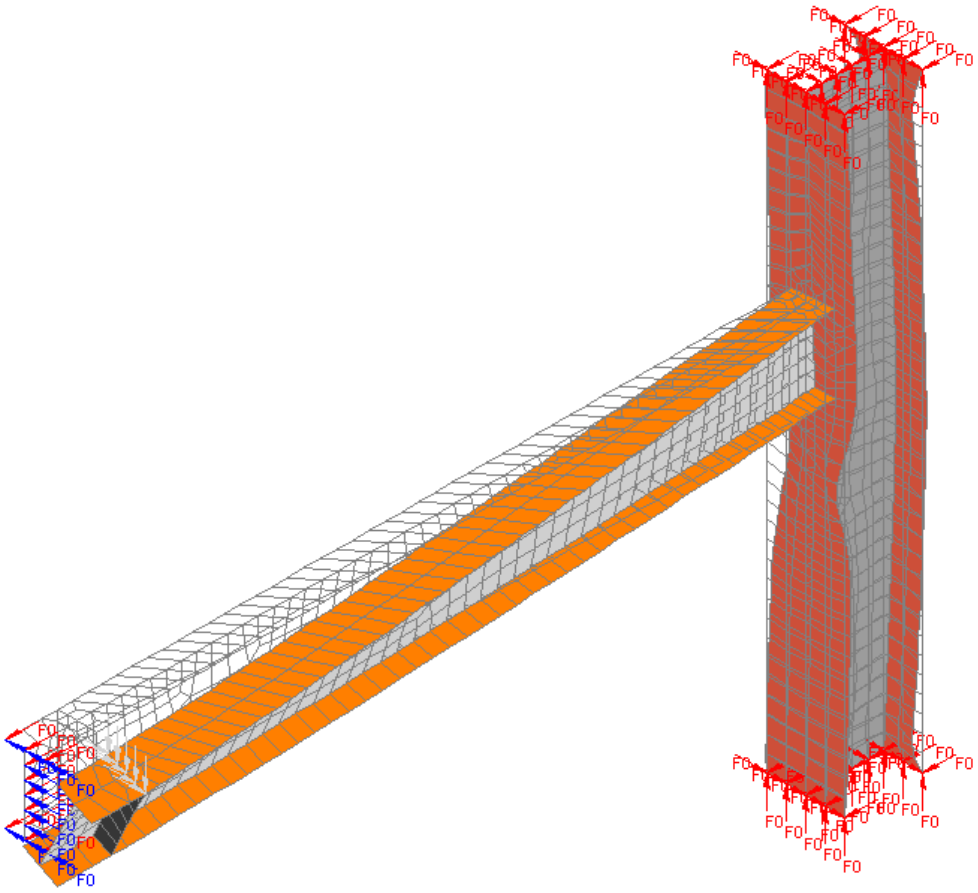
Tutorial for GID-SAFIR 3D Structural Analysis

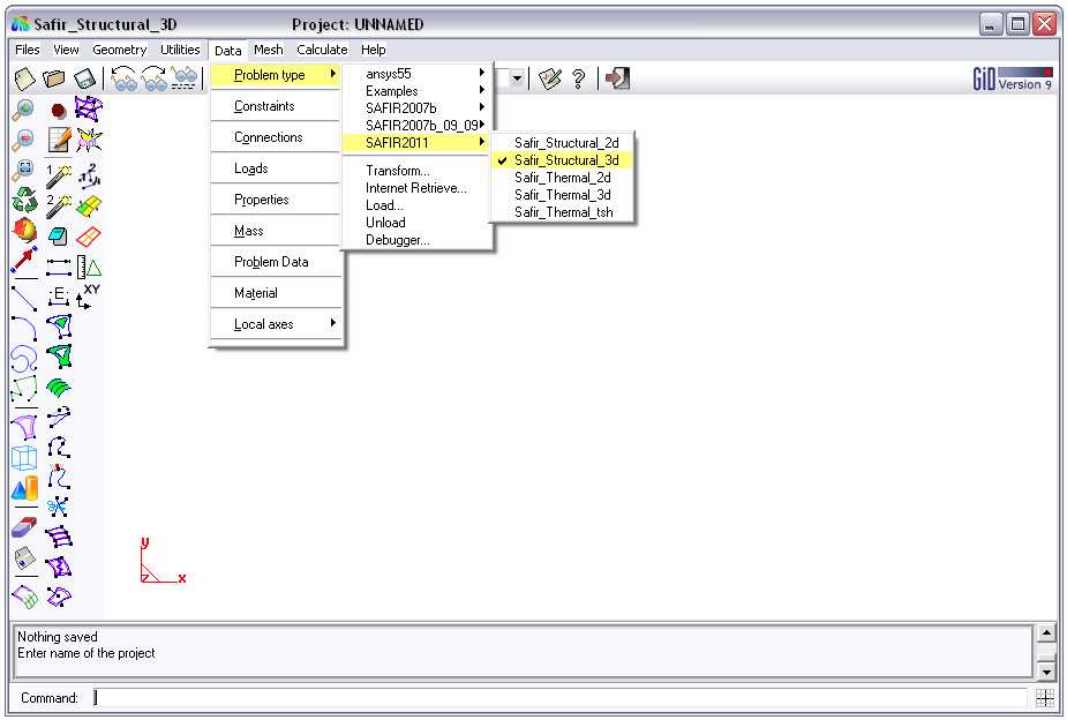
Exercise n°10 Beam Shell 3D



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.: ***BeamShell_3D***

GiD creates a directory with the name ***BeamShell_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.

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* 

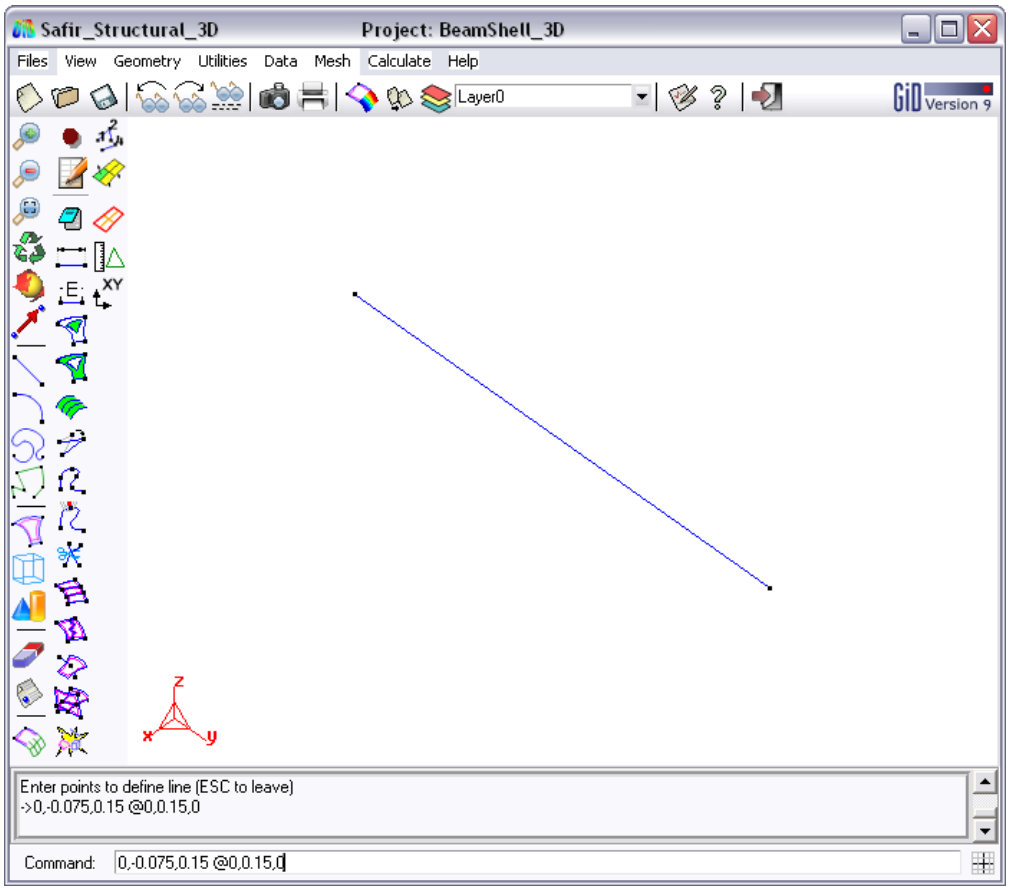
Enter in the command line (at the bottom of the widows):

0,-0.075,0.15 @0,0.15,0 and press *[Enter]*

Select from the pull down menu:

- *View->Rotate->isometric*

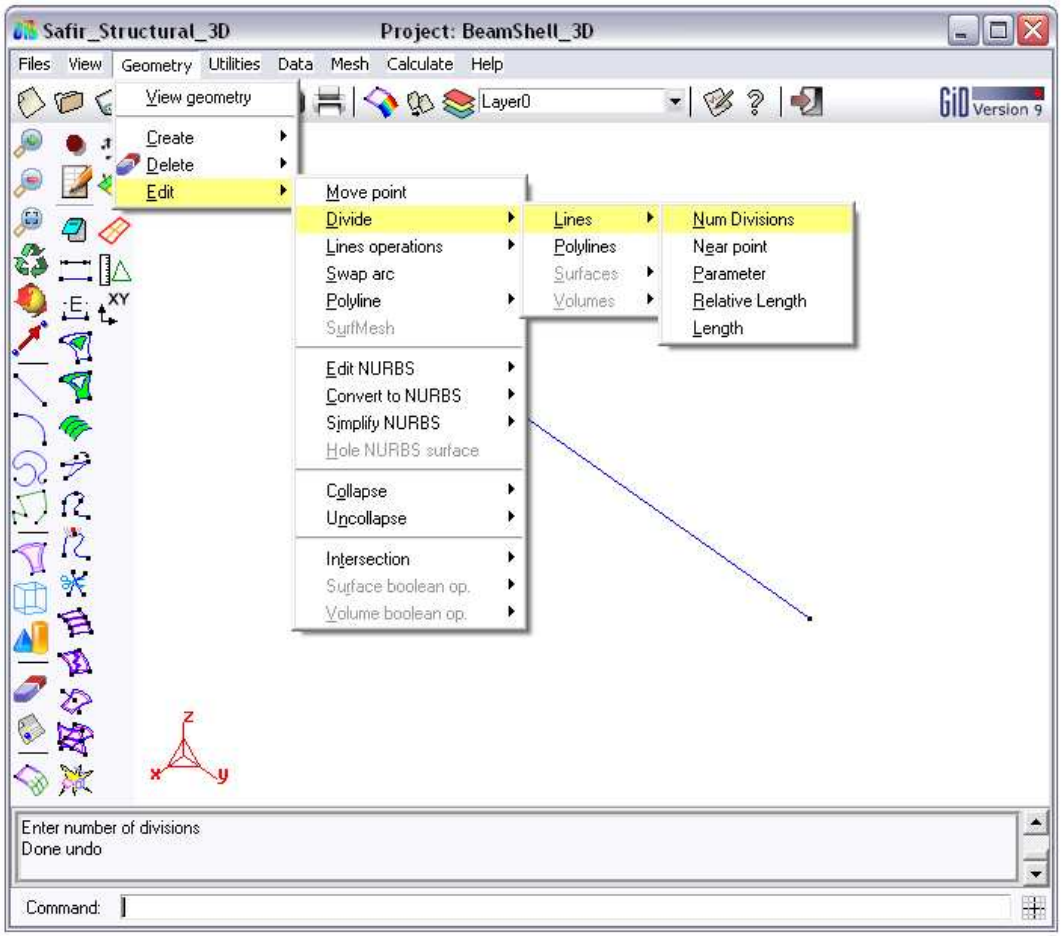
Press on *[F11]*



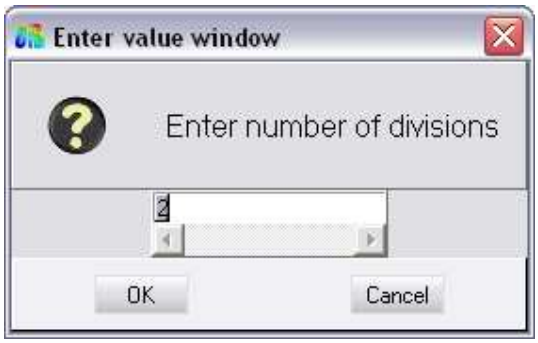
The first line is created.

In order to divide this line in 2, select from the pull down menu:

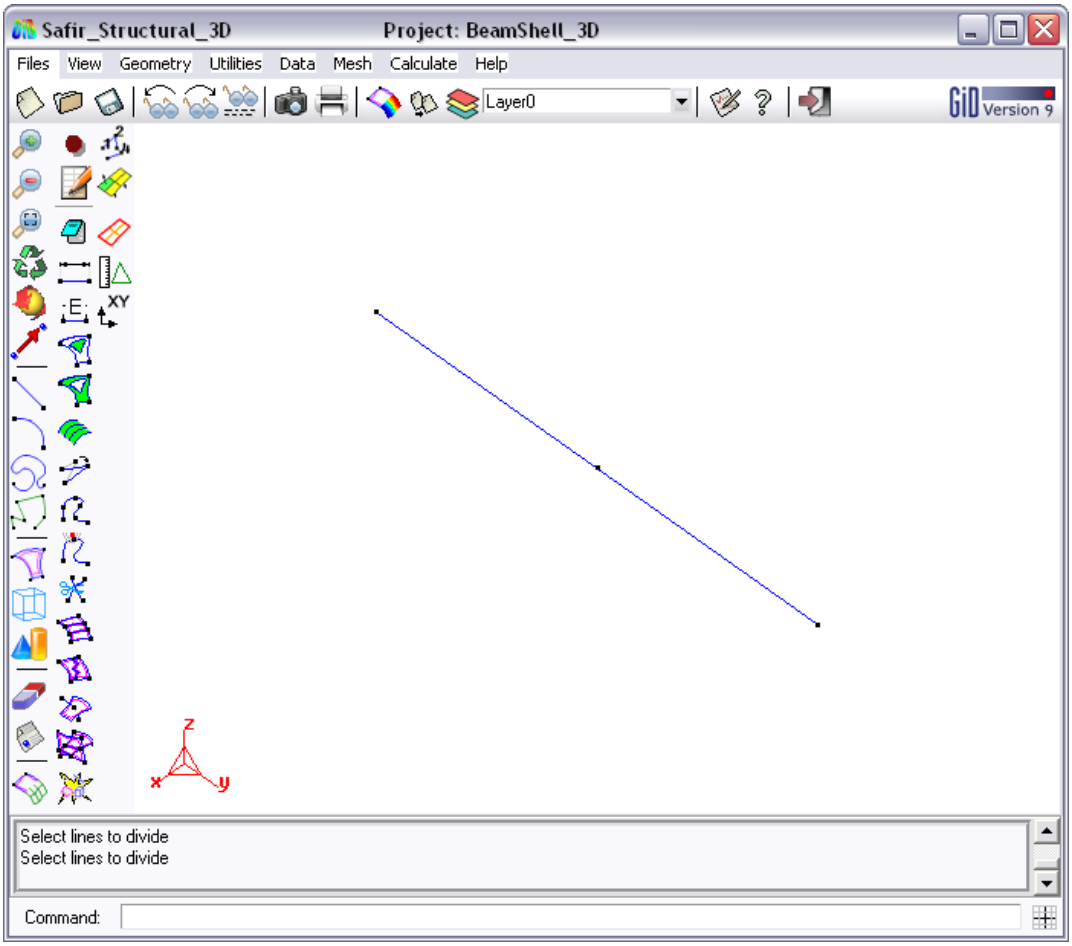
- ***Geometry->Edit-> Divide-> Lines-> Num Divisions***



Put 2 as number of divisions and press ***OK*** or ***[Enter]***



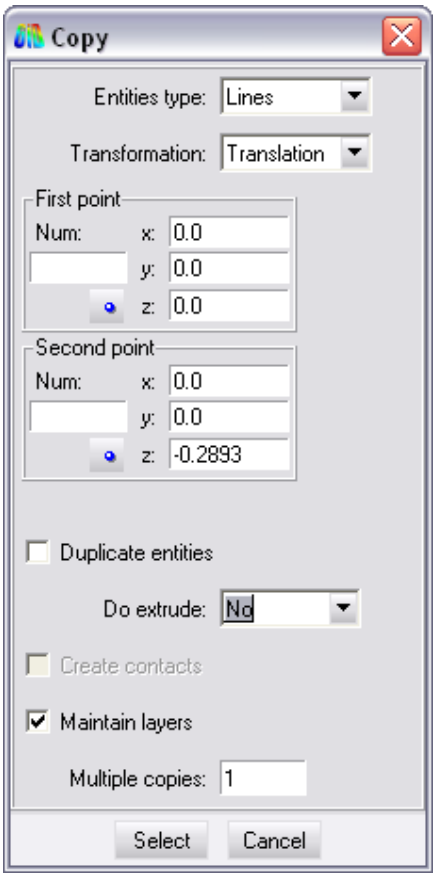
Select the line and press ***[Esc]***



Select from the pull down menu:

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

  And fill as shown below:



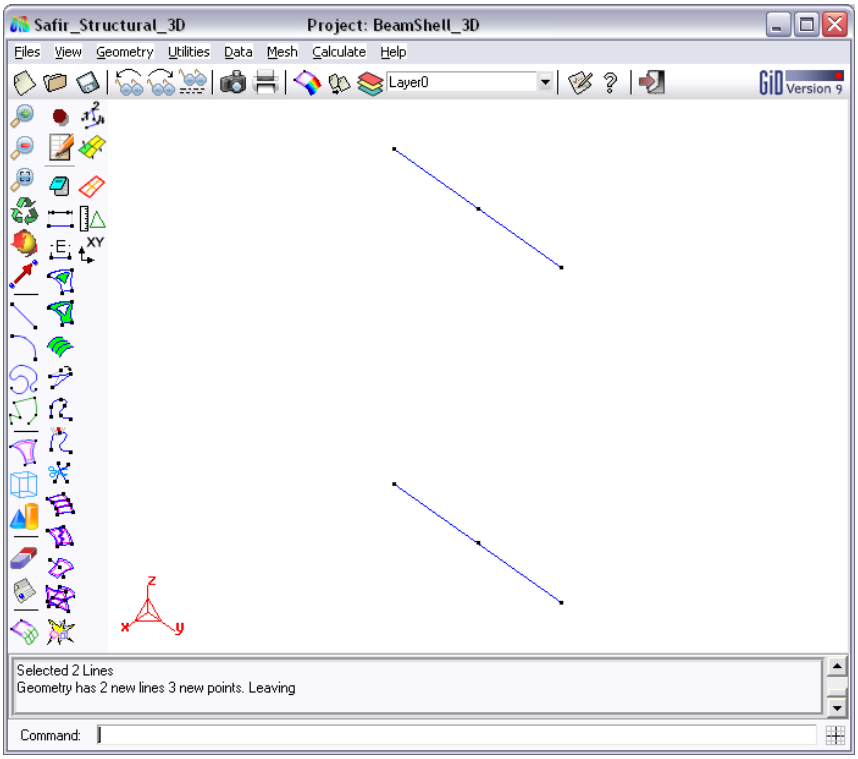
As Entity type, use: ***Lines***

Enter for Second point: ***z = -0.2893***

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

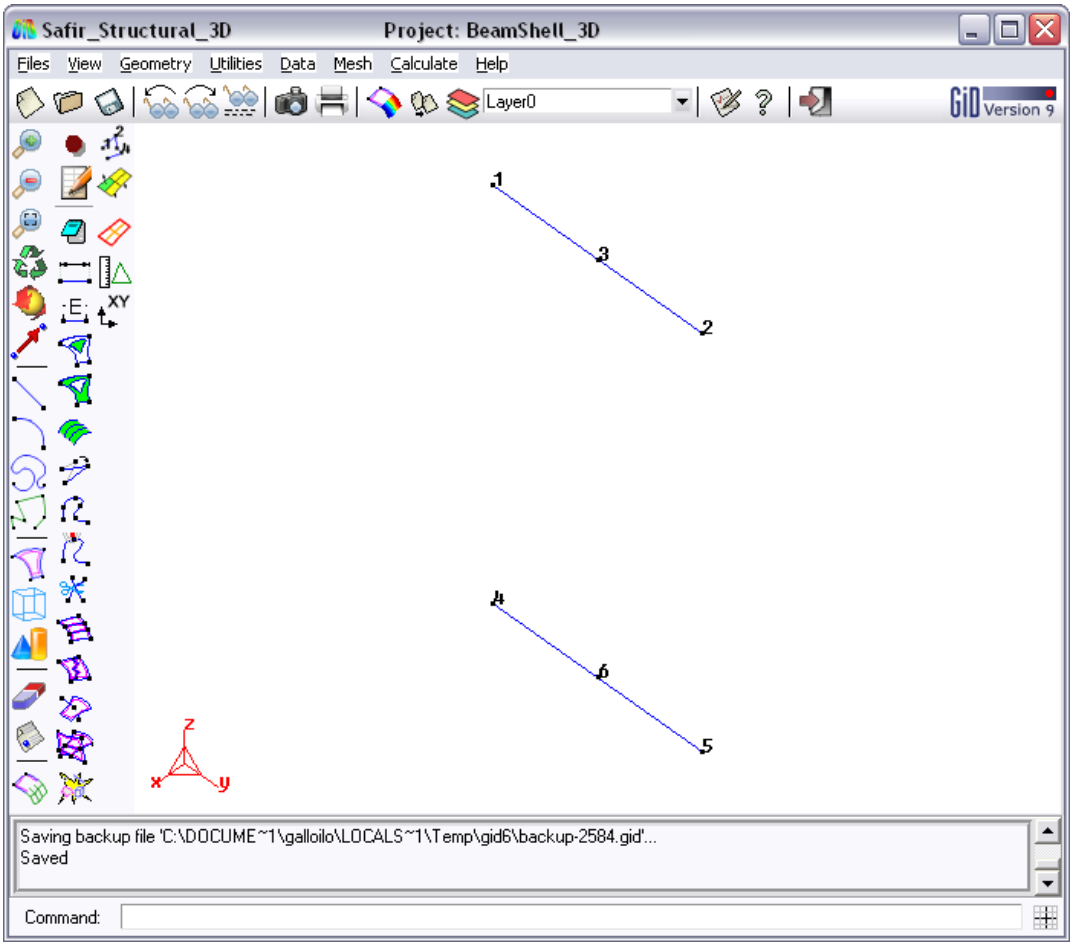
Press on ***[F11]***

⚠ *In this case we want to design an IPE 300. In order to get the exact dimension we have to take into account the flange thickness (for an IPE 300, tf = 10.7 mm)*



Select from the pull down menu:

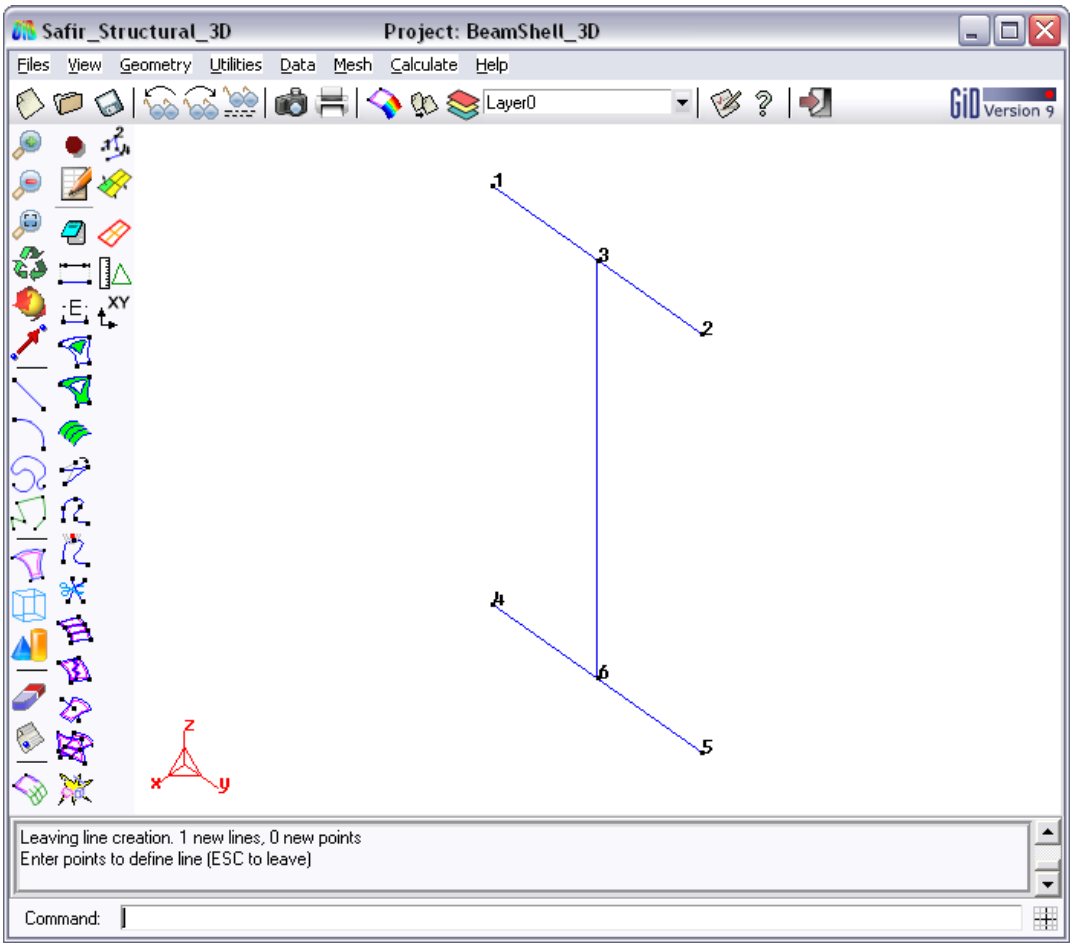
- ***View-> Label-> All in-> Node***



To connect point 3 and 6, select from the pull down menu:

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

Press *[Ctrl + a]* and select the node *3* then the node *6* and press *[Esc]*



Select from the pull down menu:

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

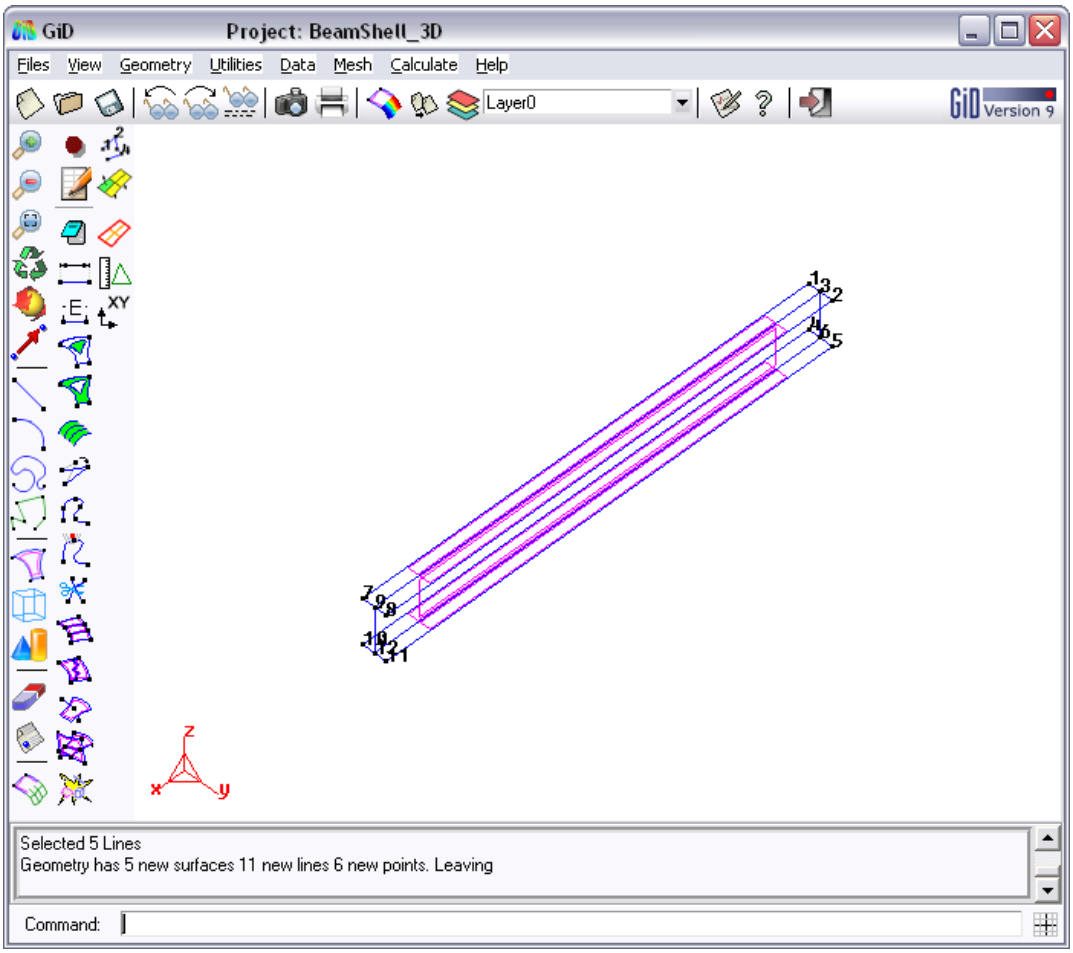
For Entity type, use: ***Lines***

Enter for Second point: ***x = 2.8***

Do extrude: **Surfaces**

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

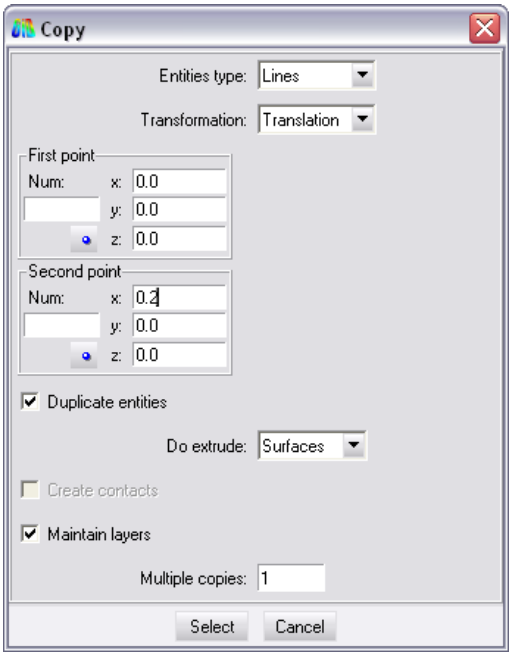
Press on ***[F11]***



Select from the pull down menu:

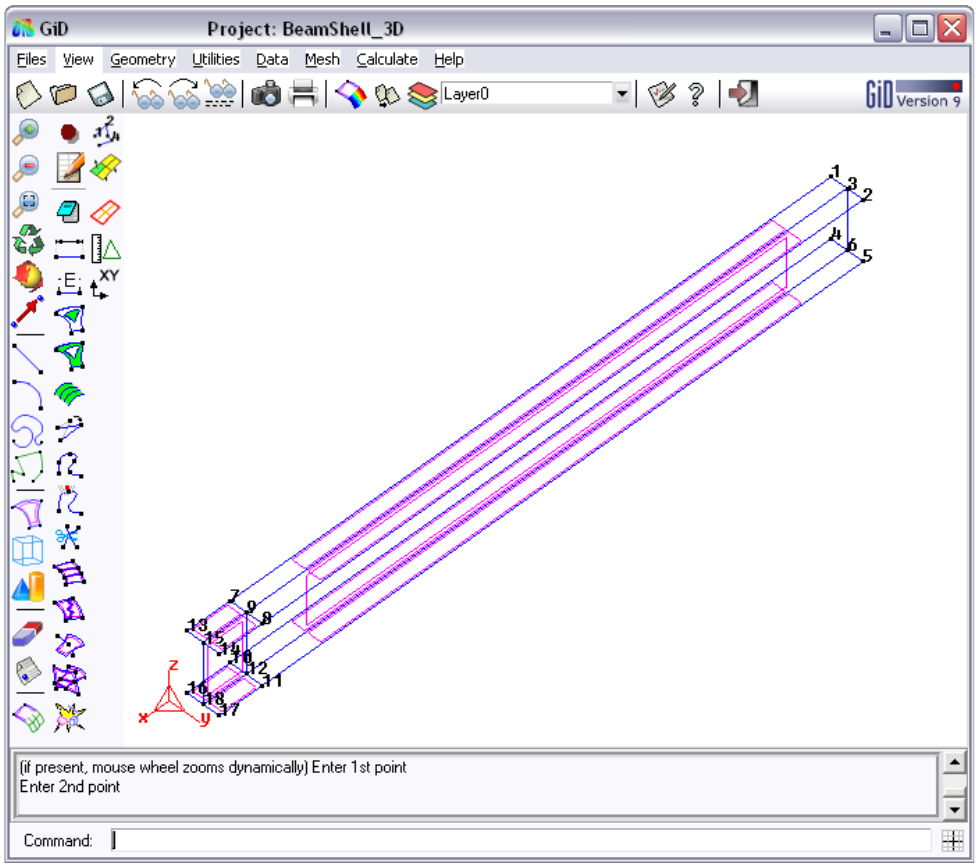
- ***Utilities->Copy*** ***or [Ctrl + c]***

Fill as below:



Select all lines ***7-9***, ***8-9***, ***10-12***, ***11-12***, ***9-12*** and press ***[Esc]***

Press on ***[F11]***



⚠ *To make your selection easier, use all the view mode as view-> rotate-> trackball or view-> zoom-> in. To get back to basic view, select from the pull down menu: view-> rotate-> isometric and press [F11]*

To connect points 7 to 10 and 8 to 11, select from the pull down menu:

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

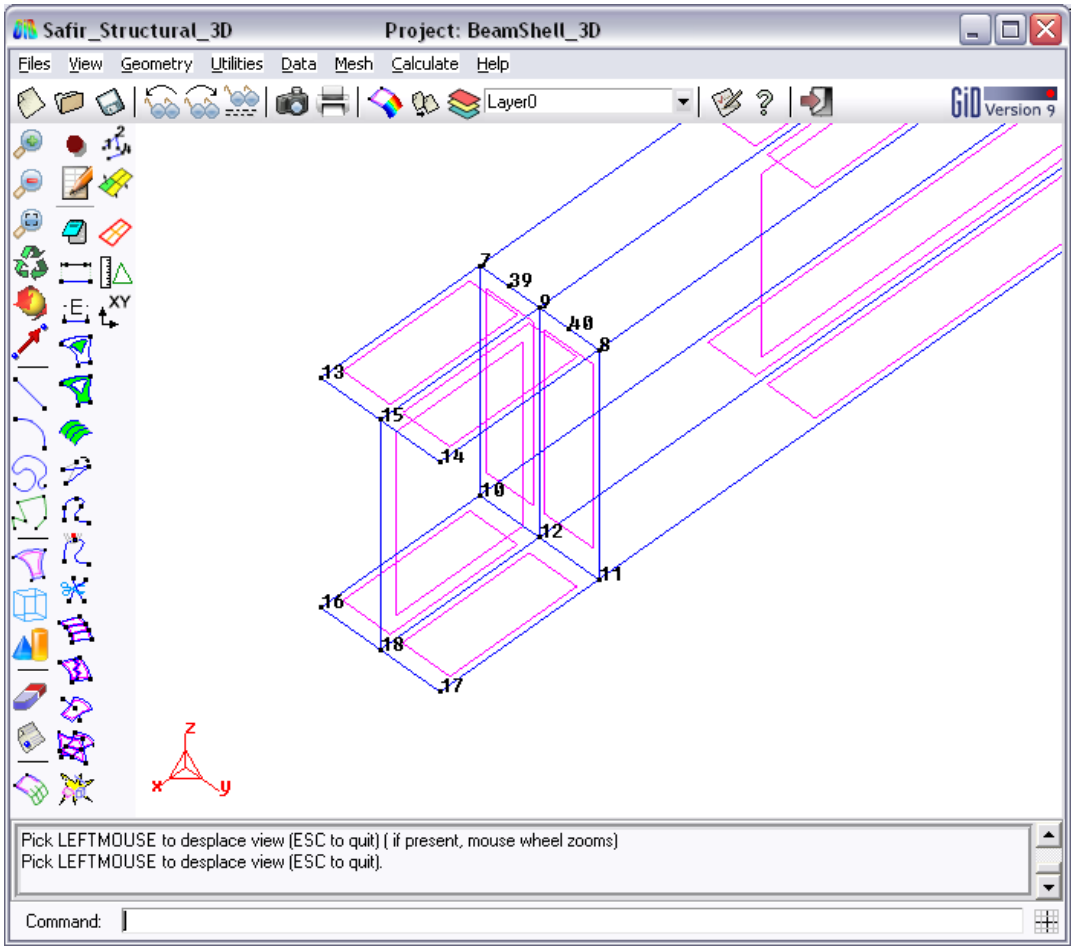
Press ***[Ctrl + a]*** and select the node ***7*** then the node ***10*** and press ***[Esc].*** Do the same operation with node ***8*** and ***11.***

To create surfaces, select from the pull down menu:

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

Select lines ***8-11***, ***9-12***, ***8-9***, ***11-12*** and press [Esc]

Select lines ***7-9***, ***10-12***, ***7-10***, ***9-12*** and press [Esc]



Select from the pull down menu:

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

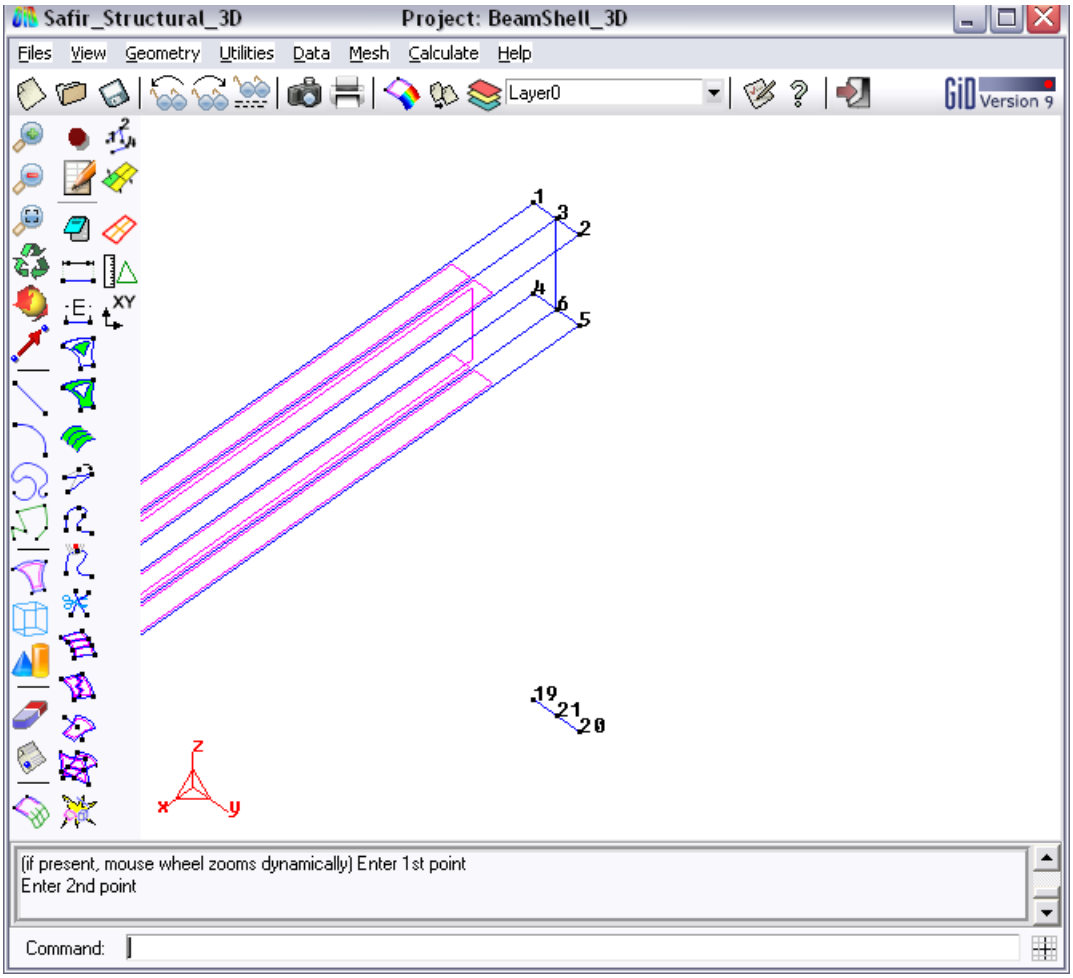
And fill as shown below:

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

Enter for Second point: ***z = -1.35***

Select lines ***4-6*** and ***5-6*** and press ***[Esc]***

Press on ***[F11]***



Press again on ***[Ctrl + c]***:

For Entity type, use: ***Points***

Enter for Second point: ***y = 0.075***

Do Extrude: ***Lines***

Select point ***20*** and press ***[Esc]***

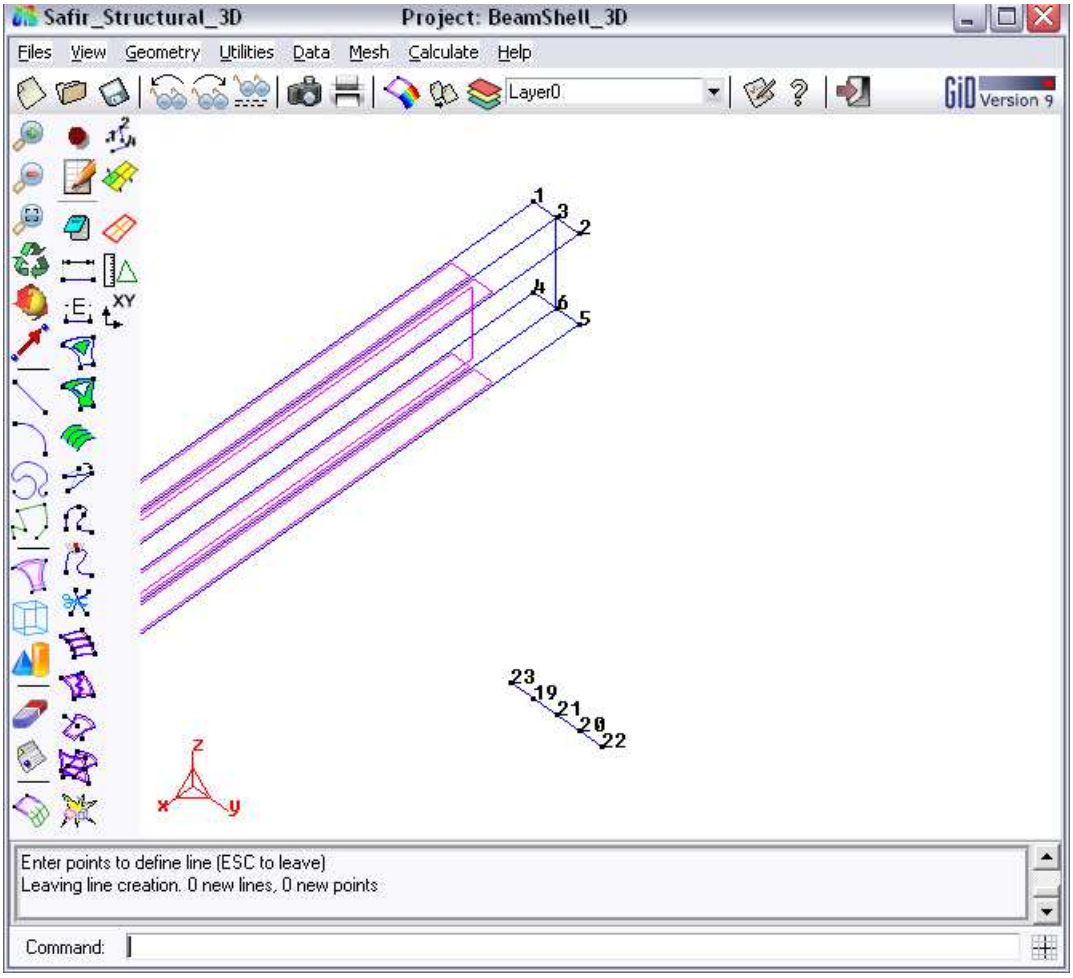
Press again on ***[Ctrl + c]***, an fill as below

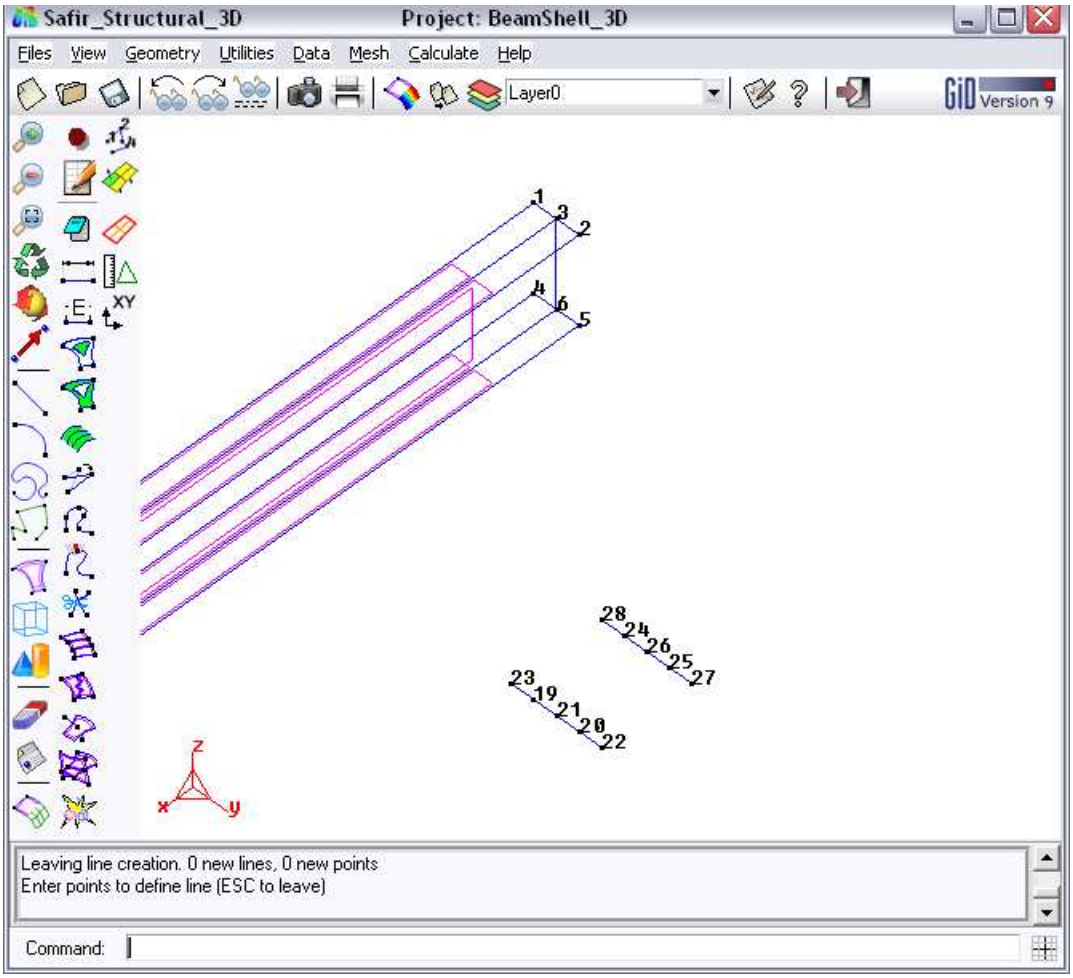
For Entity type, use: ***Points***

Enter for Second point: ***y =- 0.075***

Do Extrude: ***Lines***

Select point ***19*** and press ***[Esc]***





Use *[Ctrl + c]*, and fill as below

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

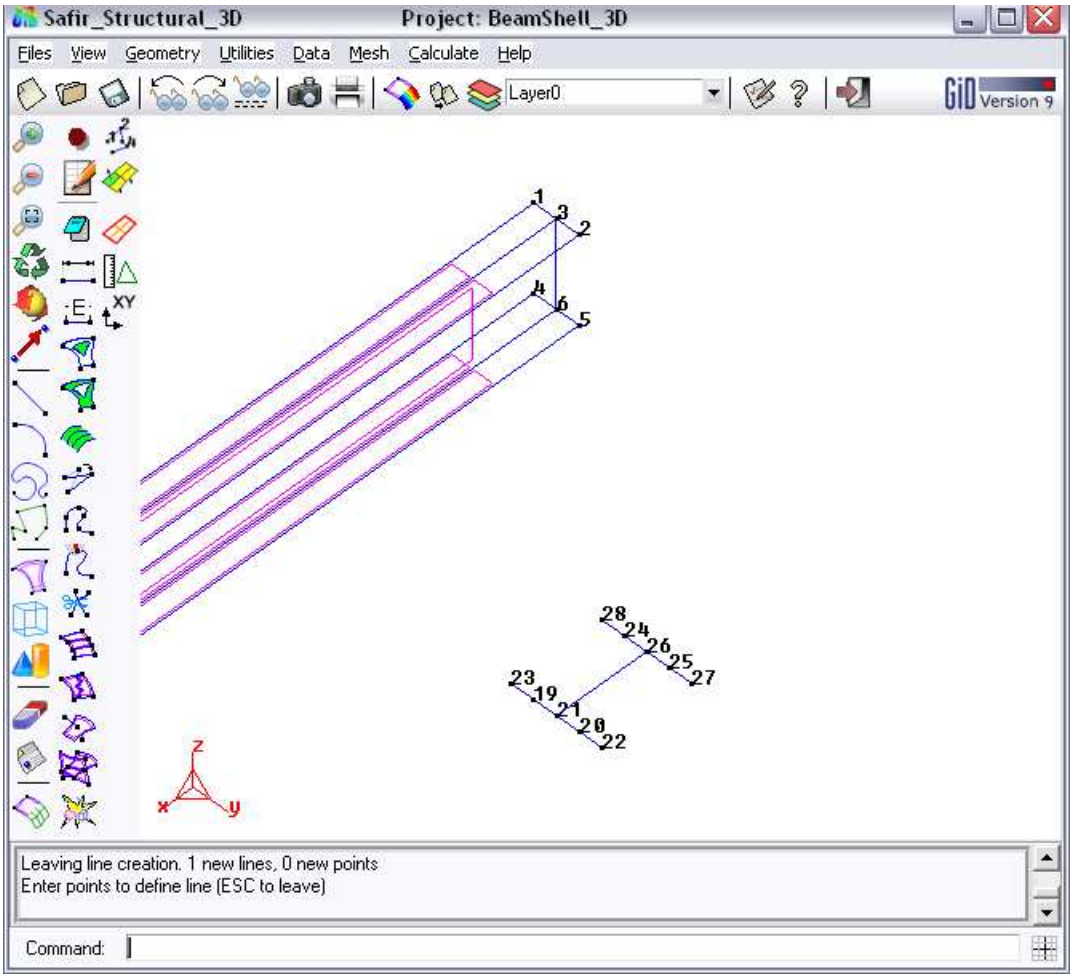
Enter for Second point: x ***= -0.2893***

Select line ***22-23*** and press ***[Esc]***

To connect the points, select:

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

Press ***[Ctrl + a]*** and pick points ***21*** and ***26*** and press ***[Esc]***

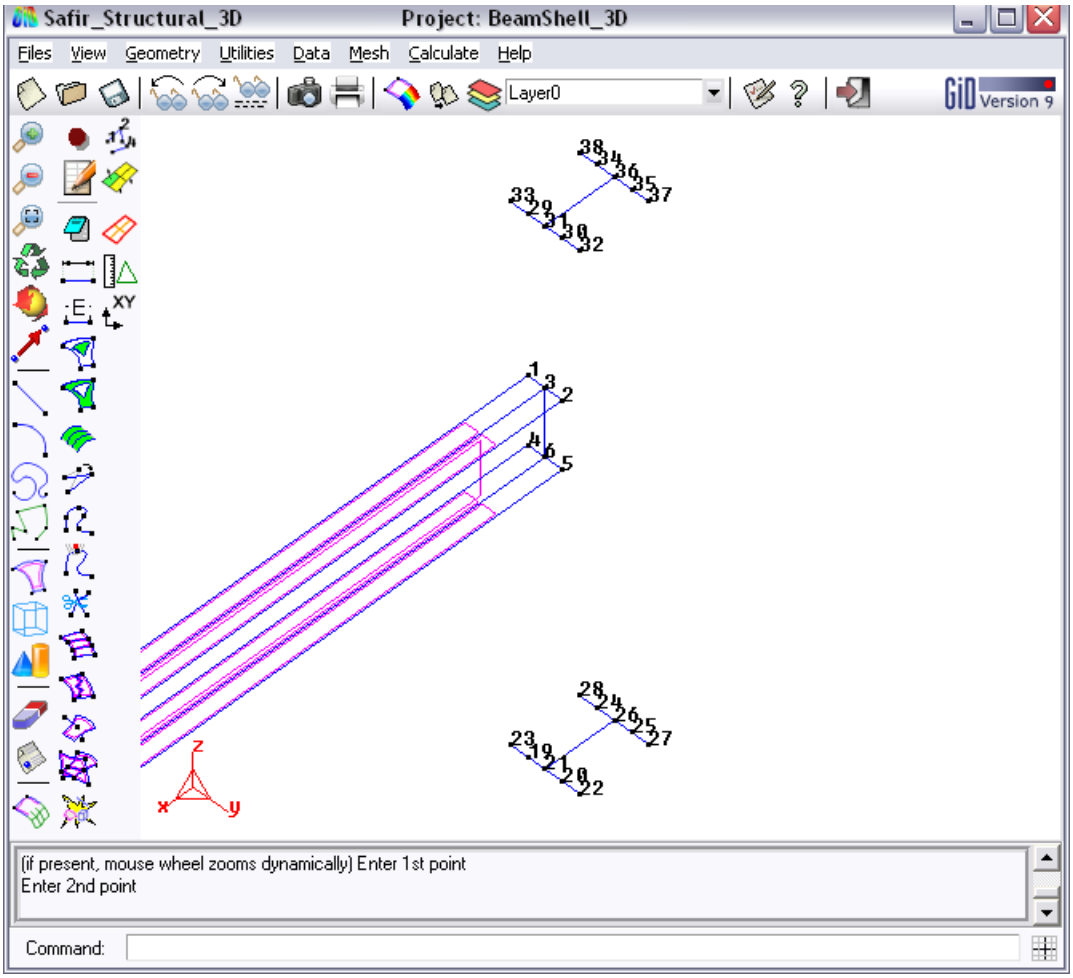


Use *[Ctrl + c]*, and fill as below

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

Enter for Second point: ***z= 2.35***

Select all the section you just created (lines ***22-23***; ***21-26*** and ***27-28***) and press *[Esc]*


Safir_Structural_3D
Project: BeamShell_3D
Files View Geometry Utilities Data Mesh Calculate Help
Layer0
GiD Version 9
(if present, mouse wheel zooms dynamically) Enter 1st point
Enter 2nd point
Command:

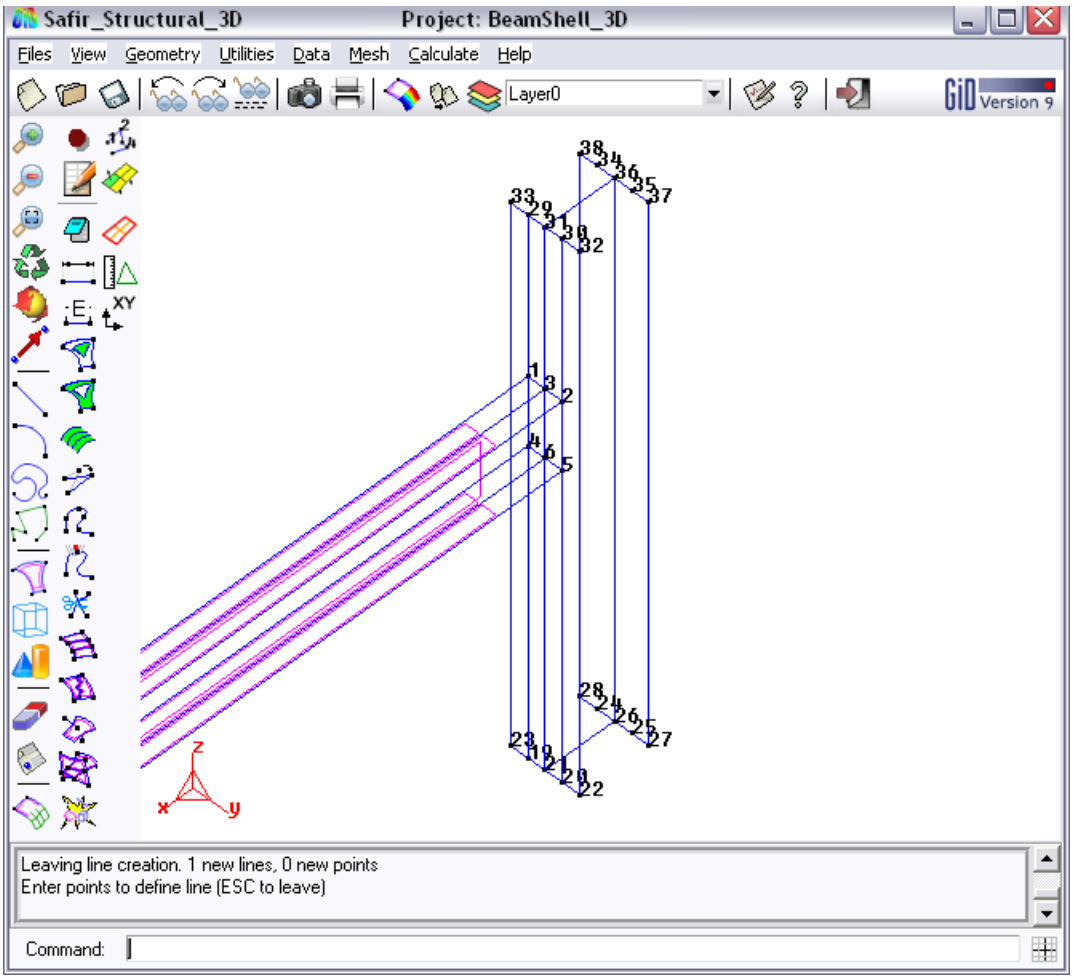
To connect the points, select:

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

Press *[Ctrl-a]* and pick points *38* and *28* and press *[Esc]*

Do the same operation with points:

36 § 26; *37 § 27*; *33 § 23*; *29 § 1*; *1 § 4*; *4 § 19*; *31 § 3*; *3 § 6*; *6 § 21*; *30 § 2*; *2 § 5*; *4 § 20* and *32 § 22*

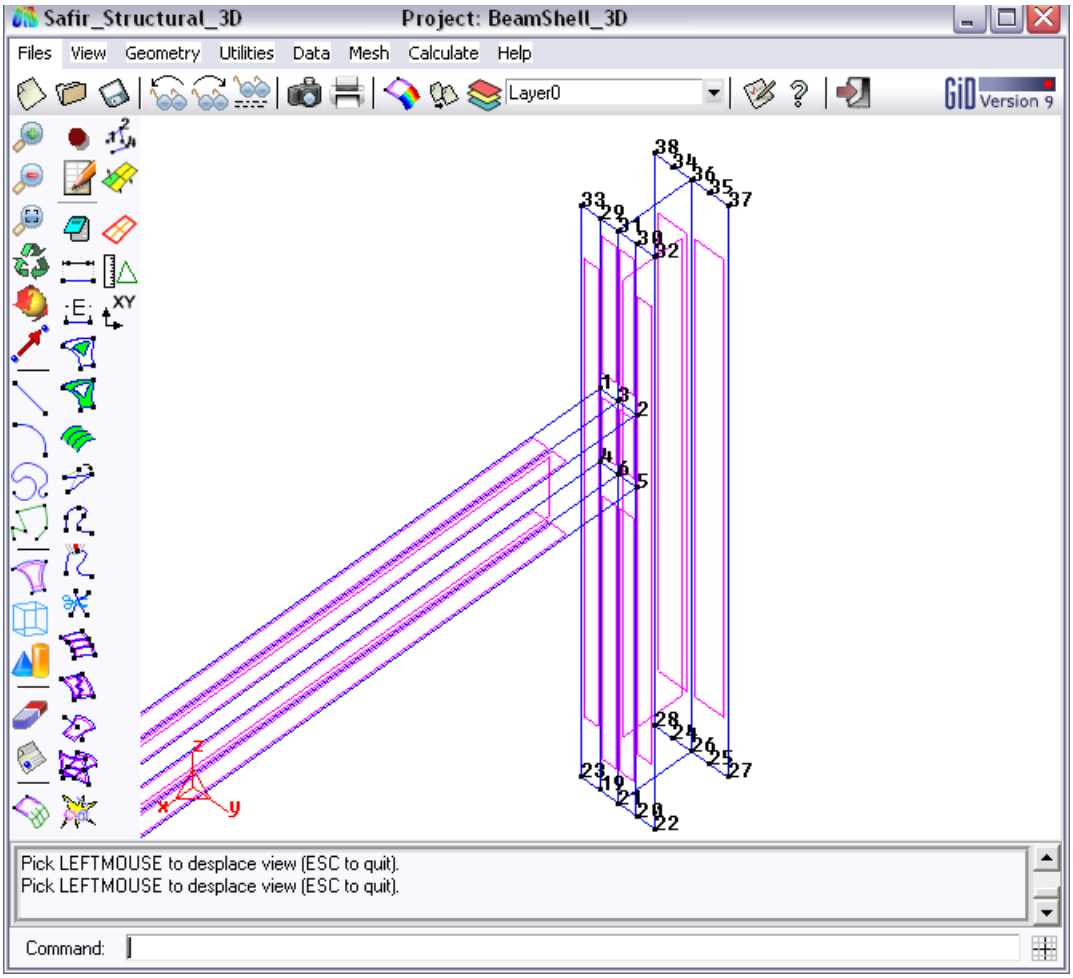


To create surfaces, select from the pull down menu:

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

Select all contour line of the rectangle delimited by node *38*, *36*, *26* and *28* and press *[Esc]*

Do the same operation in order to create NURBES surface for each rectangle you just created (as shown below)



3. Define constraints for the supports:

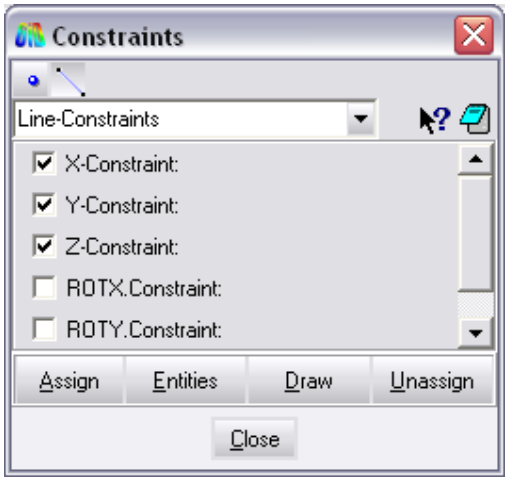
From the pull down menu select

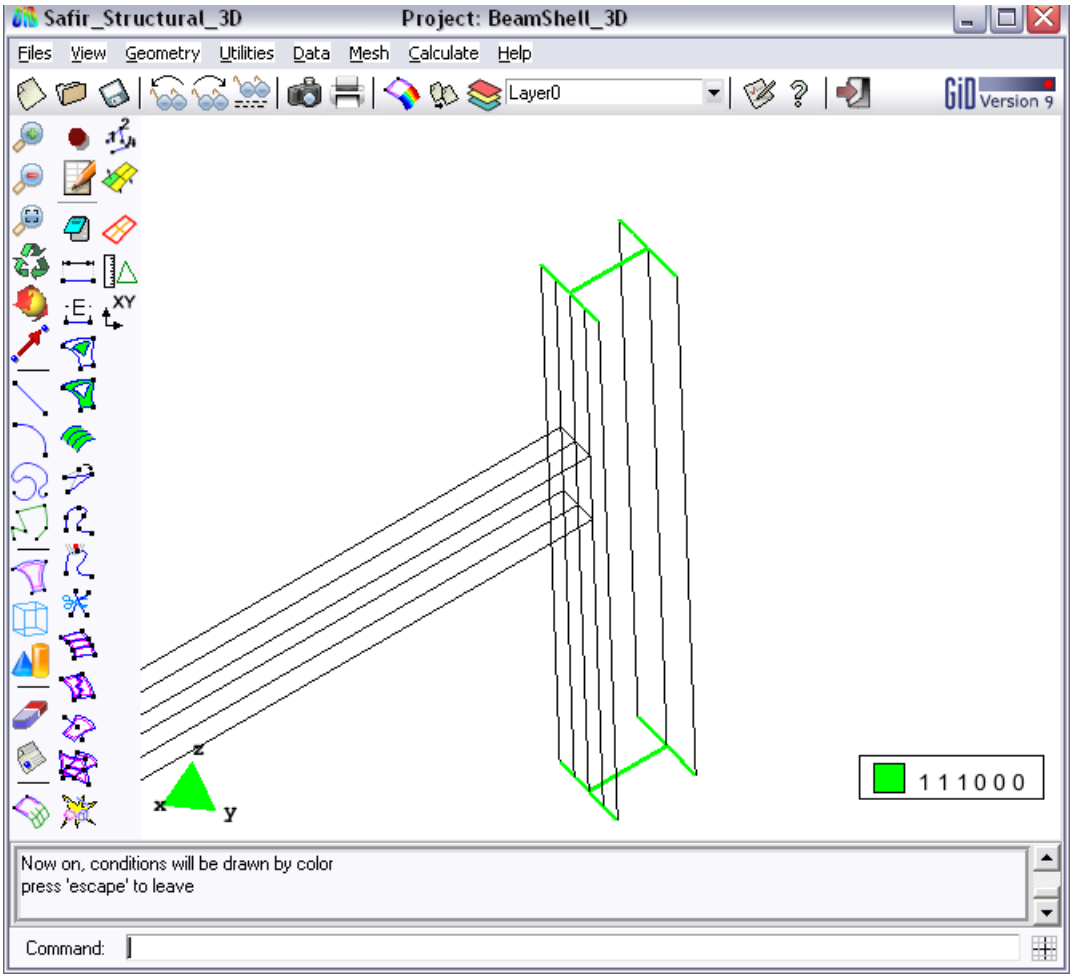
- ***Data->Constraints***

Select the line tab

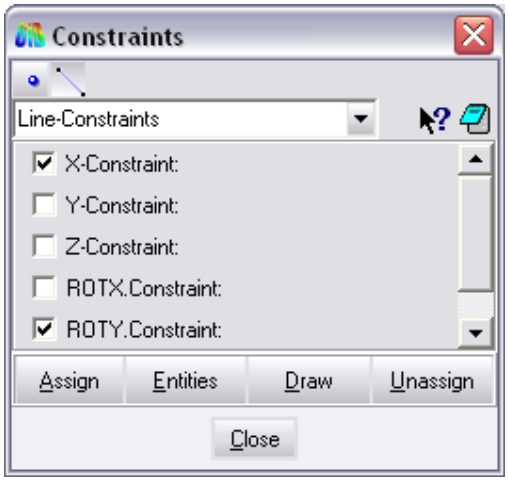
Select ***x***,***y*** and ***z*** constraints and assign them to the base and top lines of the column.

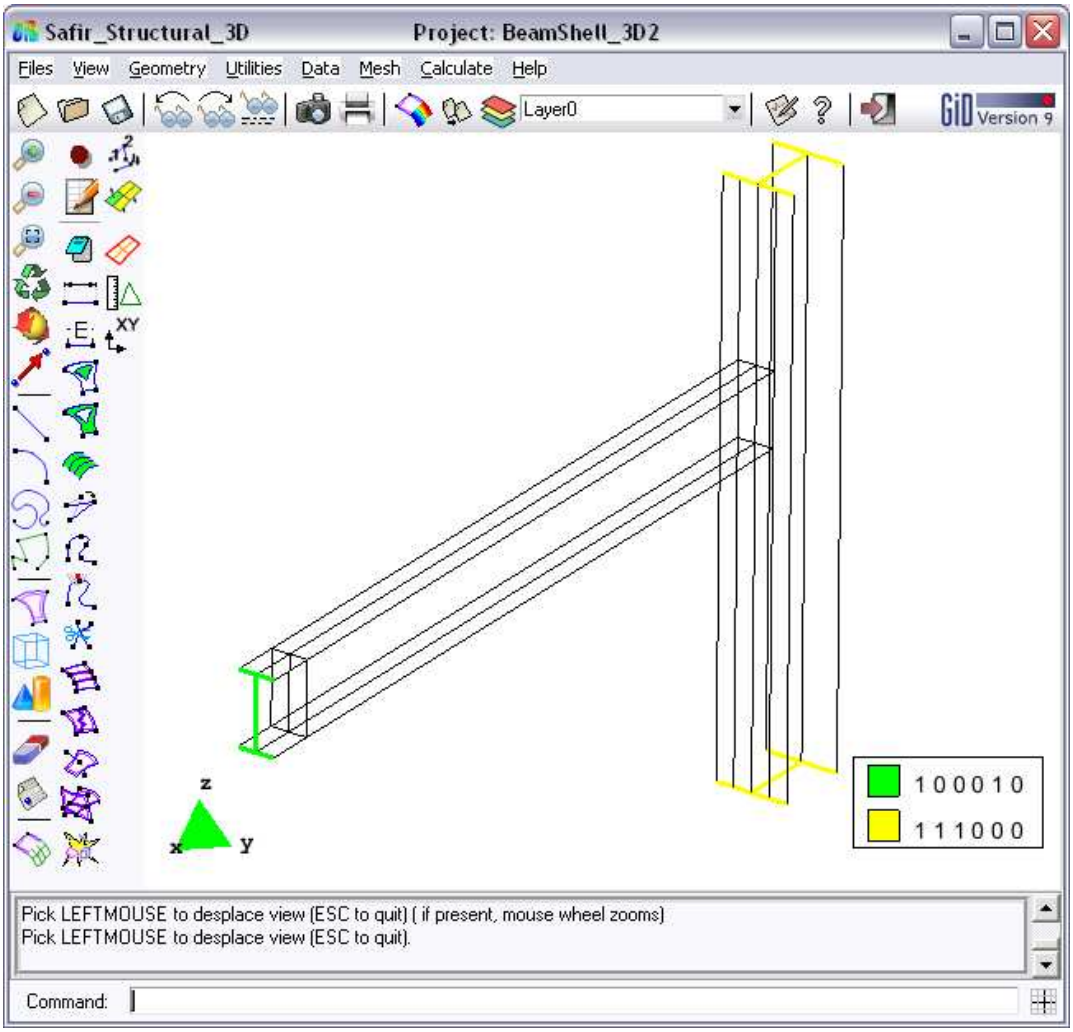
In the dial box, with ***Draw->Colors*** you can display the constraints.





Do the same operations to assign a X constraint and a Y rotation constraint to the exterior end part of the beam (as shown below)





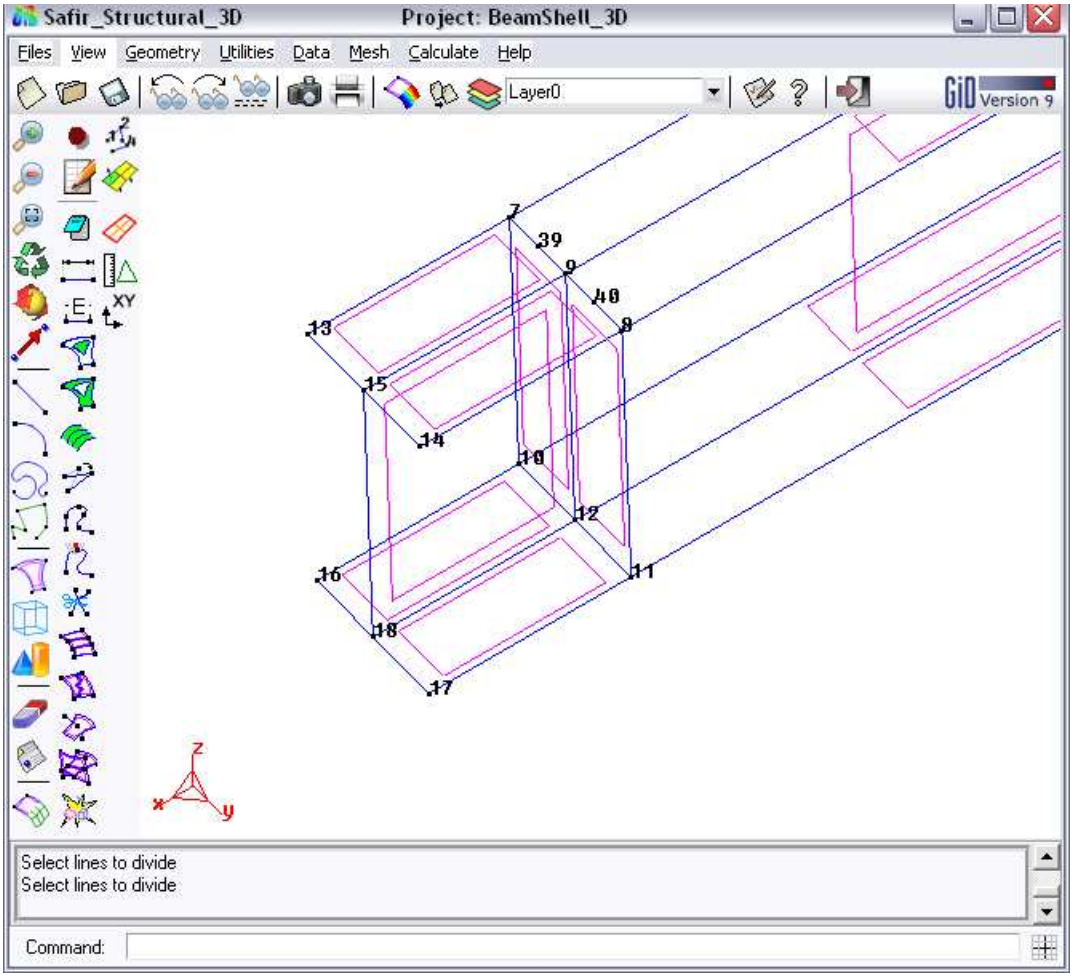
Press ***Finish*** or ***[Esc]*** to leave this view mode

4. Define loads

From the pull down menu select:

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

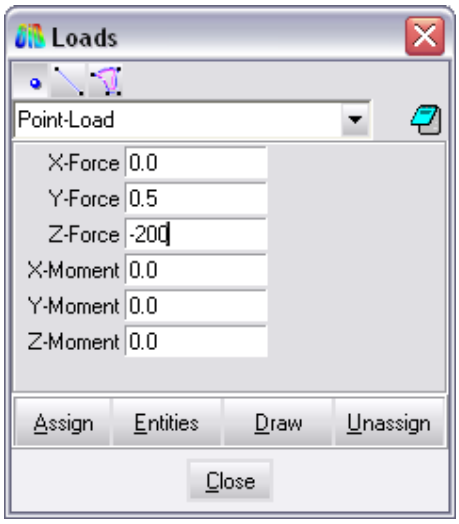
Divide lines *7-9* and *9-8* in *2*



From the pull down menu select:

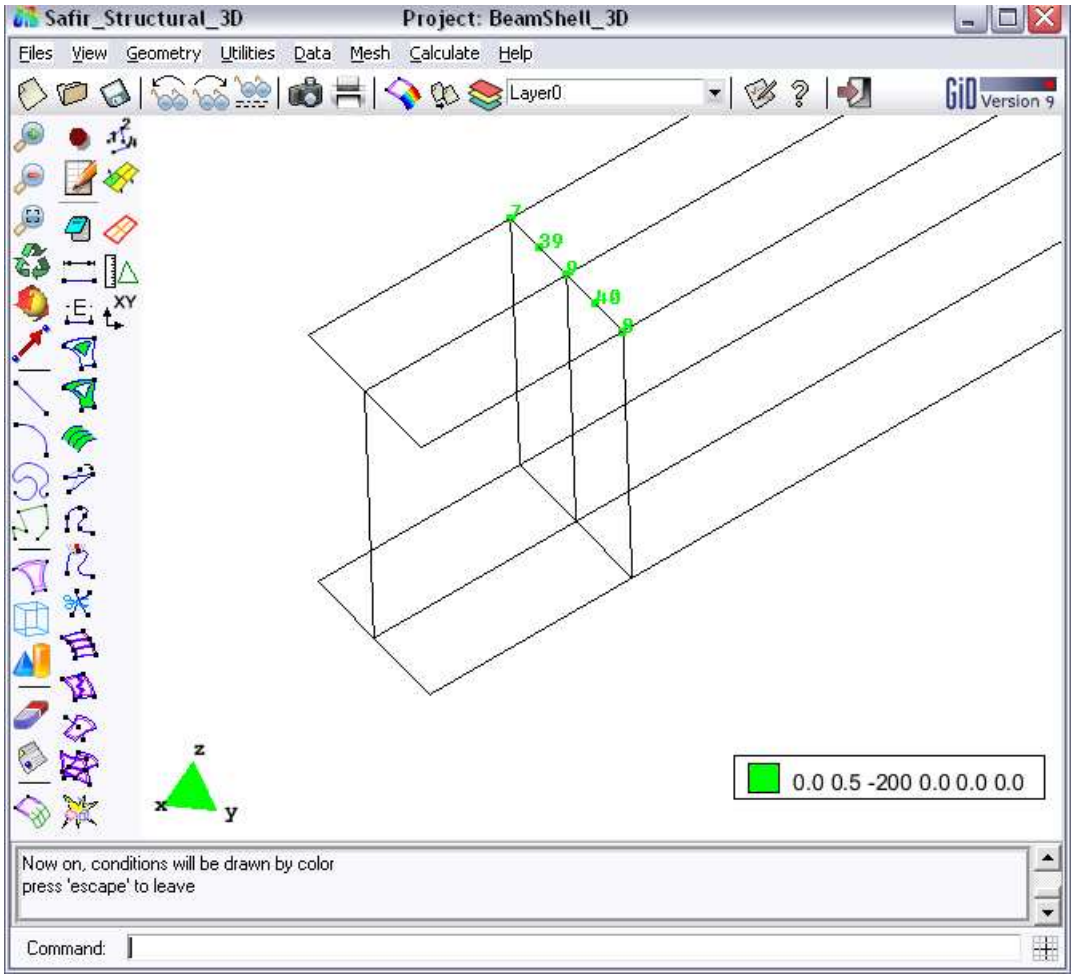
- ***Data->Loads***

And fill as shown below:



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

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



5. Assign temperature files (.TSH files)

The objective is to assign .tsh files named column_flange, column_web, beam_flange, beam_web and stiffener to system surfaces

Select from the pull down menu:

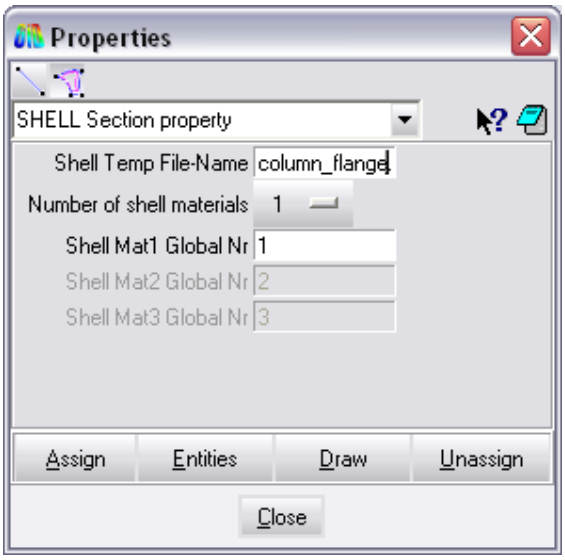
- ***Data-> Properties***

Select the shell tab put:

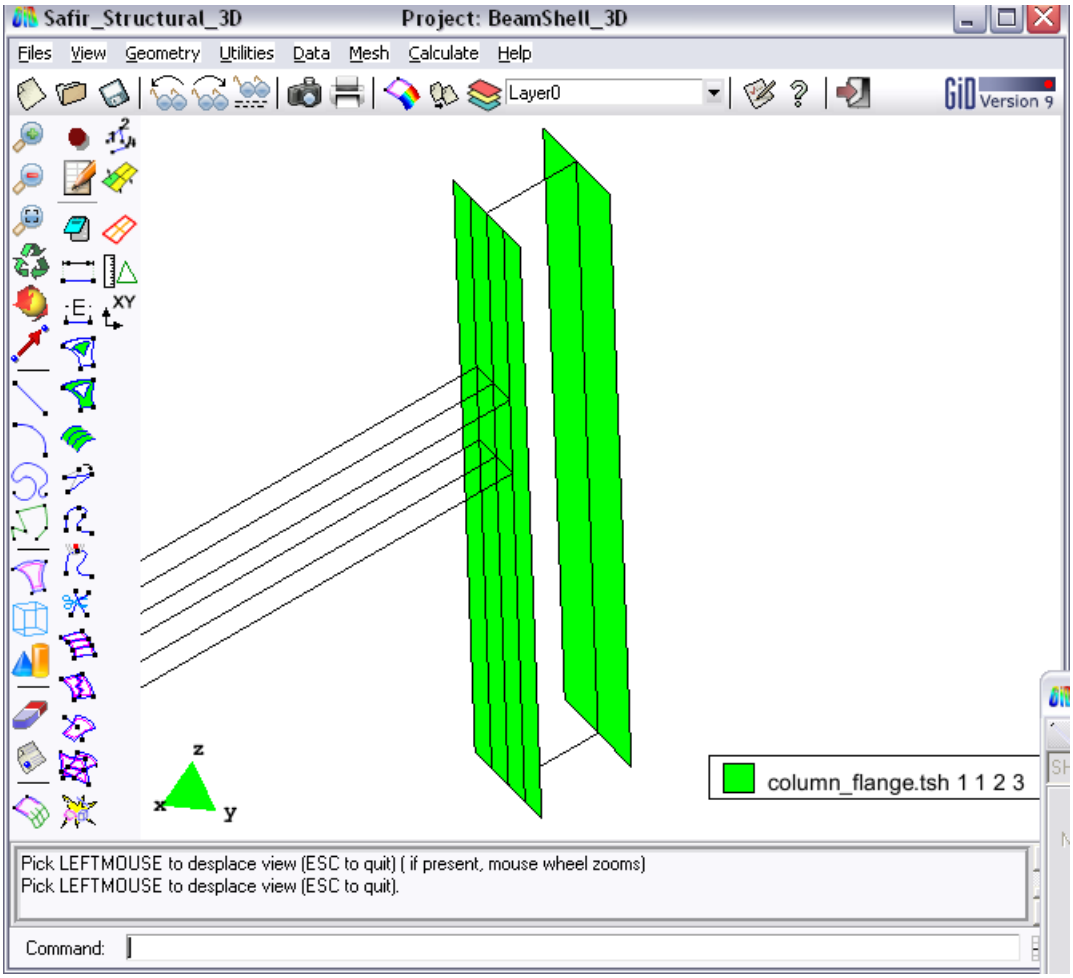
As shell temp file-name: ***column_flange.tsh***

As number of shell materials: ***1***

As shell mat1 global nr: ***1***



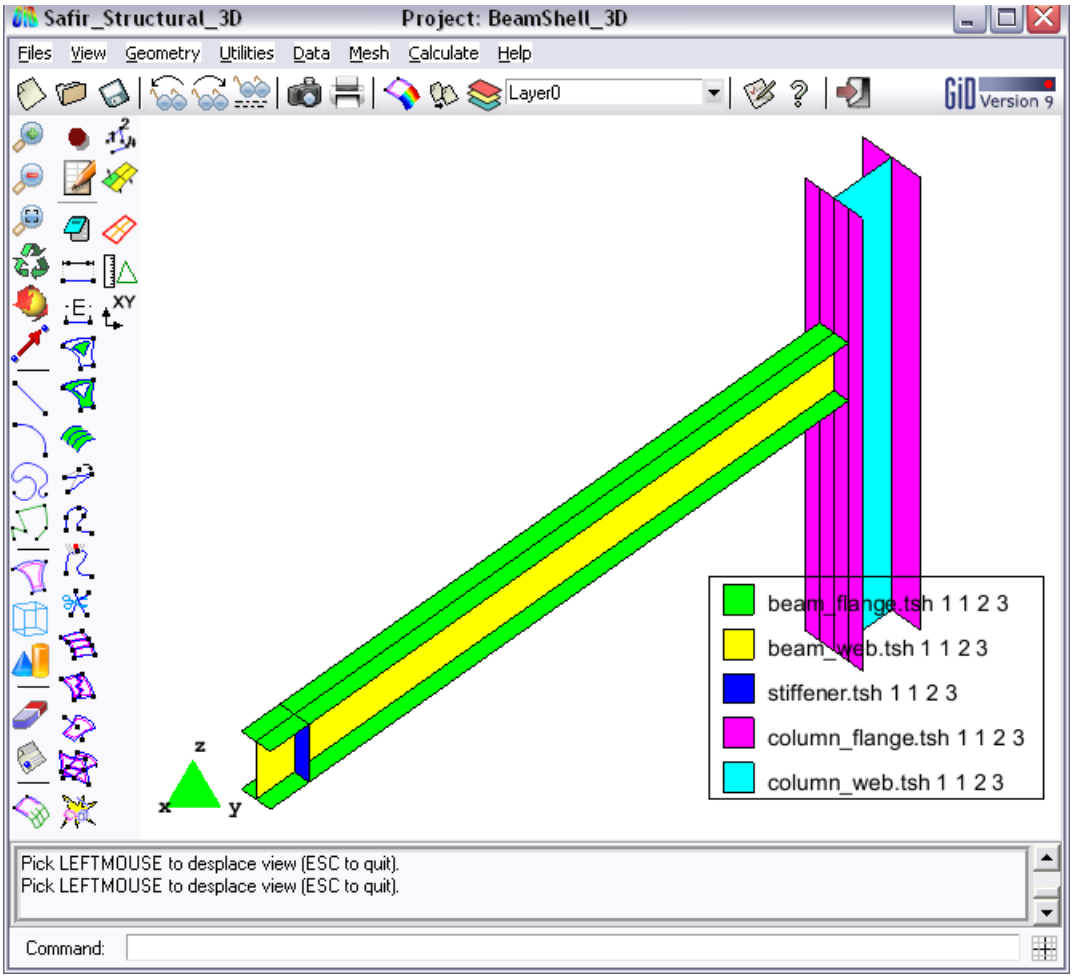
Click on ***Assign*** and select all column flanges corresponding surfaces and press ***[Esc]***



Do the same operation with column_web, beam_flange, beam_web and stiffener

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

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

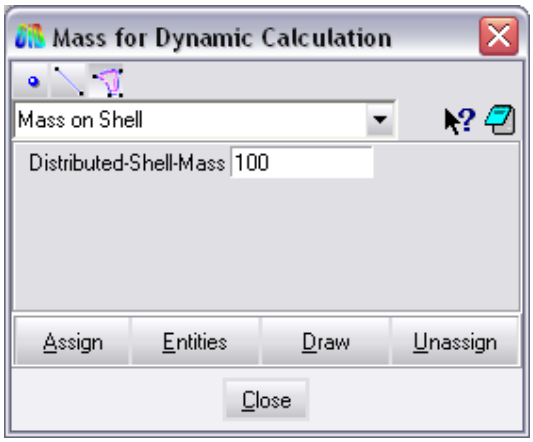


6. Define the Mass:

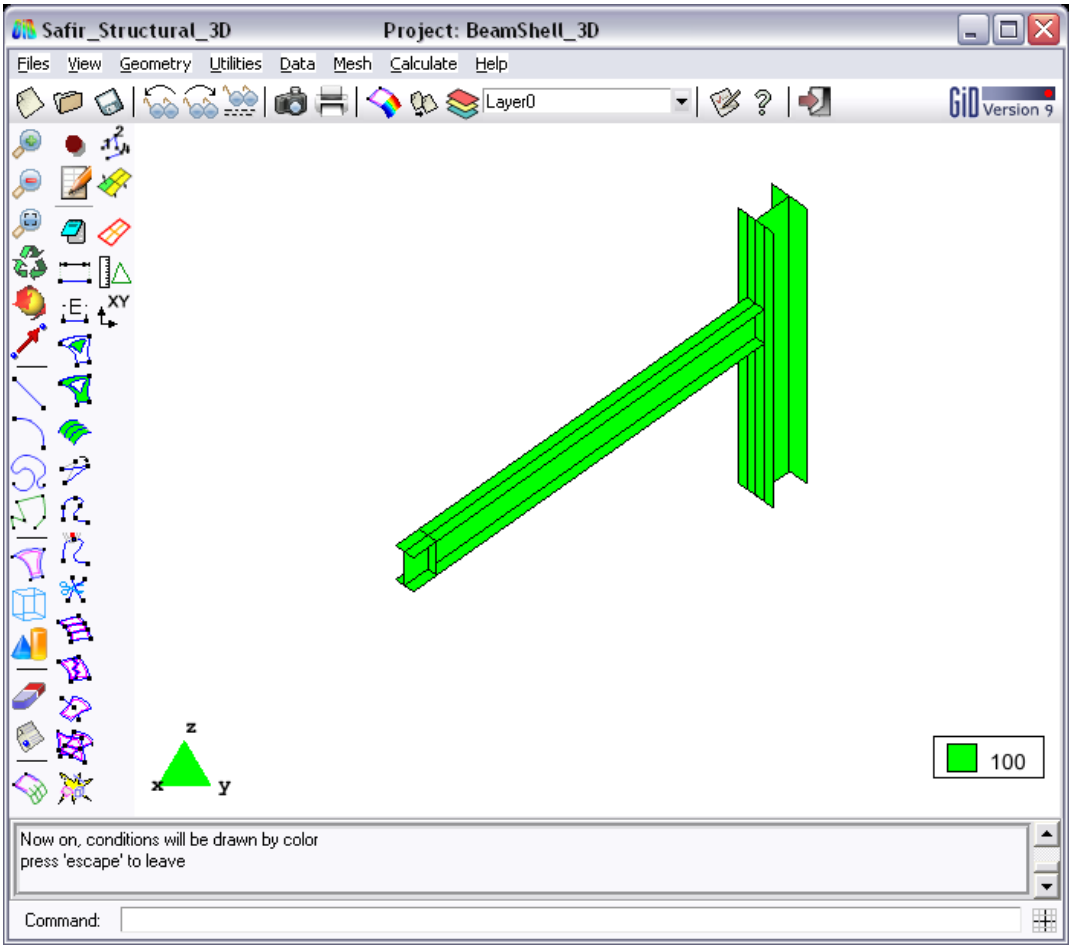
Select from the pull down menu:

- *Data-> Mass*

Select the Shell tab and fill as below



Click on ***Assign*** and select all surfaces of the structure and press ***[Esc]***

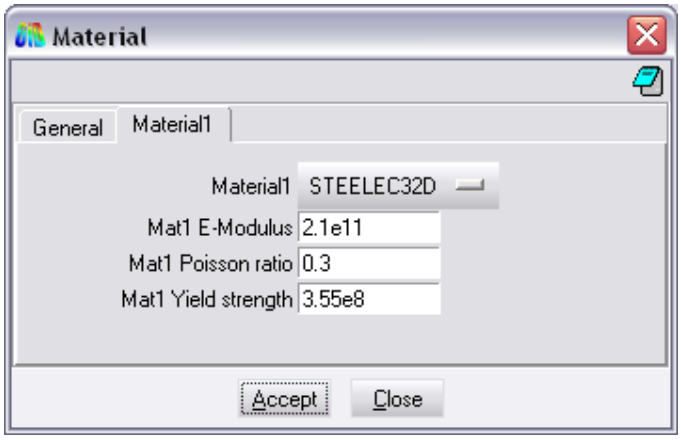


7. Define global materials:

To define material 1 select from the pull down menu:

- *Data->Material*

In the general tab, put one material. In the material1 tab, fill as shown below

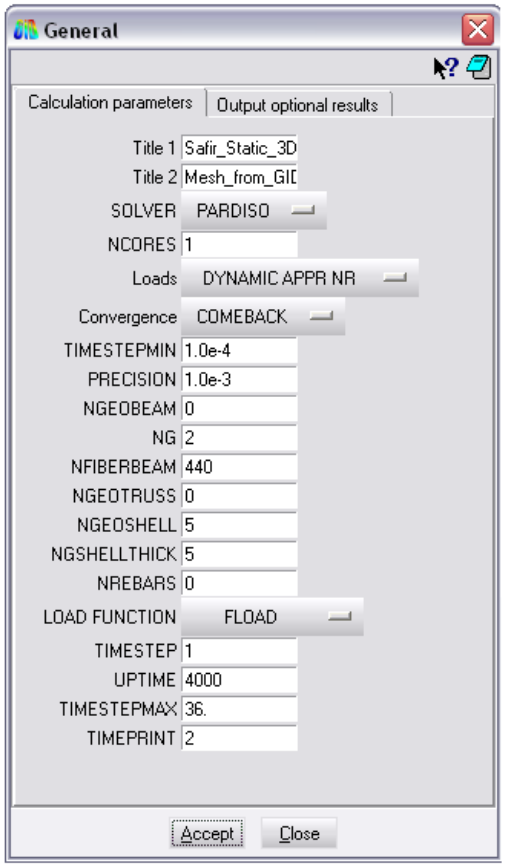


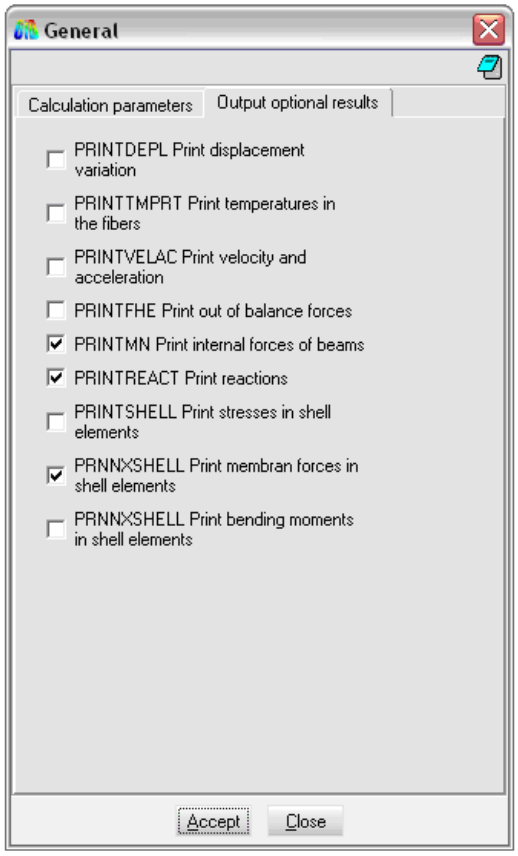
8. Define general data

Select from the pull down menu:

- *Data->Problem Data*

And fill as shown below





Enter the following

NGEOSHELL= 5 (the number of .tsh files)

NGSHELLTHICK = 5 (Number of point integration)

TIMSTEP, UPTIME, TIMEPRINT as needed

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

⚠ *Ngeoshell is the number of .TSH files (5 in this case)*

⚠ *The Postprocessor Diamond can't open a file bigger than 1.1 Go. It's important to choose your Timestep 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

9. Generate the mesh:

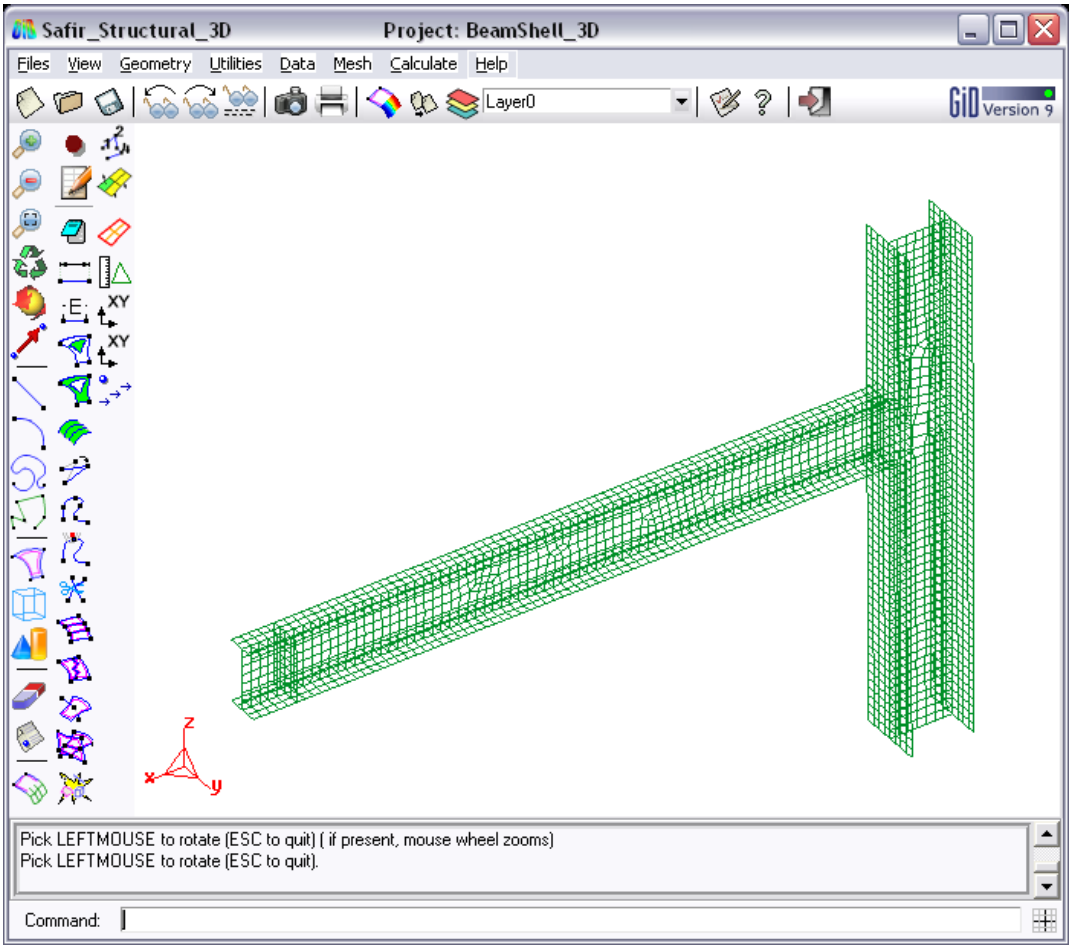
Select from the pull down menu:

- *Mesh->Element Type->Quadrilateral*

Select all the structure

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

Enter the element size of 0.04



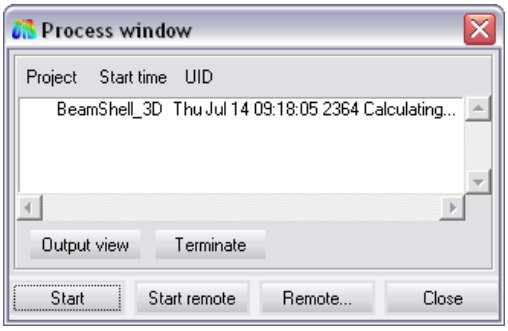
If you are not satisfied with the mesh repeat meshing and change the element size.

10. Start the calculation:

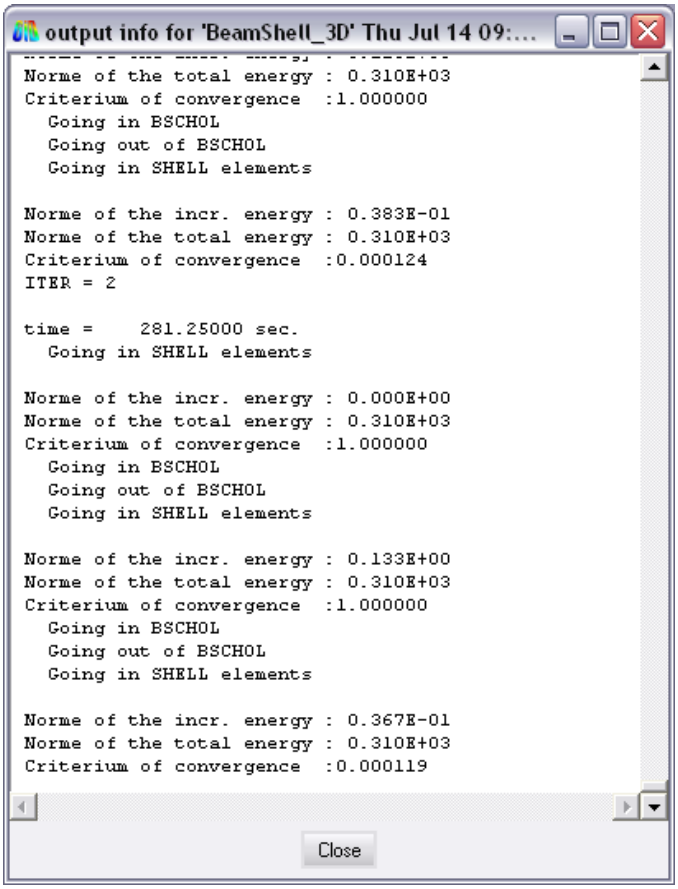
Copy/paste column_flange, column_web, beam_flange, beam_web and the stiffener .TSH files into the BeamShell_3D.gid diretory

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 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 .TSH files into the project directory, or if you entered a wrong number for NGEOSHELL.*

⚠ *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.*