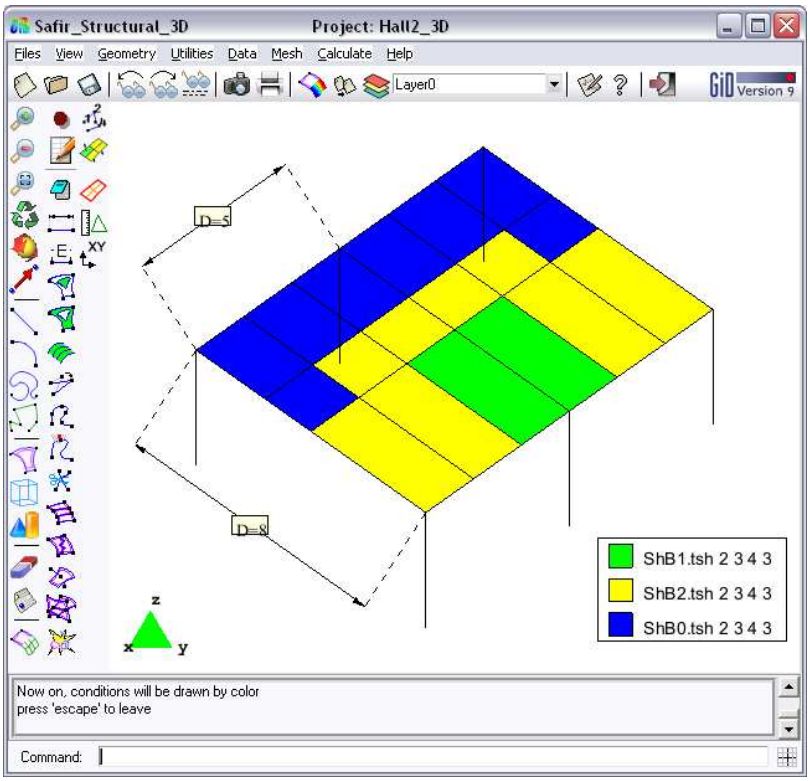
Shell Mat1 Global Nr = ***3***

Shell Mat1 Global Nr = ***4***

Assign it as shown below:



Do the same operation with files ***ShB1.tsh*** and ***ShB2.tsh***



To display Properties select in the dialog box:

- ***Draw->Colors***

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

Now you have to open the formation GiD-Safir file and the Hall2_3D.gid file

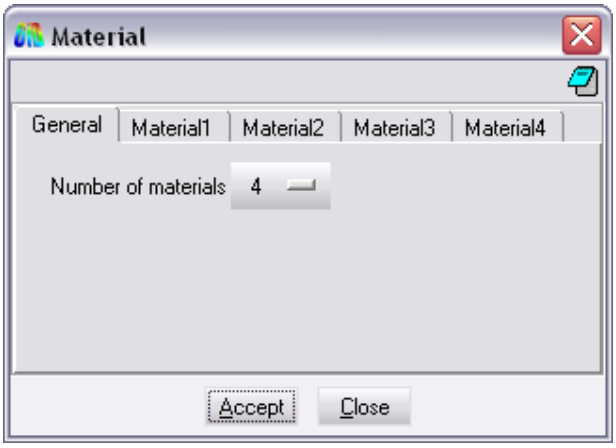
Select the 3 .TSH files (ShB0.TSH, ShB1.TSH and ShB2.TSH) from the tutorial file and copy them into the Hall2_3D.gid file you created

Do the same operation to select all the 6 .TEM files in the Hall2_3D.gid file

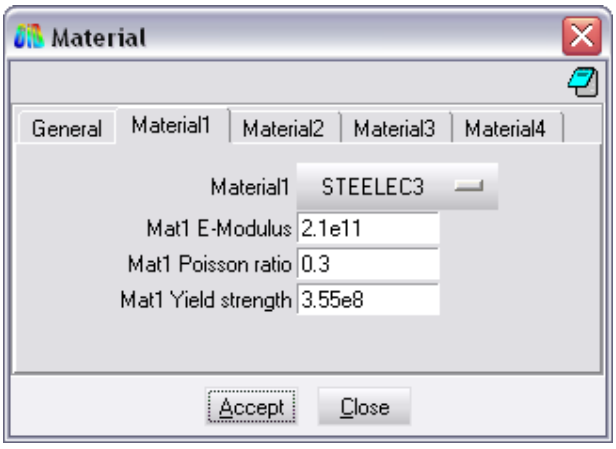
6. Modify Materials

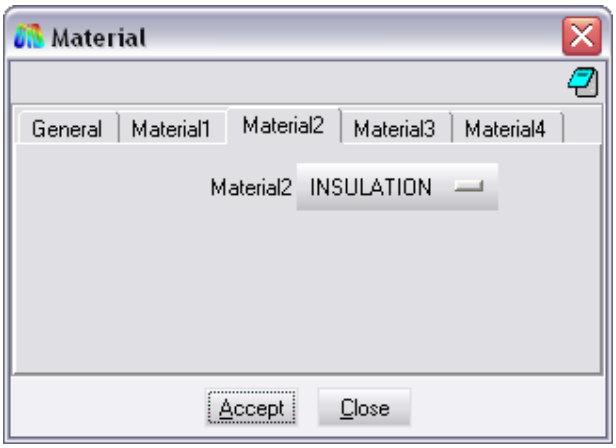
Select from the pull down menu

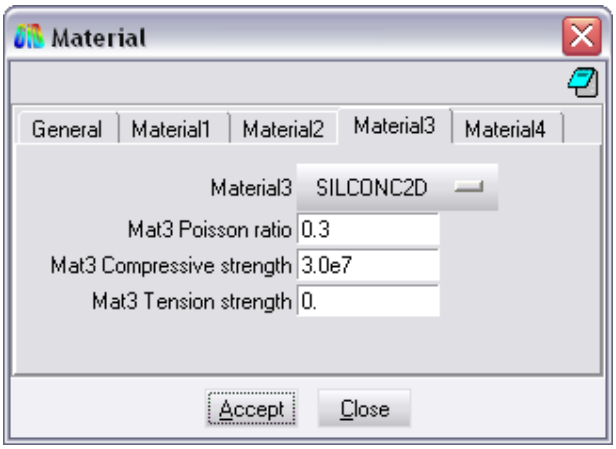
- ***Data-> Material***

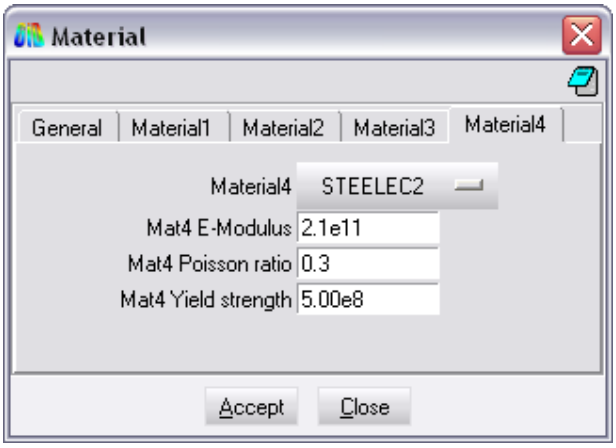


Put 4 as number of materials and complete Material tabs as shown below









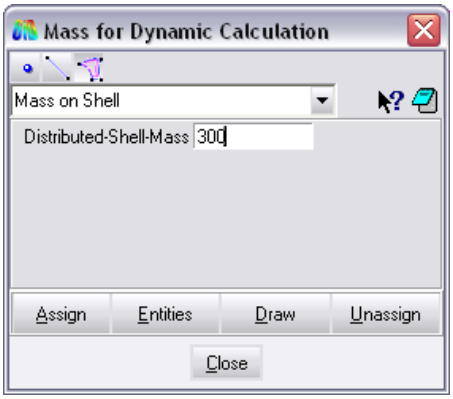
Click on ***Accept*** to confirm your modifications

7. Assign Mass

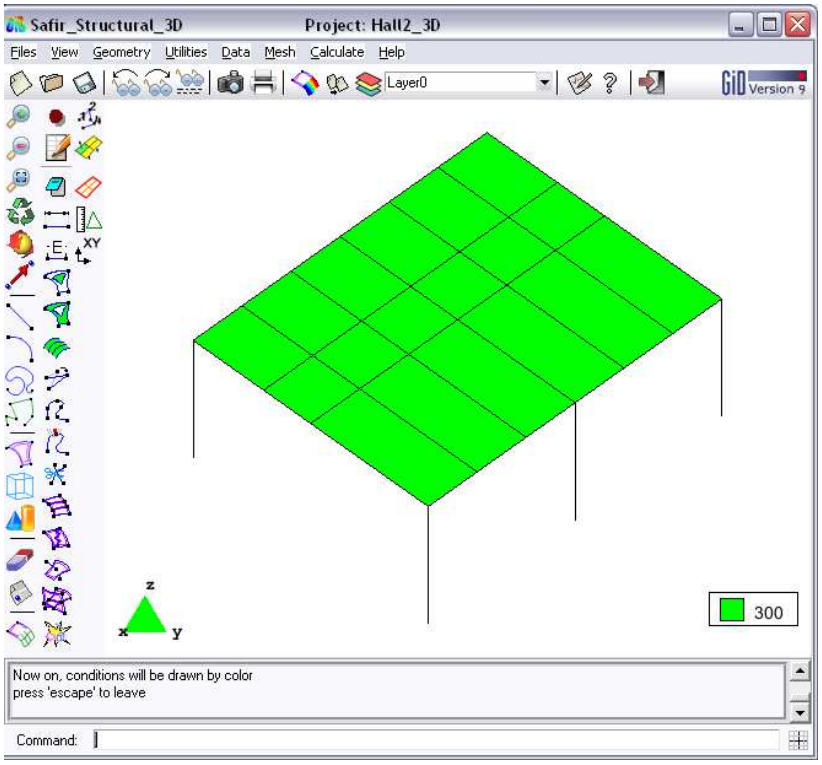
Click on:

- ***Data-> Mass***

Select Surface Mass []



Assign 300 to all surfaces



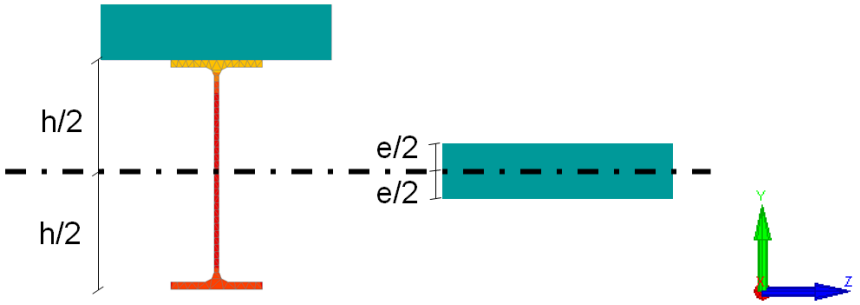
To display loads select in the dialog box:

- ***Draw->Colors***

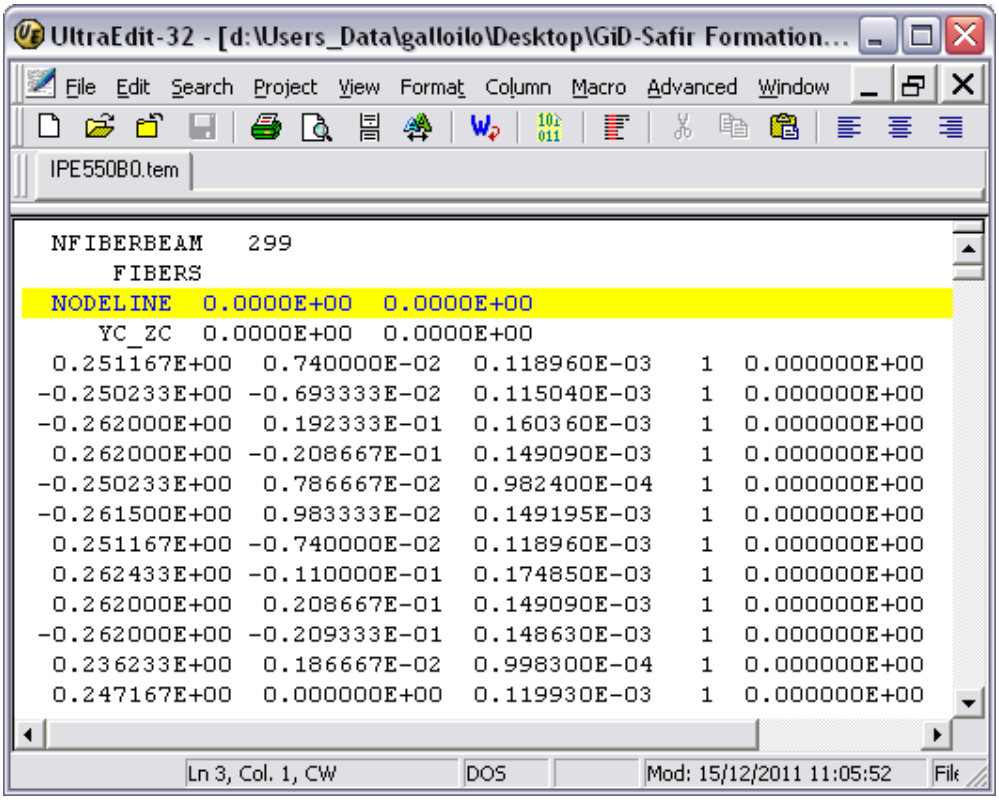
Press ***Finish*** or ***[Esc]*** to leave this view mode

8. Modify the node line in .TEM files

As already explained in exercises 3 and 4:
- If you are using the “Cross-Section” option to create a profile in thermal 2D Gid-Safir always center the profile on the 0,0 point of the plan who is located at the middle of the section height as shown below.
- If you are using the themal TSH mode, GiD-Safir will automaticly centered the shell element node line on 0,0.

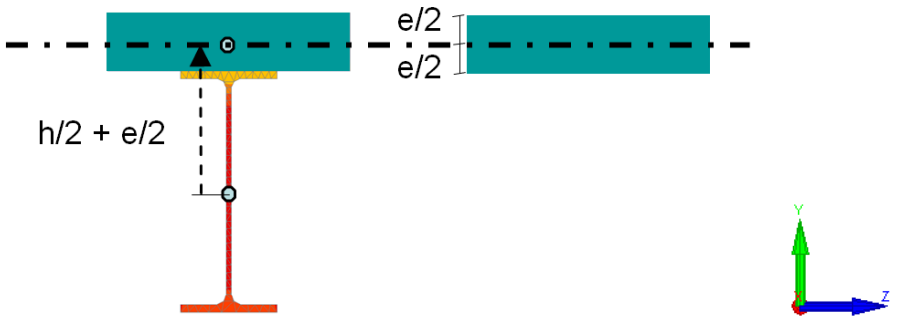


Now, open the IPE550B0.tem file:

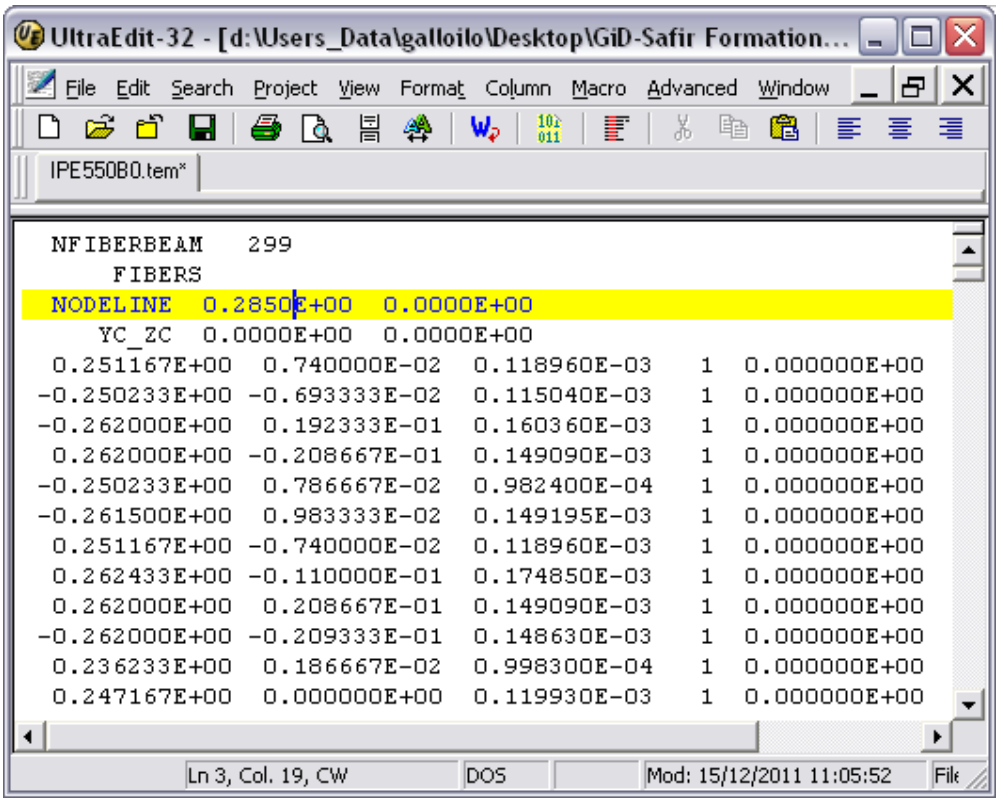


On the 3th line you can see the line “NODELINE”. The two 0.0000E+00 numbers are respectively the Y0 and the Z0 coordinate in meter of the global center.

To displace the local center as show below:

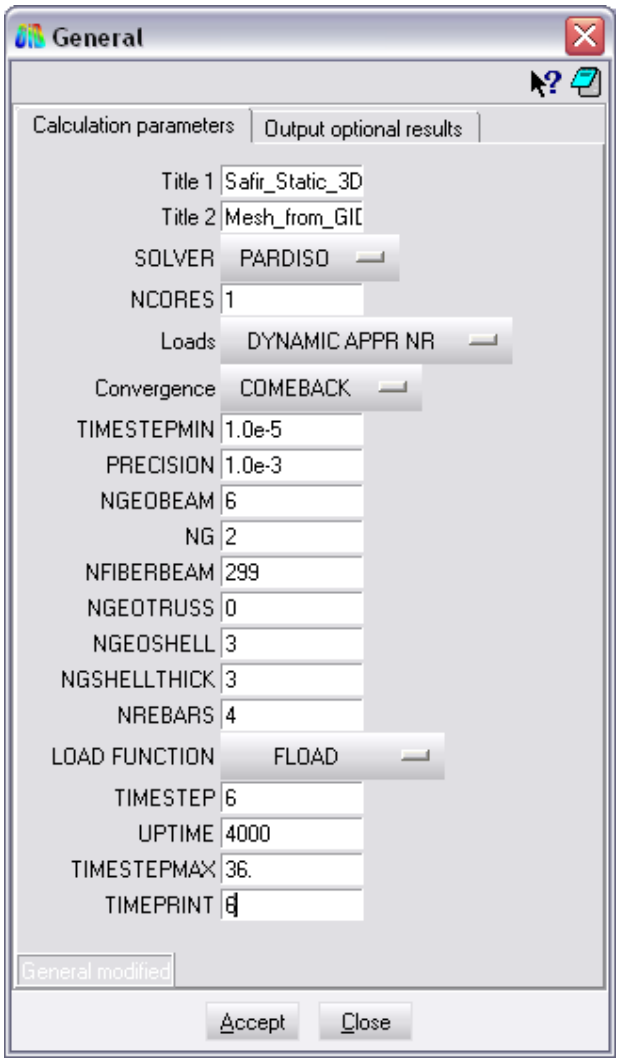


But Y0 = 0.2850E+00



Now save this file and modify the other IPE550B1 and IPE550B2 sections.

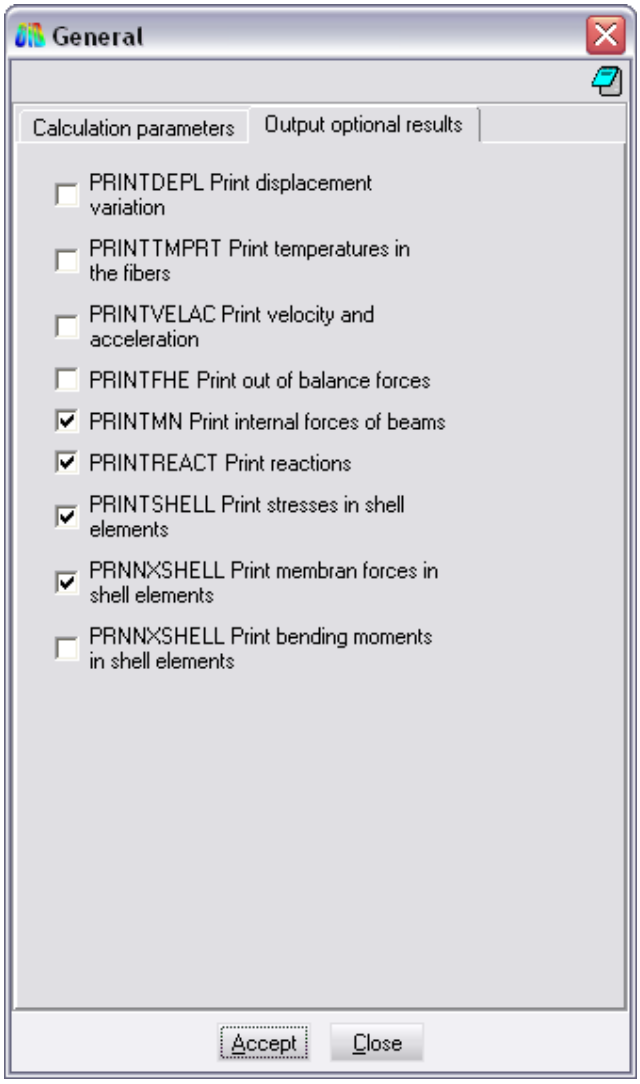
9. Modify Problem Data



In the Calculation parameters tab, put 3 as number of geoshell (NGEOSHELL) and 4 as number of rebars (NREBARS)

⚠ *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*



In the Output optional results, select what you want Gid-Safir to print, for example in this case tick PRINTSHELL and PRNNXSHELL

Click on ***Accept*** to confirm

10. Generate Mesh

From the pull down menu select preferences [], open the meshing tab and select [] (beam) as “Mesh always by default” (in the bottom of the window)

Click on ***Accept*** to confirm

⚠ *If you close open and mesh again this structure, you will have to select beam as “Mesh always by default” once again. GiD doesn’t save this preference.*

Form the pull down menu select

- ***Mesh-> Structured-> Surfaces-> Assign***

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

Enter 5 as number of cells to assign to line and select line 15-11

Press ***[Esc]***

Enter 2 as number of cells to assign to line and select line 53-44; 56-65; 10-21; 20-6; 6-19; and 18-2

Press ***[Esc]***

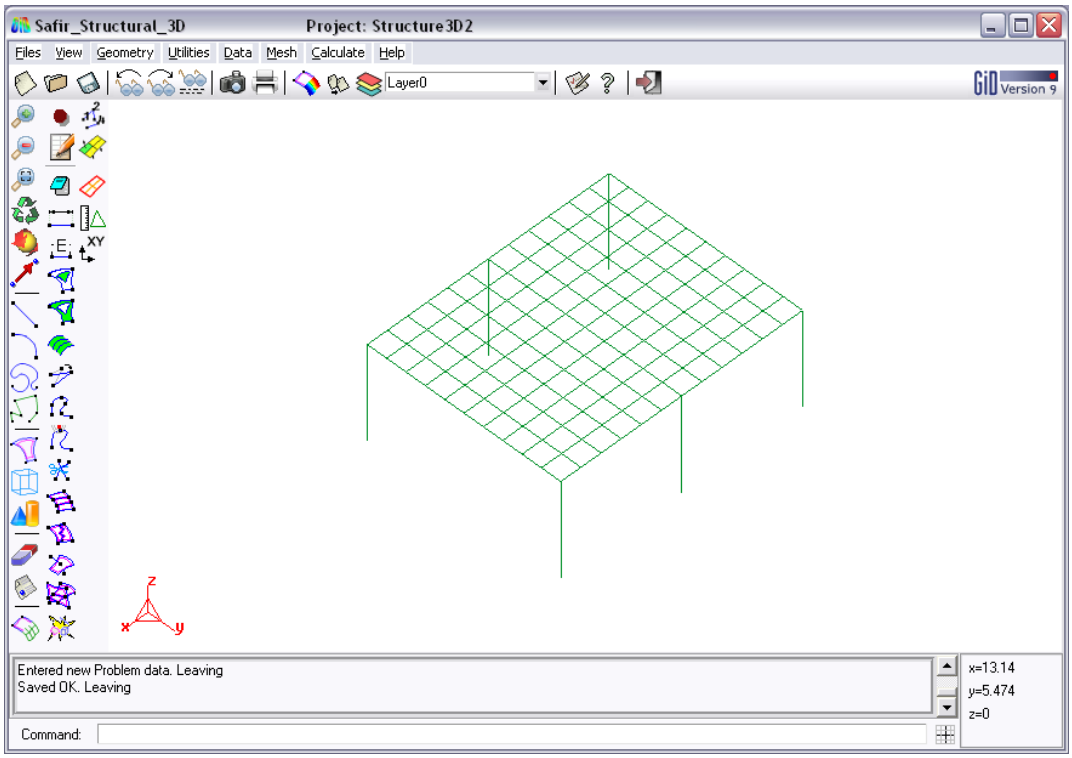
Enter 3 as number of cells to assign to line and select line 21-53 and 19-59

Press ***[Esc]*** and close the Mesh Windows

Select from the pull down menu:

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

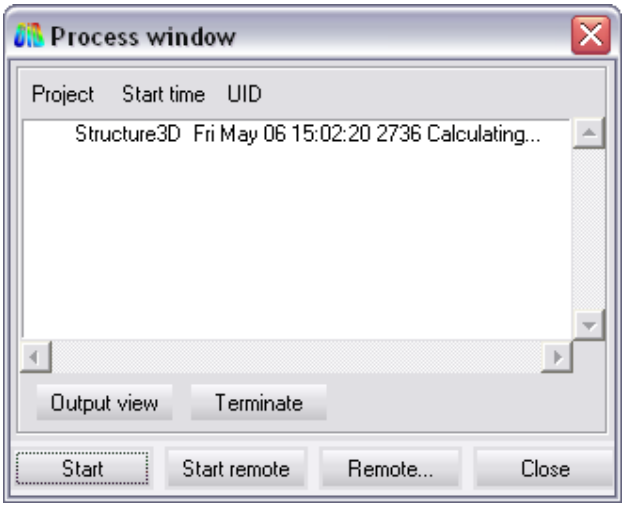
Enter the element size of 0.5 m



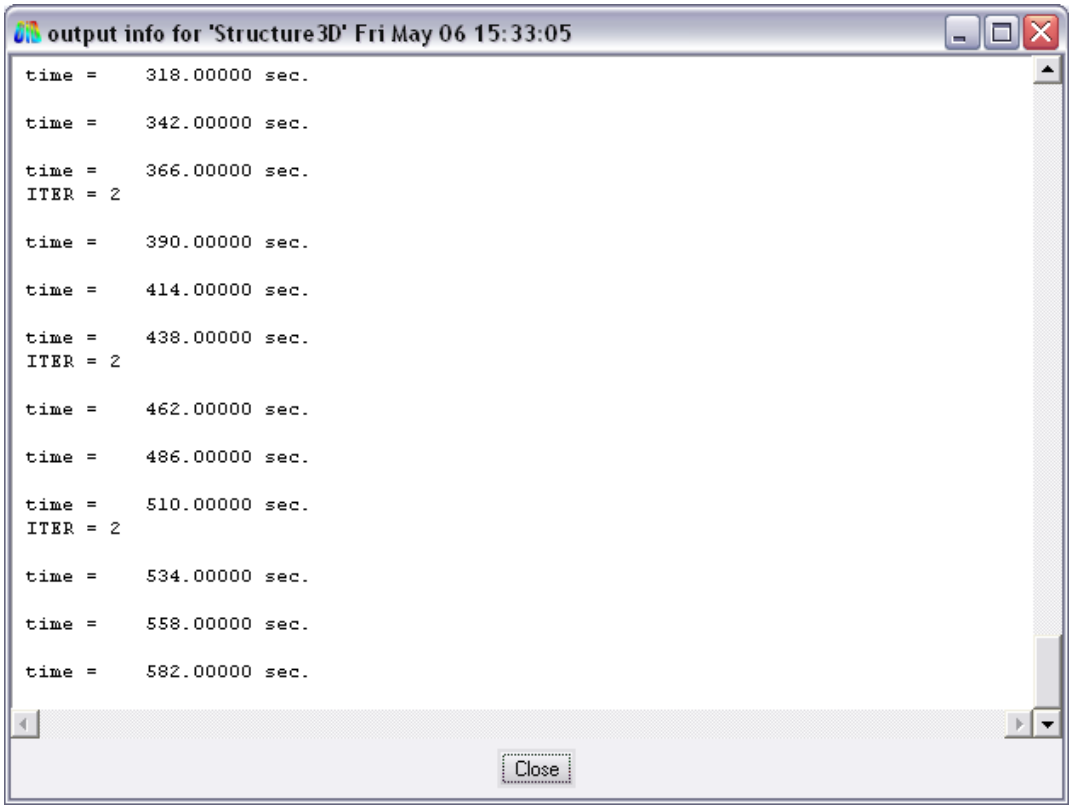
11. Start the calculation:

Select from the pull down menu:

- ***Calculate-> Calculate Windows***



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



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 on the Postprocessor Icon in the tool box.*