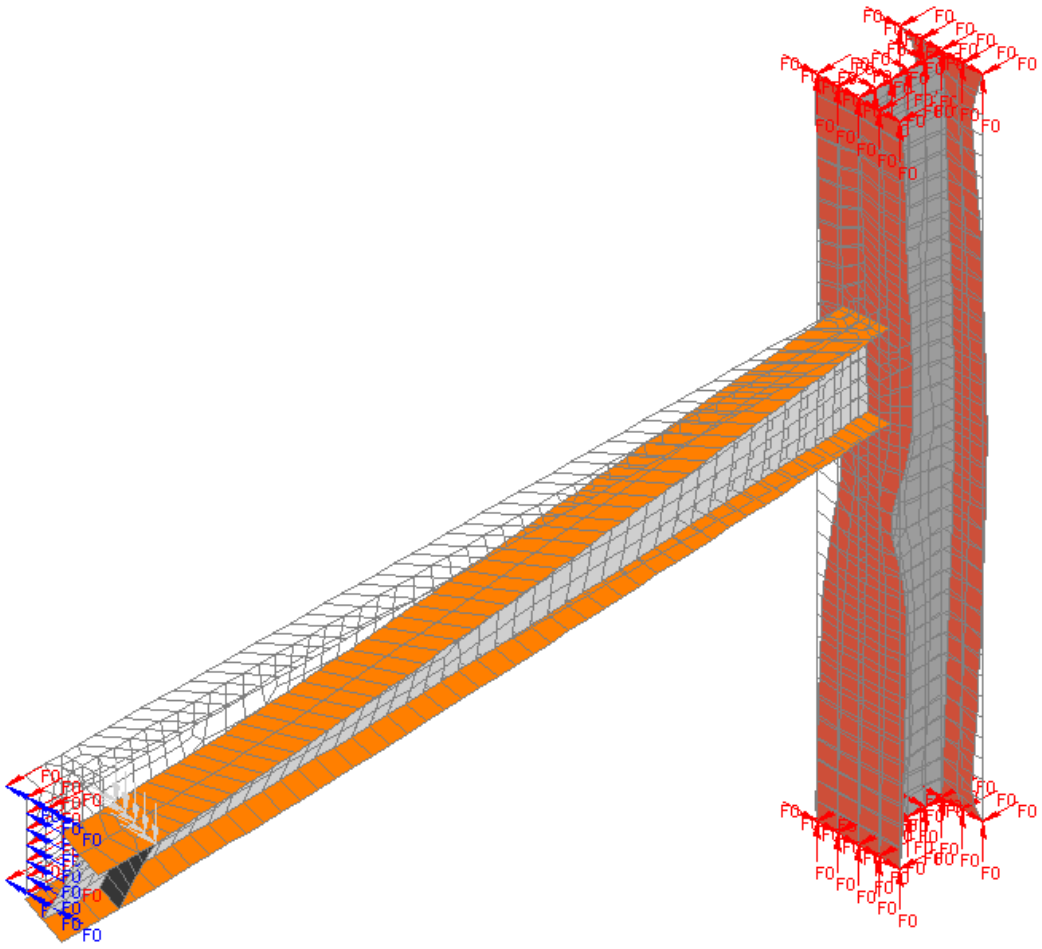
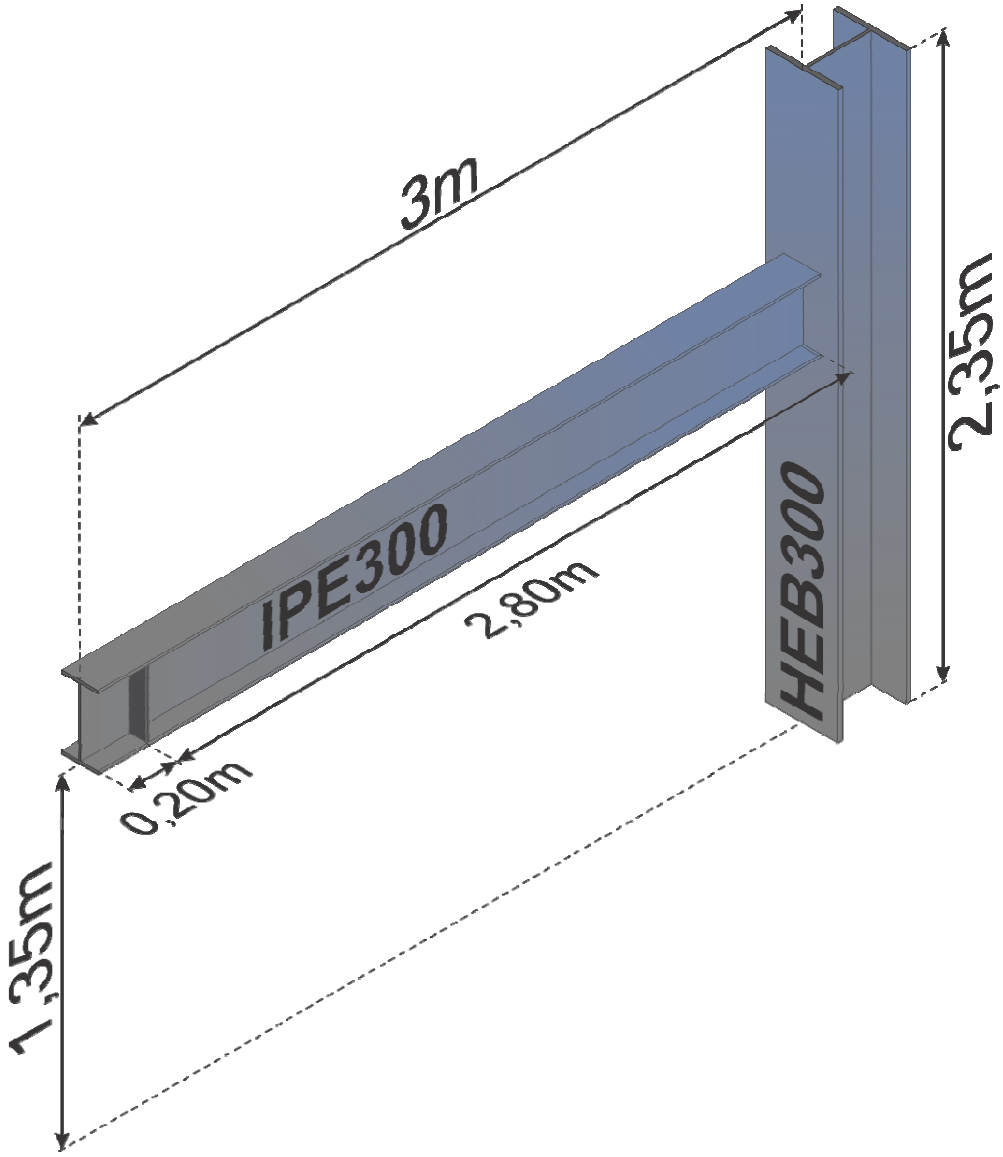
Example of GID-SAFIR

Structural Analysis

Exercise n°10 – 3D Beam-Column in Shell elements



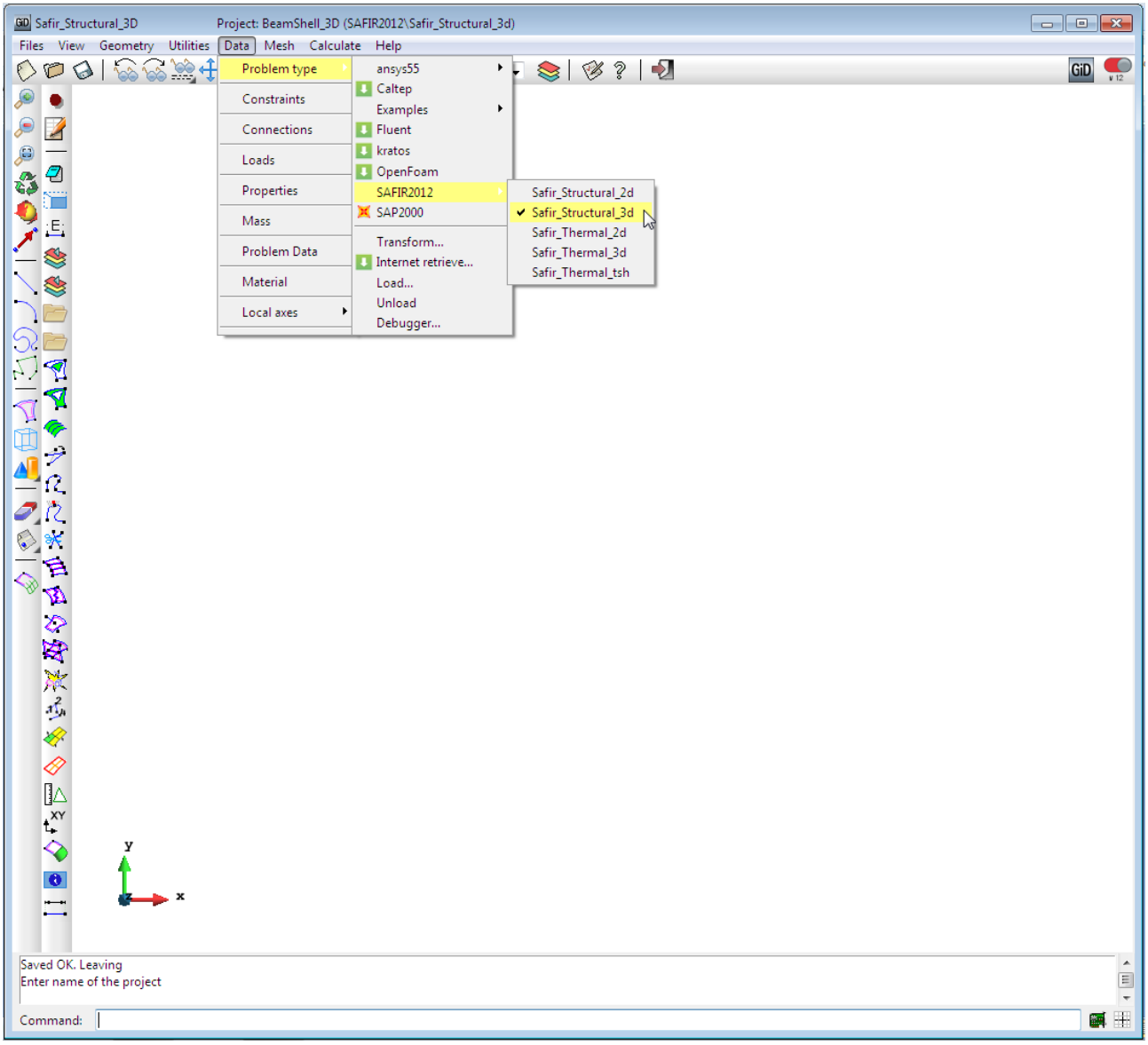
This exercise consists of a IPE300 beam connected to a HEB300 column.



1. Create a project for a 3D structural analysis

From the pull down menu select:

- ***Data->Problem type->SAFIRxxx->Safir_Structura_3d***



To save the project select :

- ***Files->Save***  ***or*** ***or [Ctrl + s]***

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

2. Create the system geometry

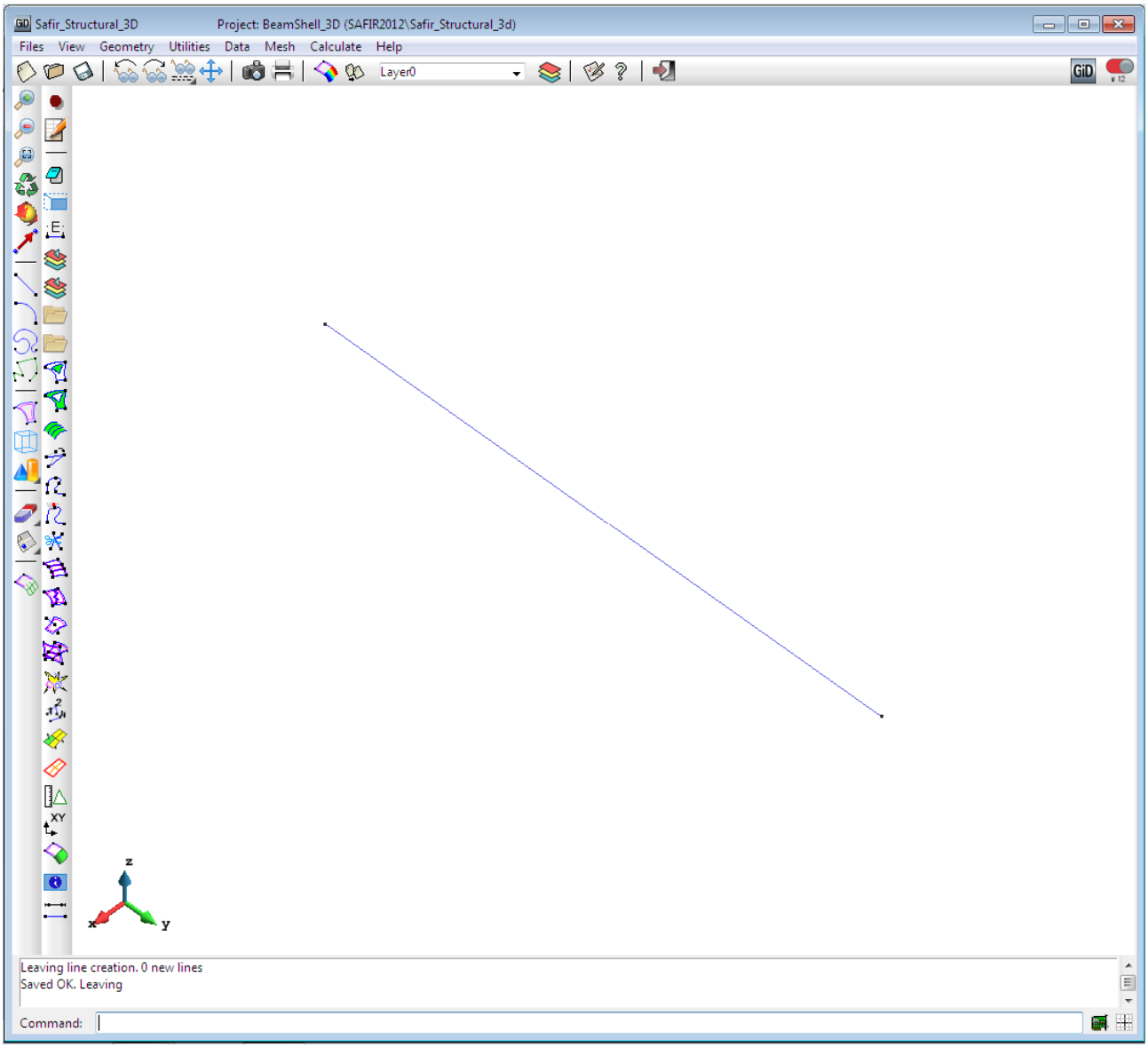
Change to the 3D isometric view. Select from the pull down menu :

- *View->Rotate->isometric*

Create the system lines:

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

To create the top flange of the IPE300 steel profile, enter in the command line (at the bottom of the windows) : *0,-0.075,0.15 @0,0.15,0* and press *Enter*, then *Esc.*

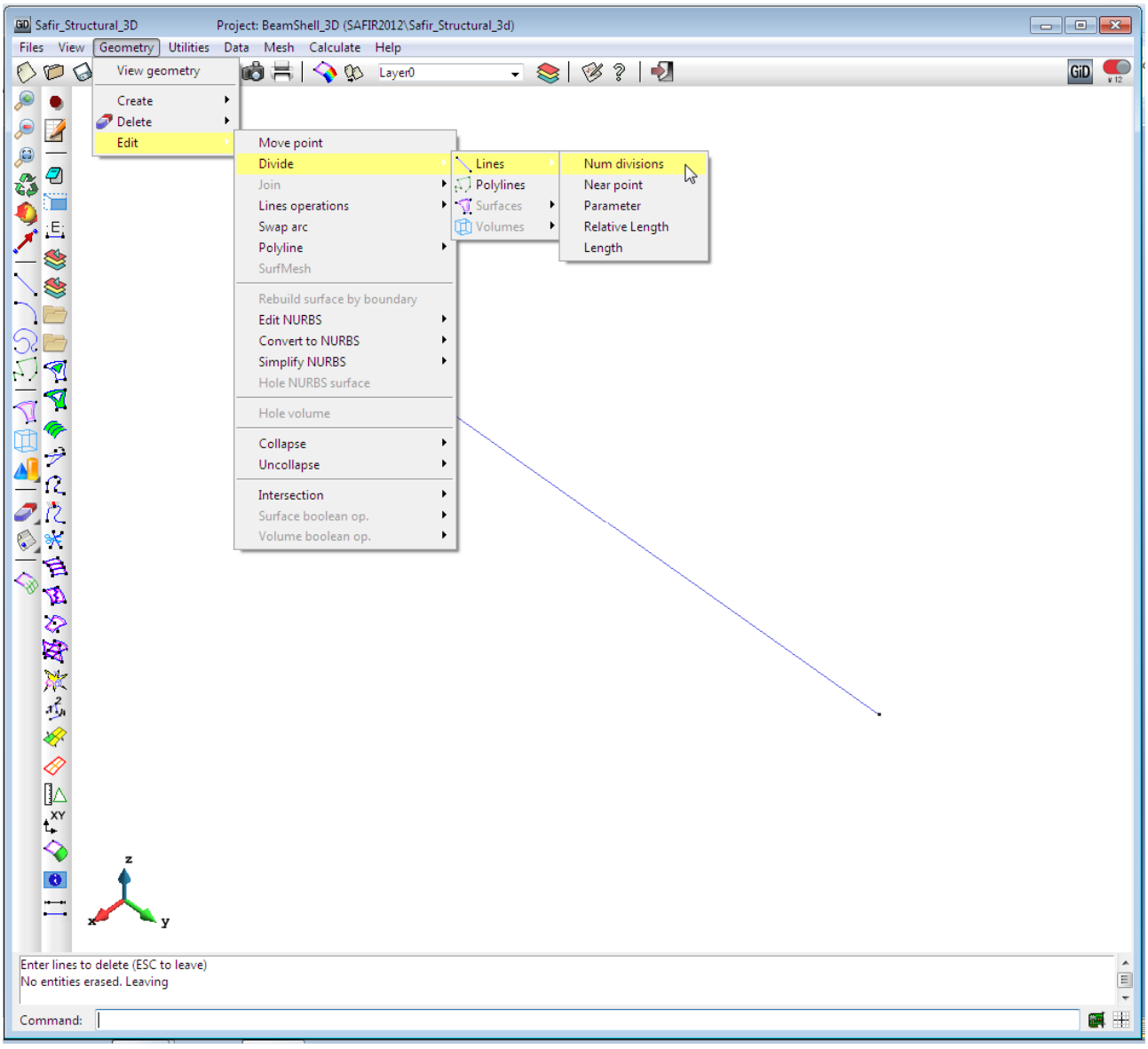


The top flange line is created.

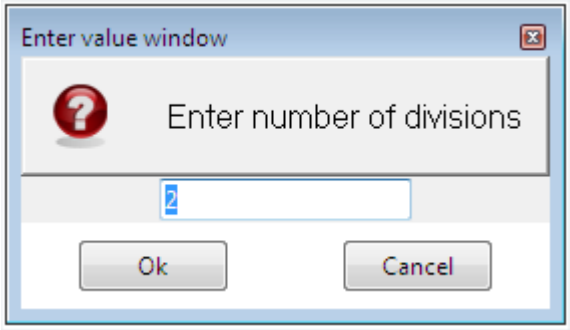
Divide this line in 2.

Select from the pull down menu:

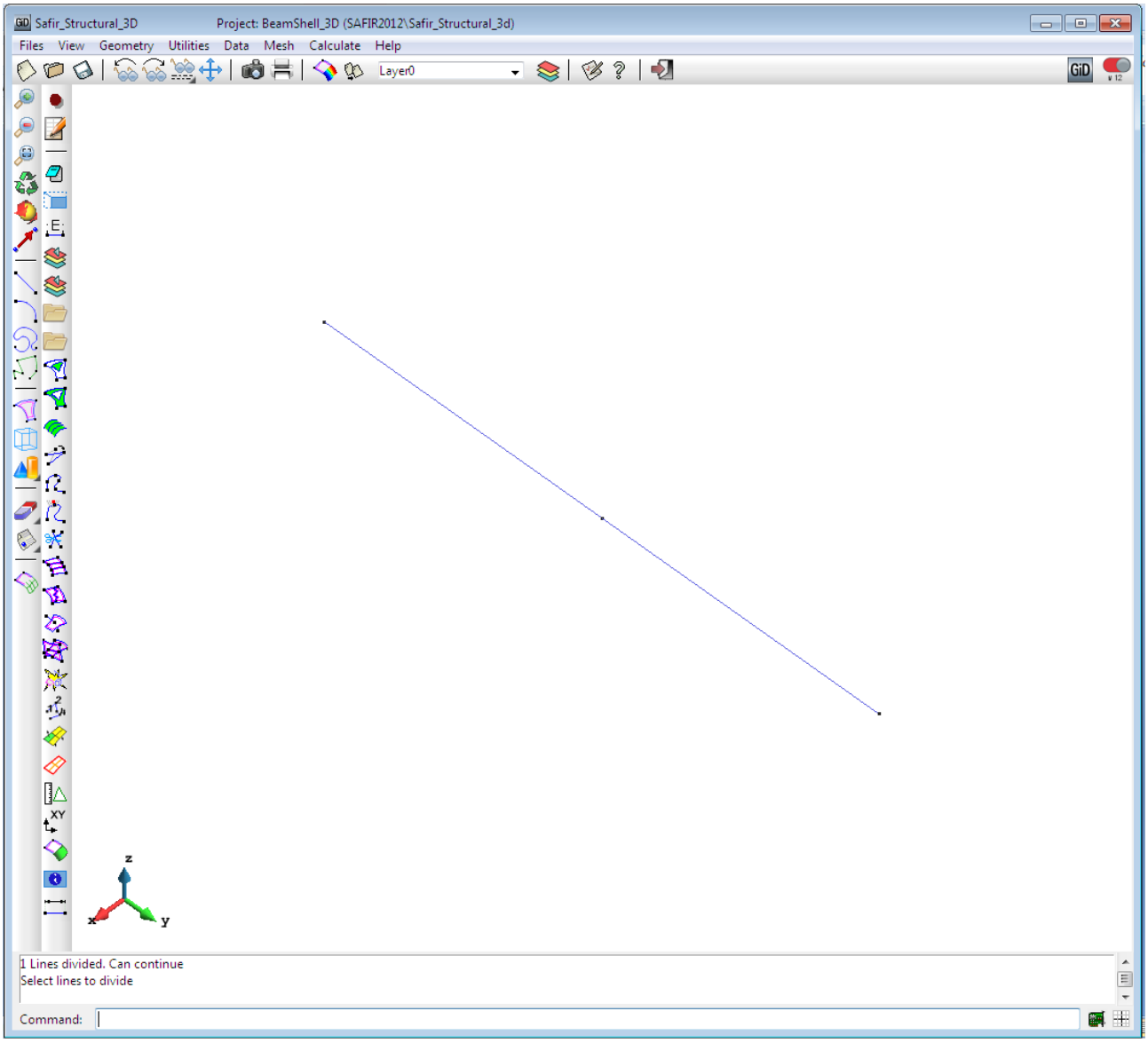
- ***Geometry->Edit-> Divide-> Lines-> Num Divisions***



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



Select the line and press *[Esc]* twice to leave the line division mode.

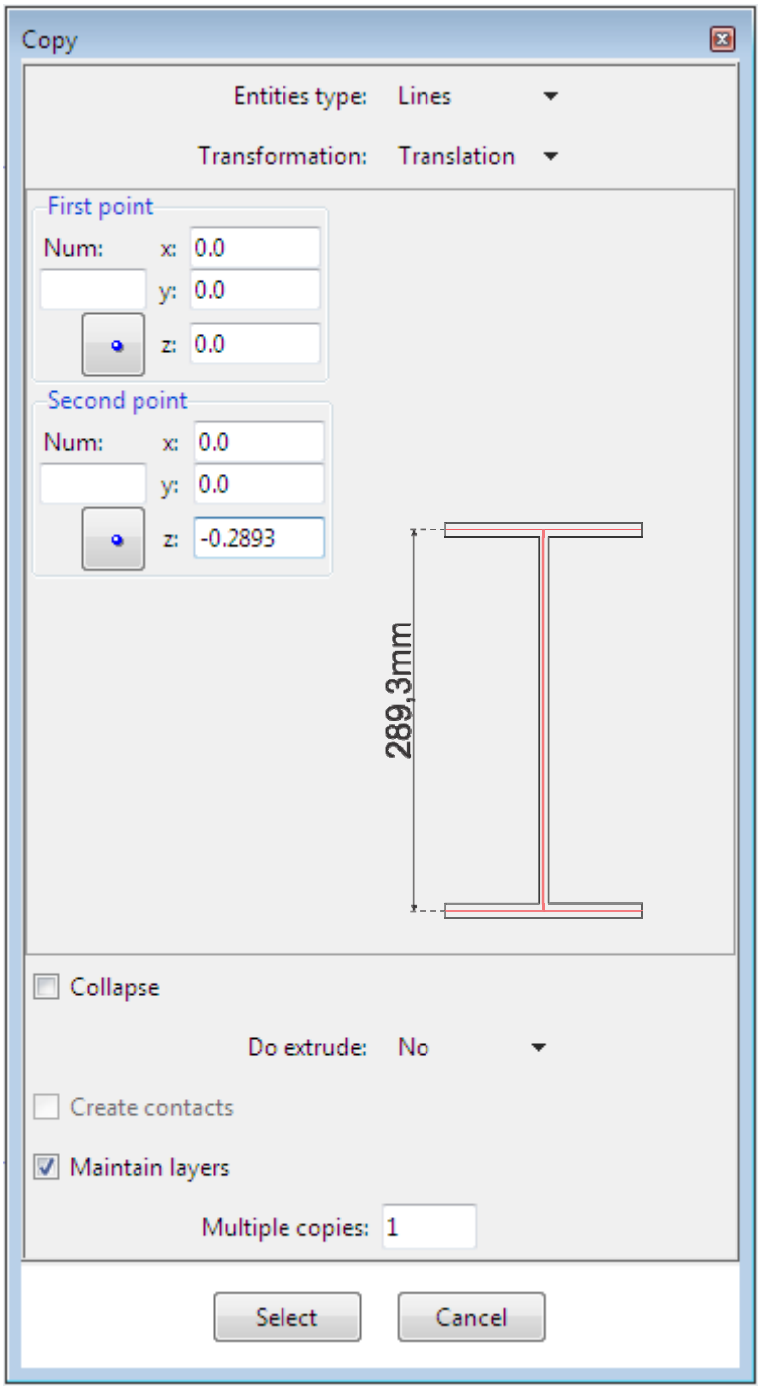


Copy this line to define the bottom flange of the profile.

Select from the pull down menu:

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

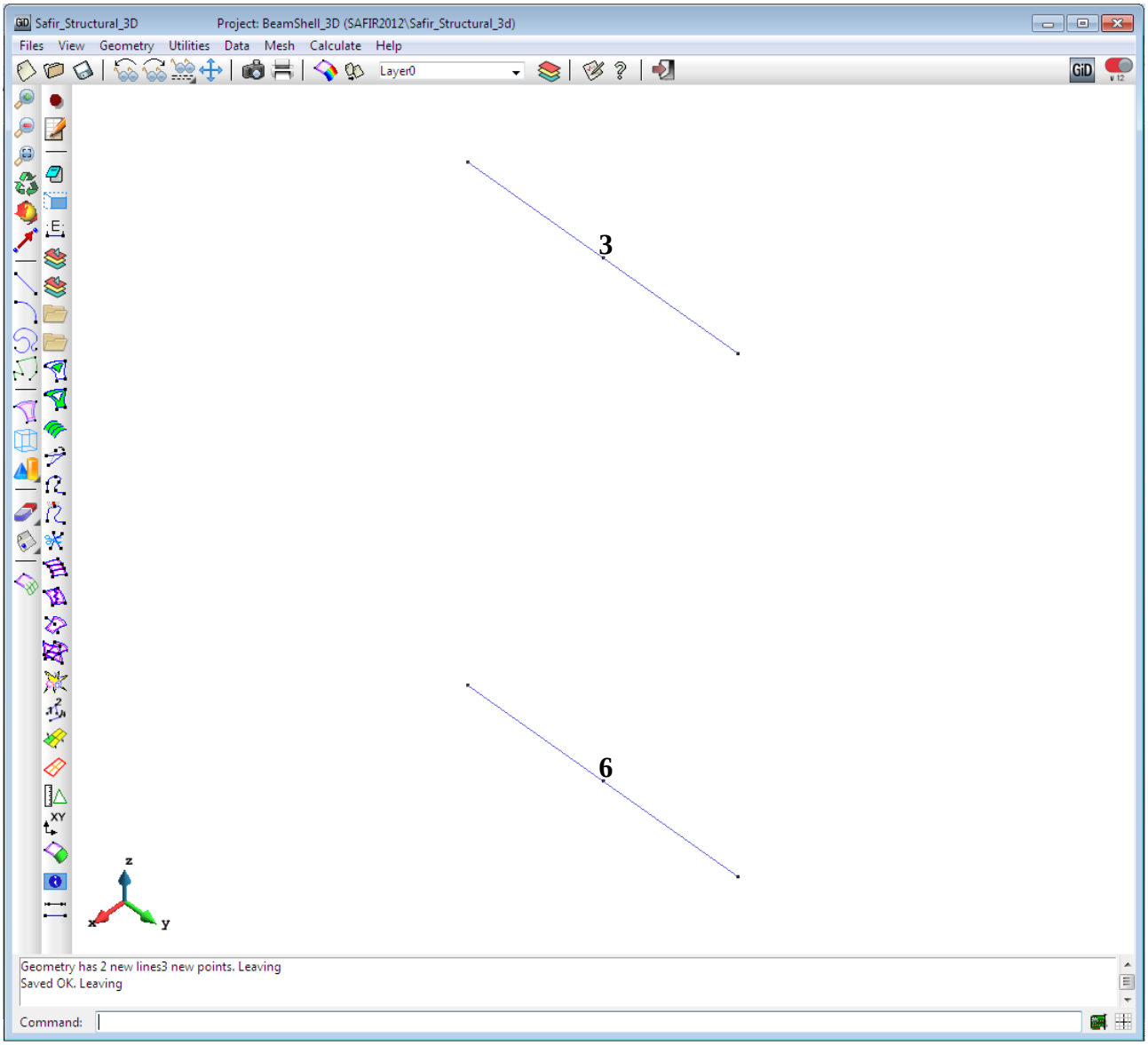
Fill the dialog box as shown below:



As *Entities type*, select: ***Lines***

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

Select all lines , press ***Finish*** then ***Cancel.***

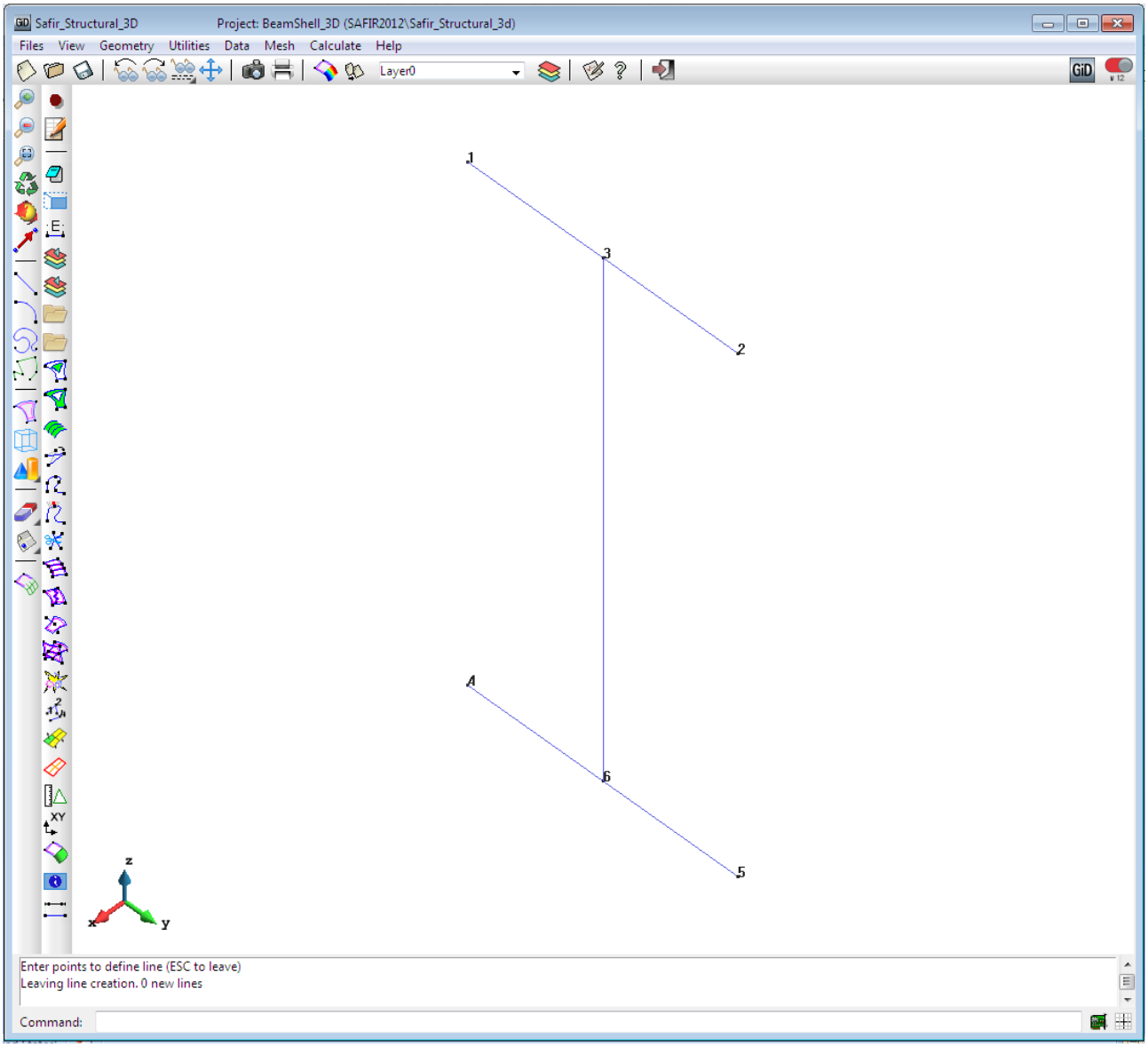


To create the web line, connect point 3 and 6.

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*** twice to leave this mode.



Now create the part of the beam going from the stiffener to the column flange.

Select from the pull down menu:

- ***Utilities->Copy***

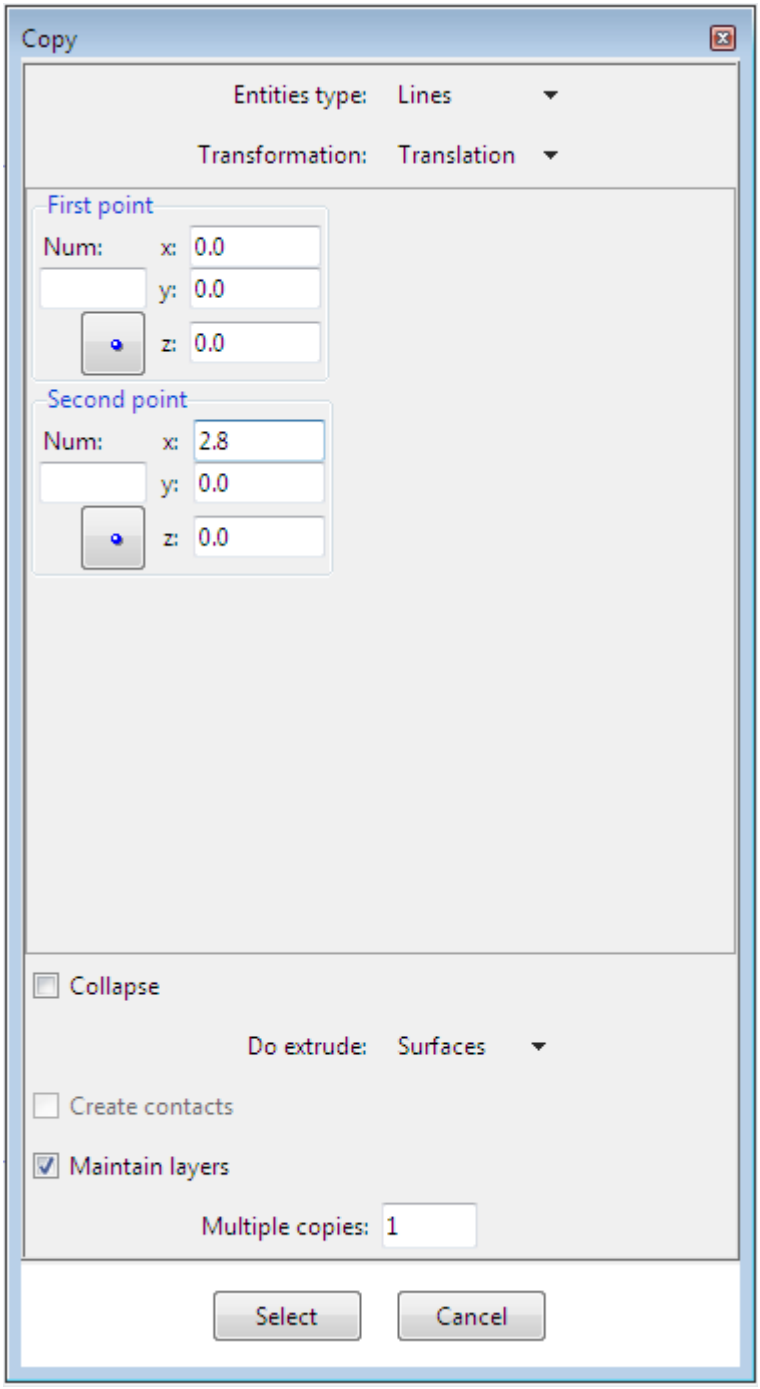
For *Entities type*, select : ***Lines***

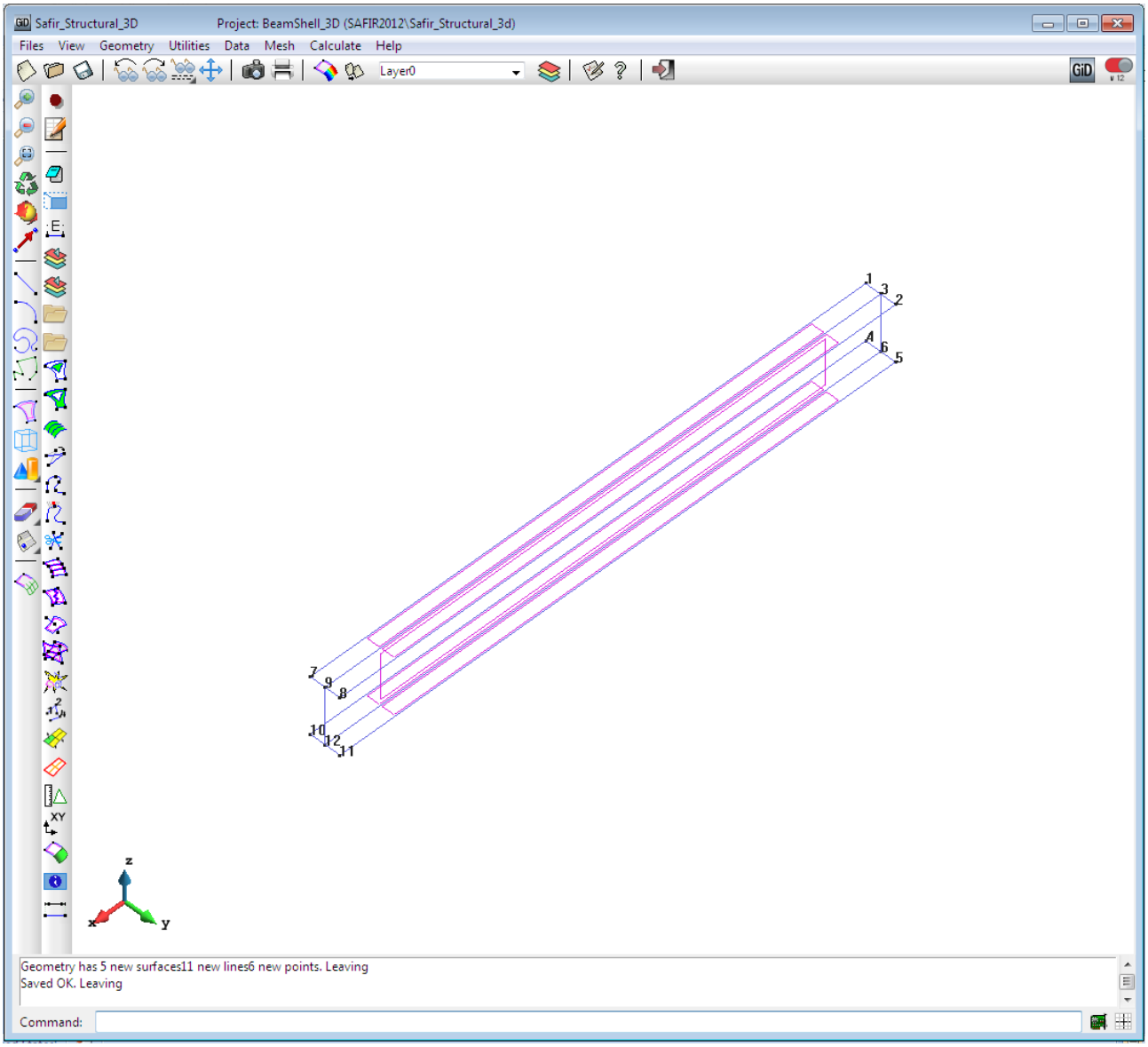
As for *Second point*, put: ***x = 2.8***

As for *Do extrude*, select : **Surfaces**

As for *Multiple copies*, put : **1**

Select all lines and press ***Finish***, then ***Cancel***.


Copy
Entities type: Lines
Transformation: Translation
First point
Num:
x: 0.0
y: 0.0
z: 0.0
Second point
Num:
x: 2.8
y: 0.0
z: 0.0
Collapse
Do extrude: Surfaces
Create contacts
Maintain layers
Multiple copies: 1
Select
Cancel

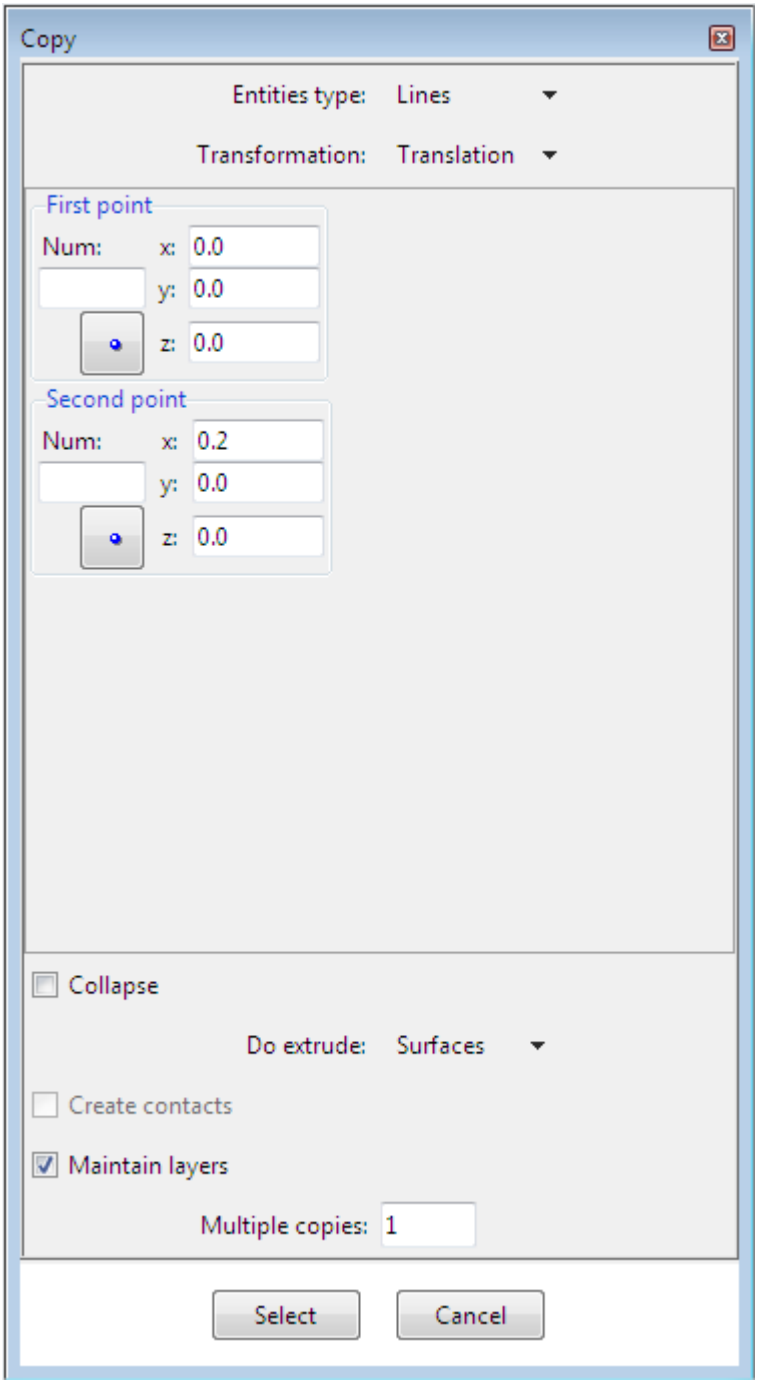

Safir_Structural_3D
Project: BeamShell_3D (SAFIR2012\Safir_Structural_3d)
Files View Geometry Utilities Data Mesh Calculate Help
Layer0
1 3 2
4 6 5
7 9 8
10 12 11
z
x
y
Geometry has 5 new surfaces11 new lines6 new points. Leaving
Saved OK. Leaving
Command:

Now create the part of the beam going from the stiffener to the beam end (0,20m).

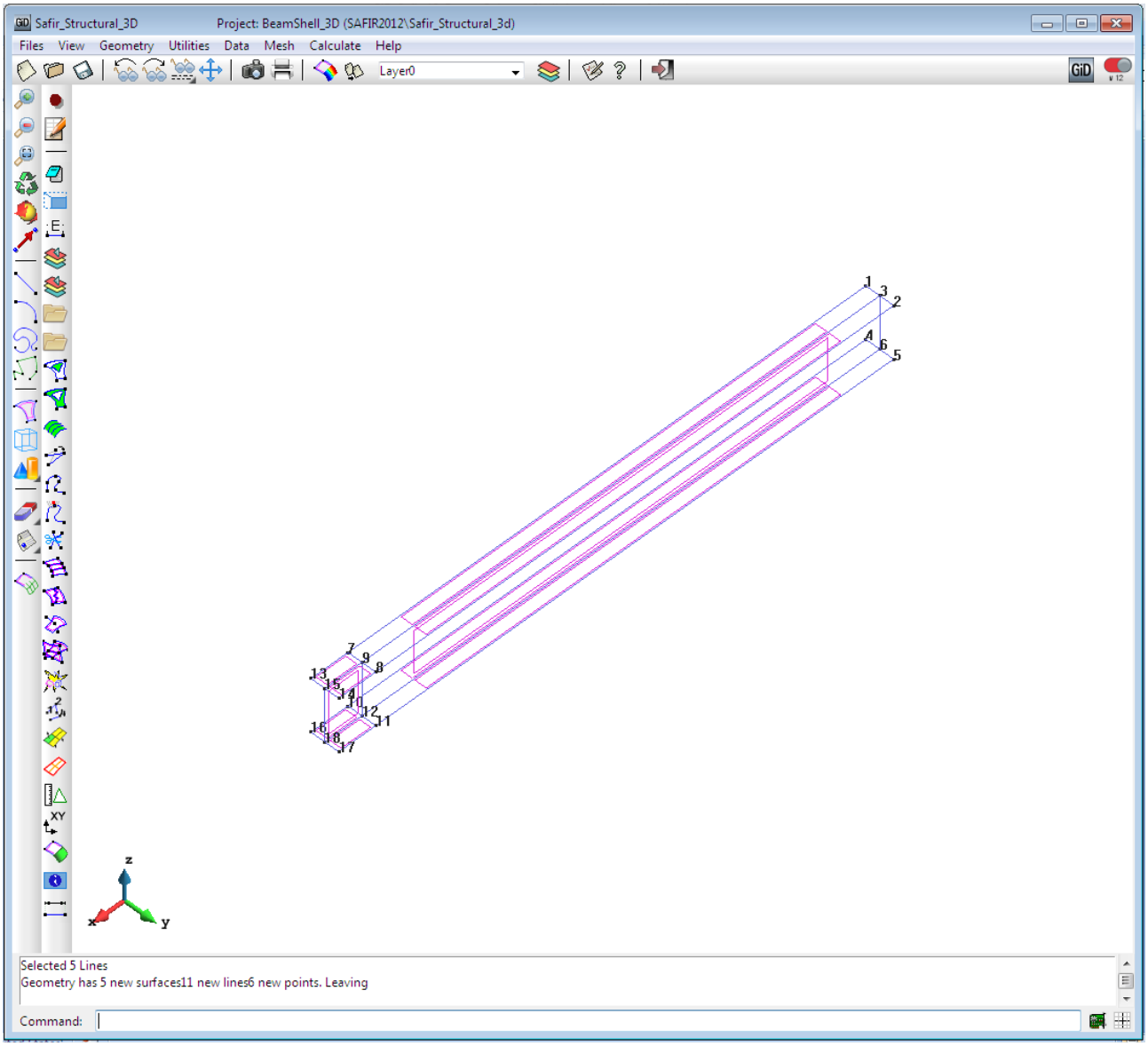
Select from the pull down menu:

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

  Fill as below:



Select the lines ***7-9-8***, ***10-12-11***, ***9-12*** and press ***Finish***, then ***Cancel***.



The IPE300 beam is modelised.

Now to create the stiffener, connect points 7 to 10 and 8 to 11.

Select from the pull down menu:

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

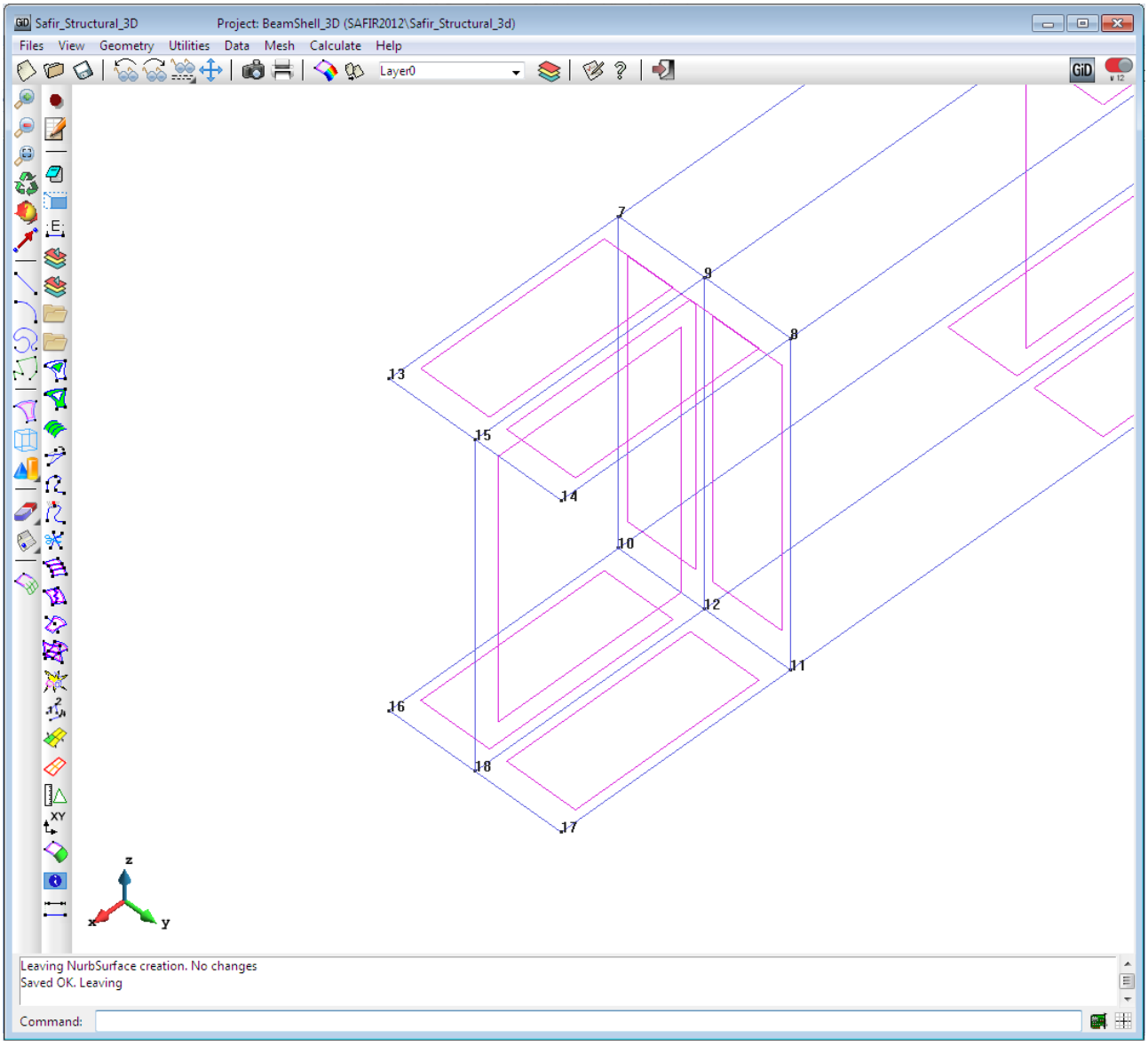
Press *[Ctrl + a]* and select the node *7*, then the node *10* and press *[Esc]*, then select nodes *8* and *11* and press *[Esc]* twice.

To create the surfaces which will model the stiffener, select from the pull down menu:

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

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

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



The stiffener is modelised.

To create the column, select from the pull down menu:

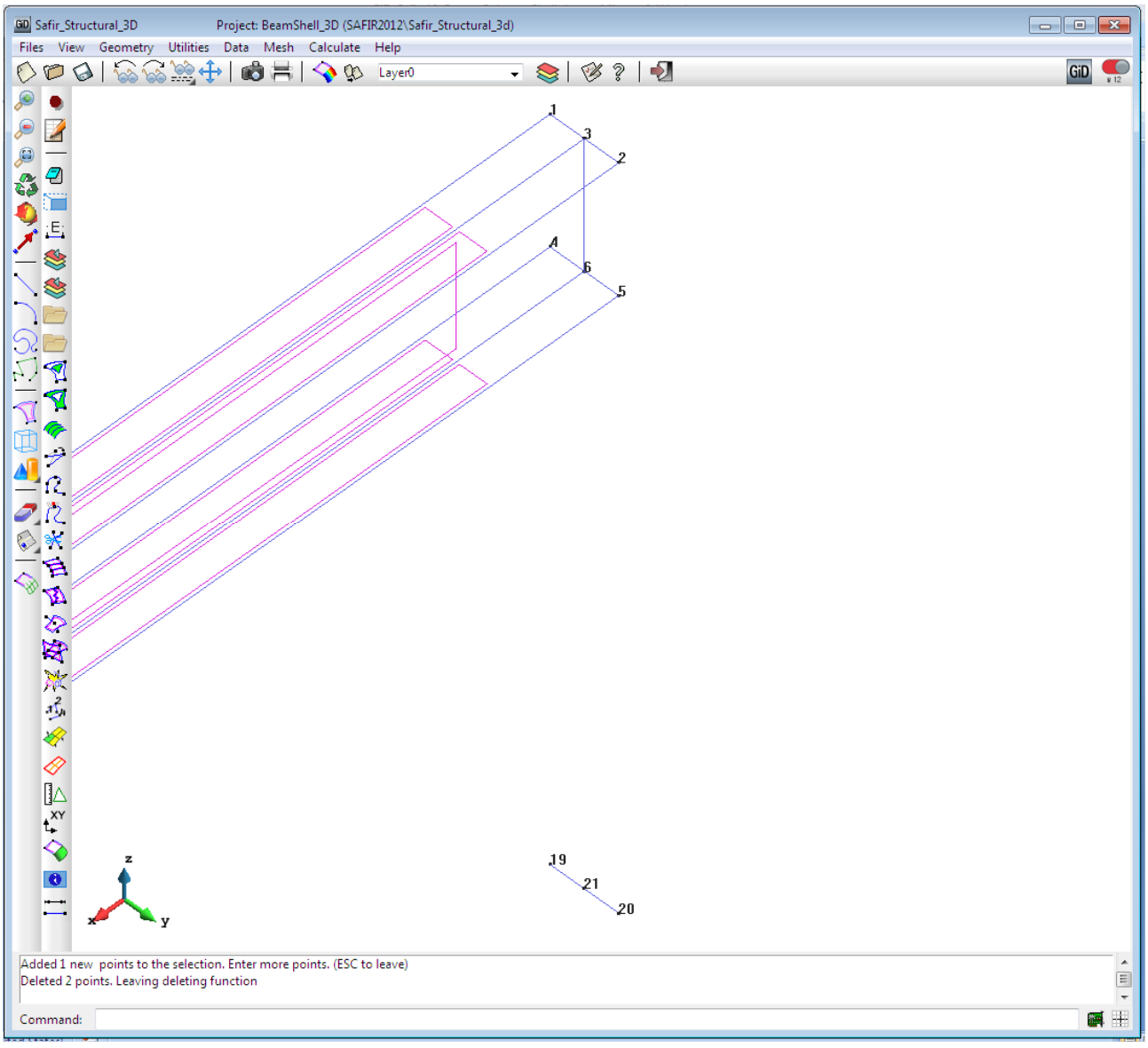
- ***Utilities->Copy*** ***or [Ctrl + c]***

Fill the *Copy* dialog box as below:

For *Entities type*, select : ***Lines***

For *Second point*, put : ***z = -1.35***

Select lines ***4-6*** and ***6-5***, and press ***[Finish]*** , then ***[Cancel]***.



Now create the points which will determine the width of the HEB300 column flange.

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

Fill the *Copy* dialog box as below:

For *Entities type*, select : ***Points***

For *Second point*, put : ***y = 0.075***

For *Do extrude*, select : ***Lines***

Select point ***20*** and press ***[Finish]***.

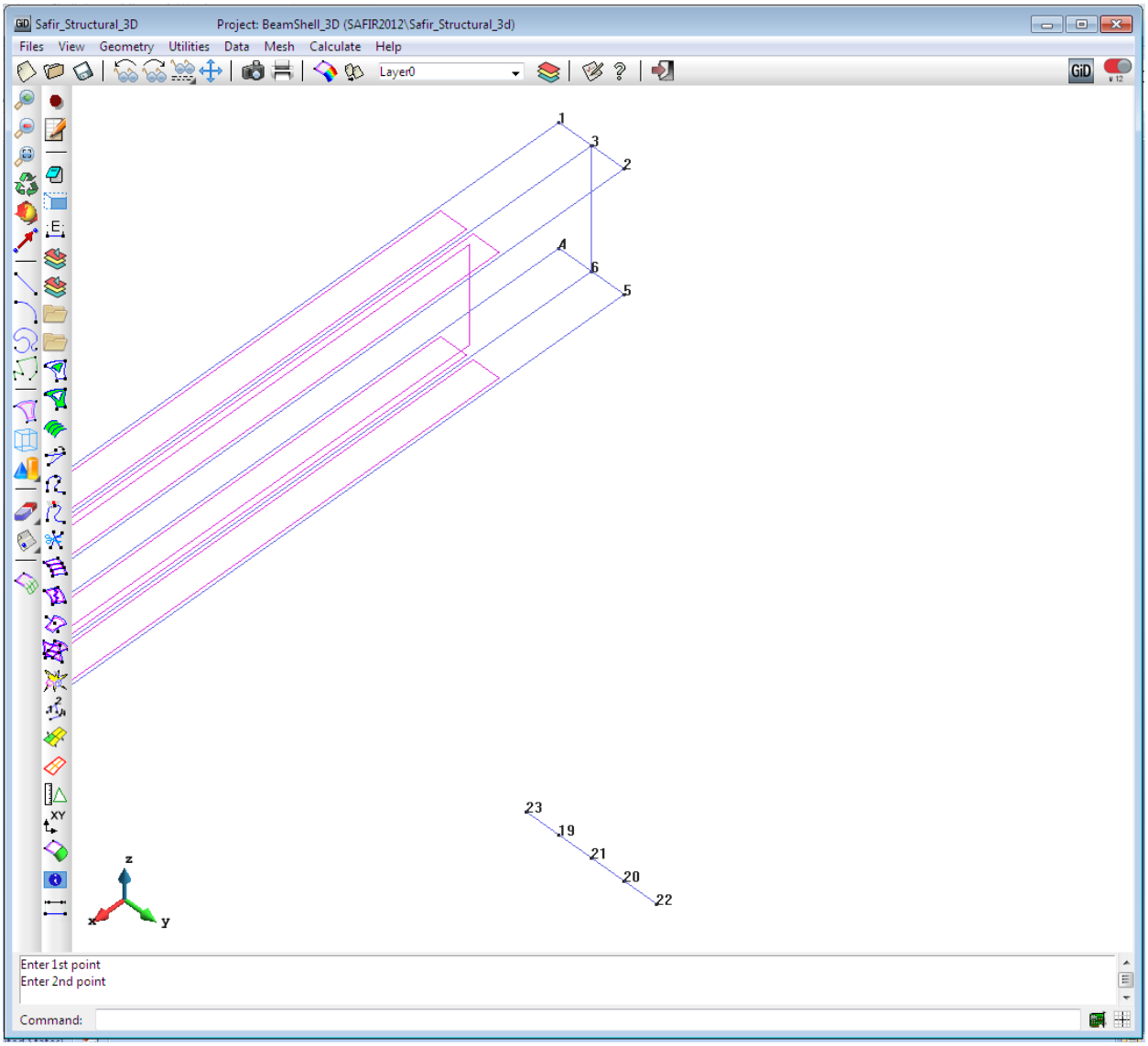
Then, for the other side :

For *Entities type*, select : ***Points***

For *Second point*, put : ***y = -0.075***

For *Do extrude*, select : ***Lines***

Select point ***19*** and press ***[Finish]***, then ***[Cancel]***.



To create the opposite flange of the column, select from the pull down menu:

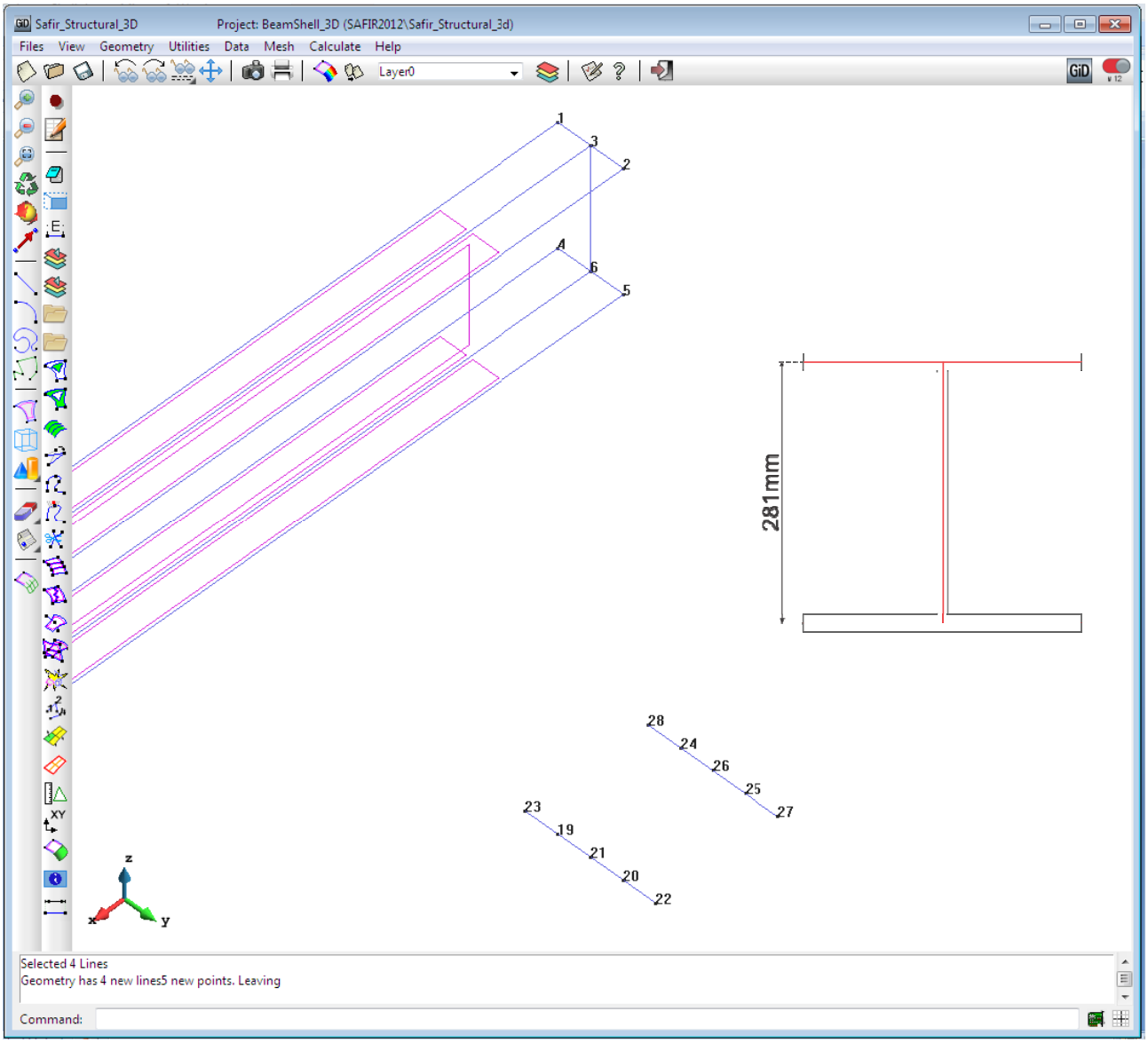
- ***Utilities->Copy*** ***or [Ctrl + c]***

Fill the *Copy* dialog box as below:

For *Entities type*, select : ***Lines***

For *Second point*, put : ***x = -0.281***

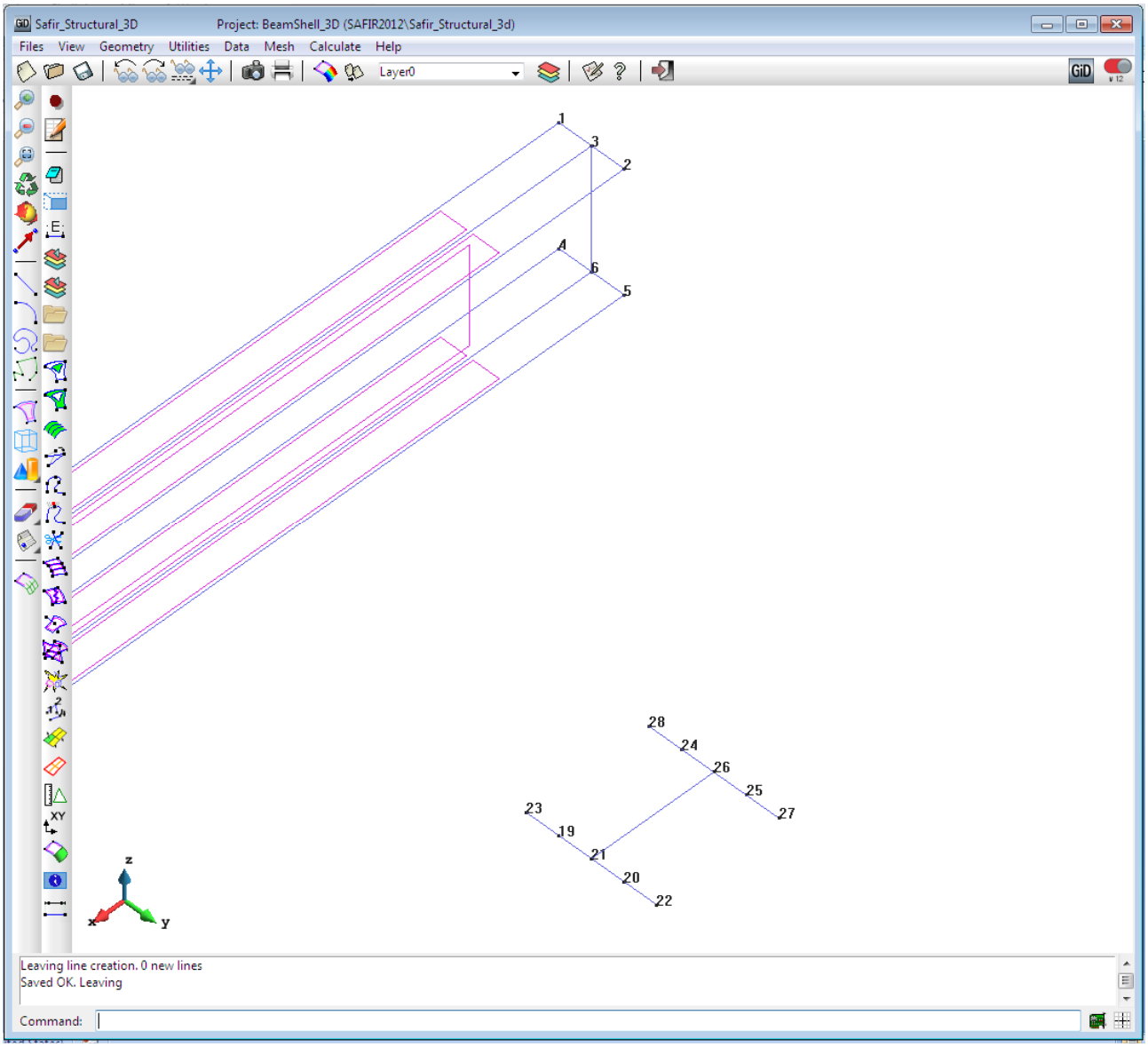
Select lines ***22-23*** and press ***[Finish]*** , then ***[Cancel]***.



To create the web line of the column, connect the points ***21*** and ***26*** :

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

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



The bottom end of the column is modelised.

Now create the top end of the column.

Select from the pull down menu:

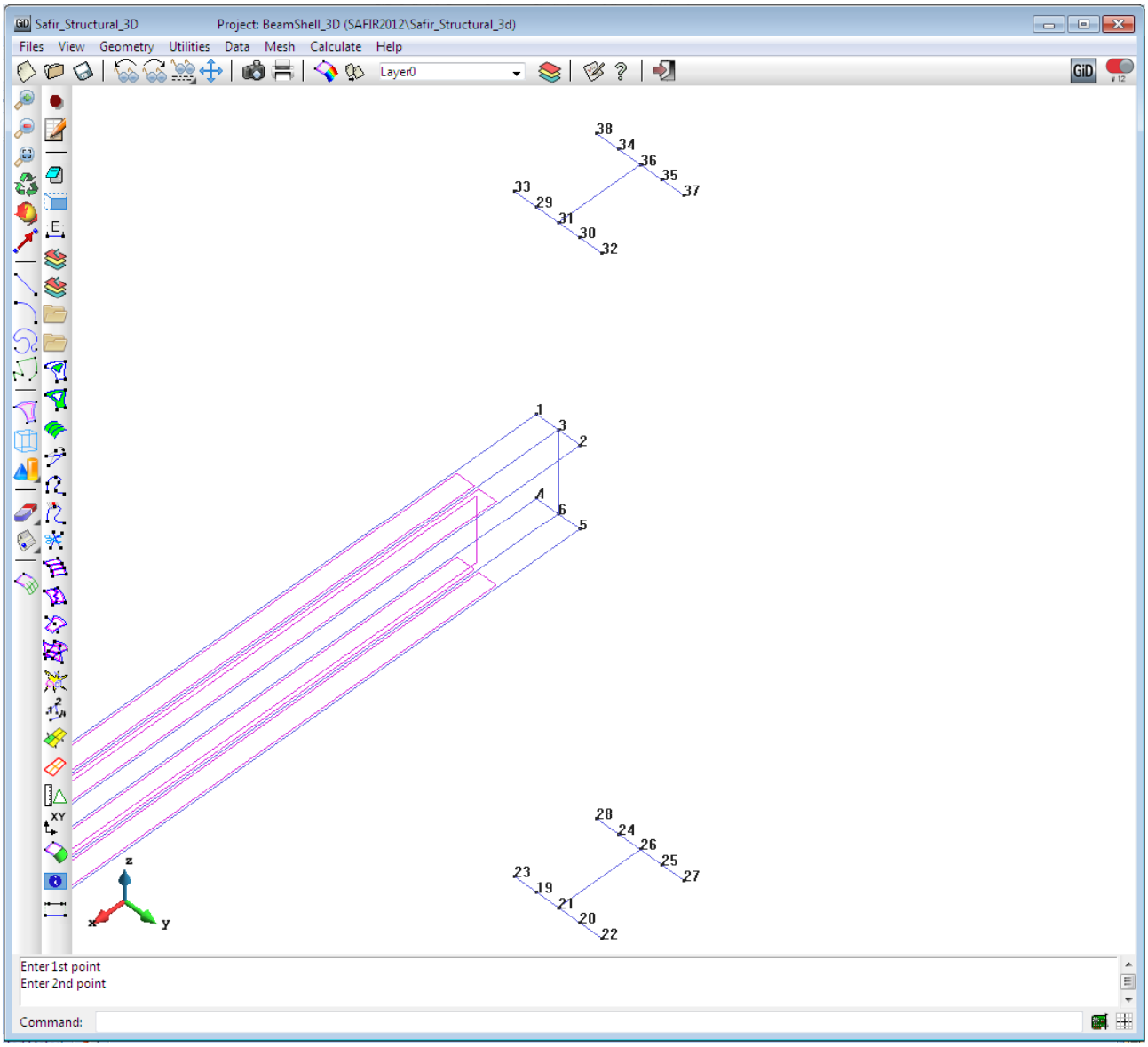
- ***Utilities->Copy*** ***or [Ctrl + c]***

Fill the *Copy* dialog box as below:

For *Entities type*, select : ***Lines***

For *Second point*, put : ***x = -2.35***

Select the bottom part of the column you just created (lines ***22 to 23***; ***21-26*** and ***27 to 28***) and press ***[Finish]***, then ***[Cancel].***

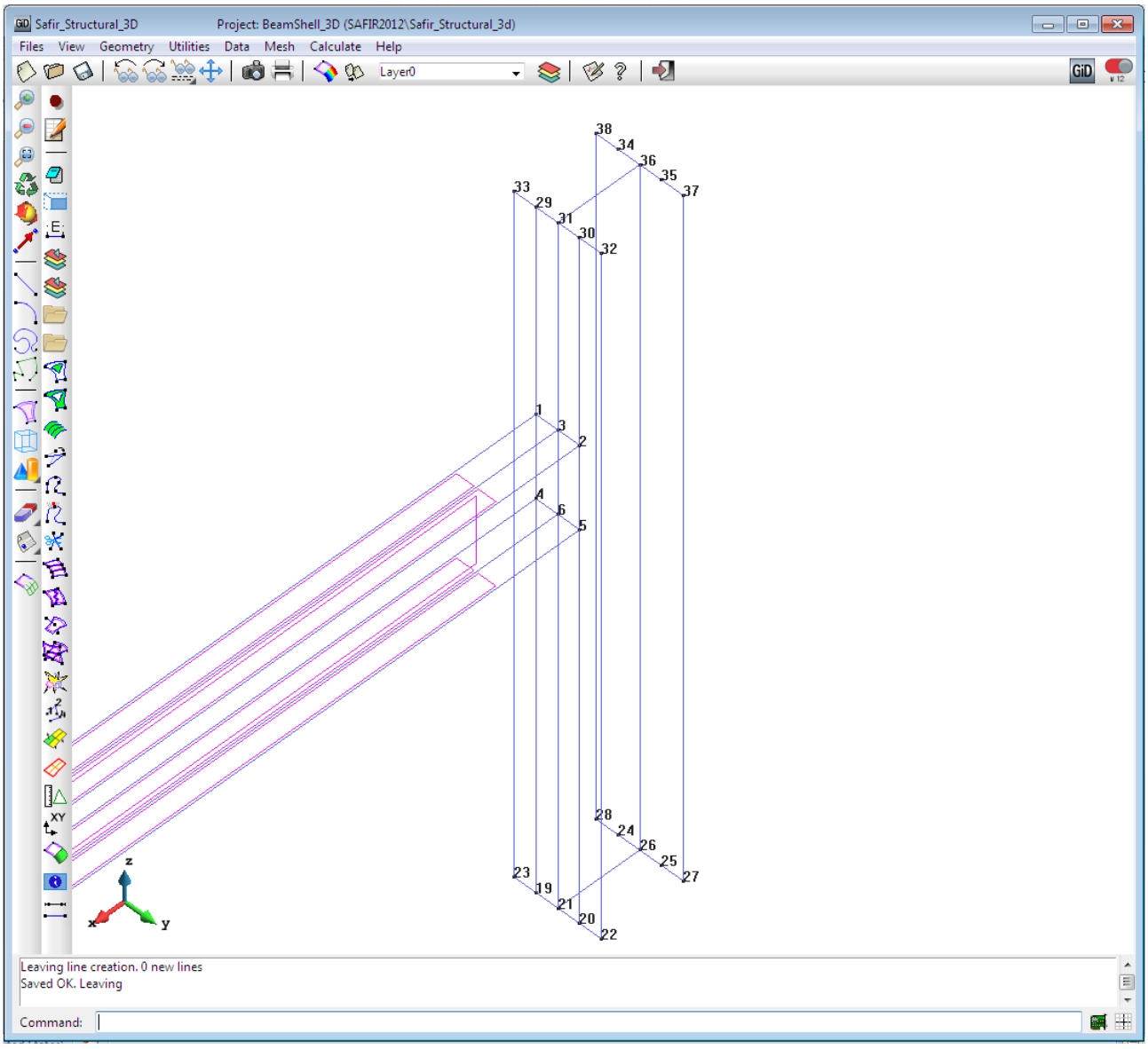


The top and bottom column ends are now created.

To connect all the points, select:

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

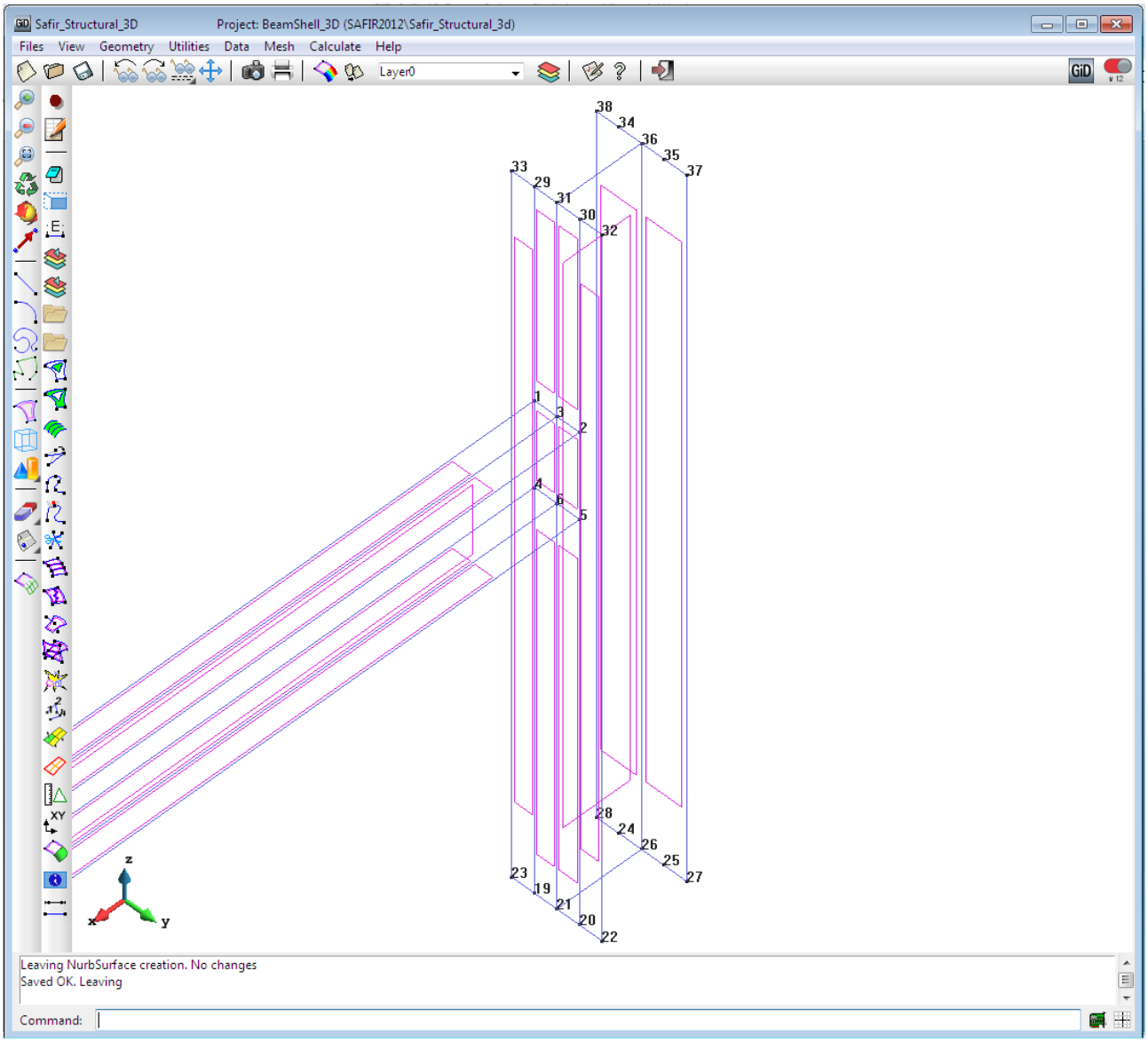
Connect all the points as shown in the figure below :



To create NURBS surfaces, select from the pull down menu:

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

Select the contour line and define the surfaces as shown in the following figure:



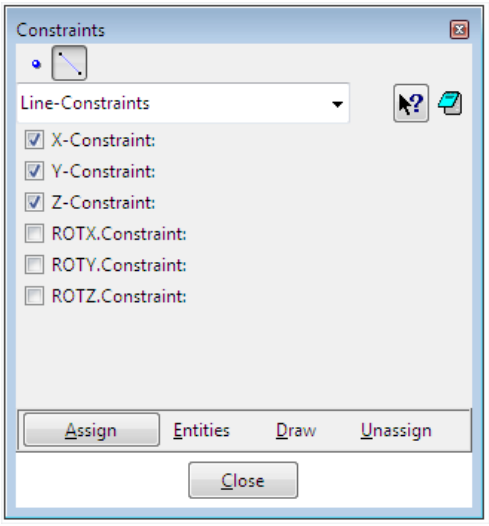
3. Define the constraints for the supports

From the pull down menu select

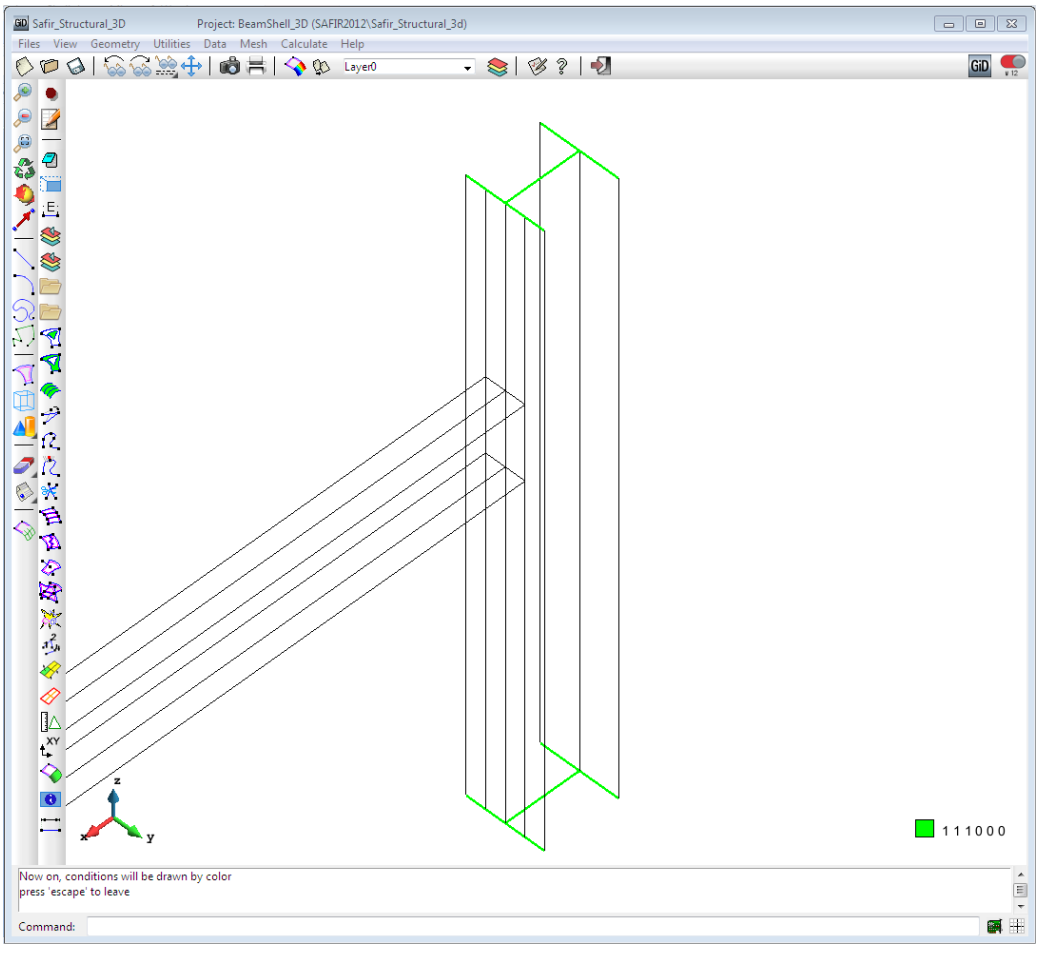
- ***Data->Constraints***

Select the *Line-Constraints* tab.

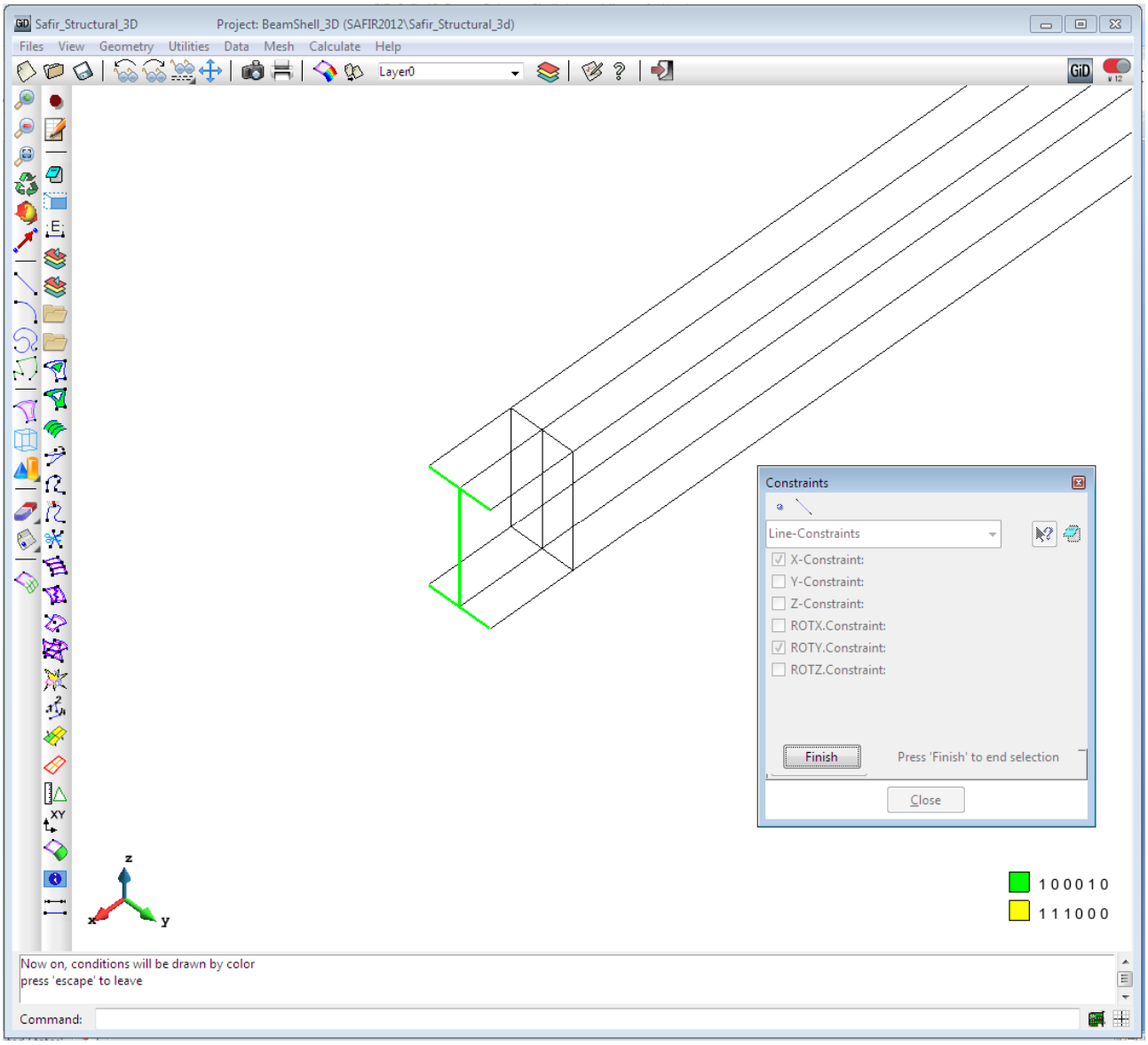
Select ***X, Y*** and ***Z-Constraint*** and assign them to the base and top lines of the column.



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



Do the same operation and assign a ***X Constraint*** and a ***ROTY-Constraint*** to the exterior end part of the beam as shown below :



Press ***Finish*** then ***Close*** to leave the *Constraints* dialog box window.

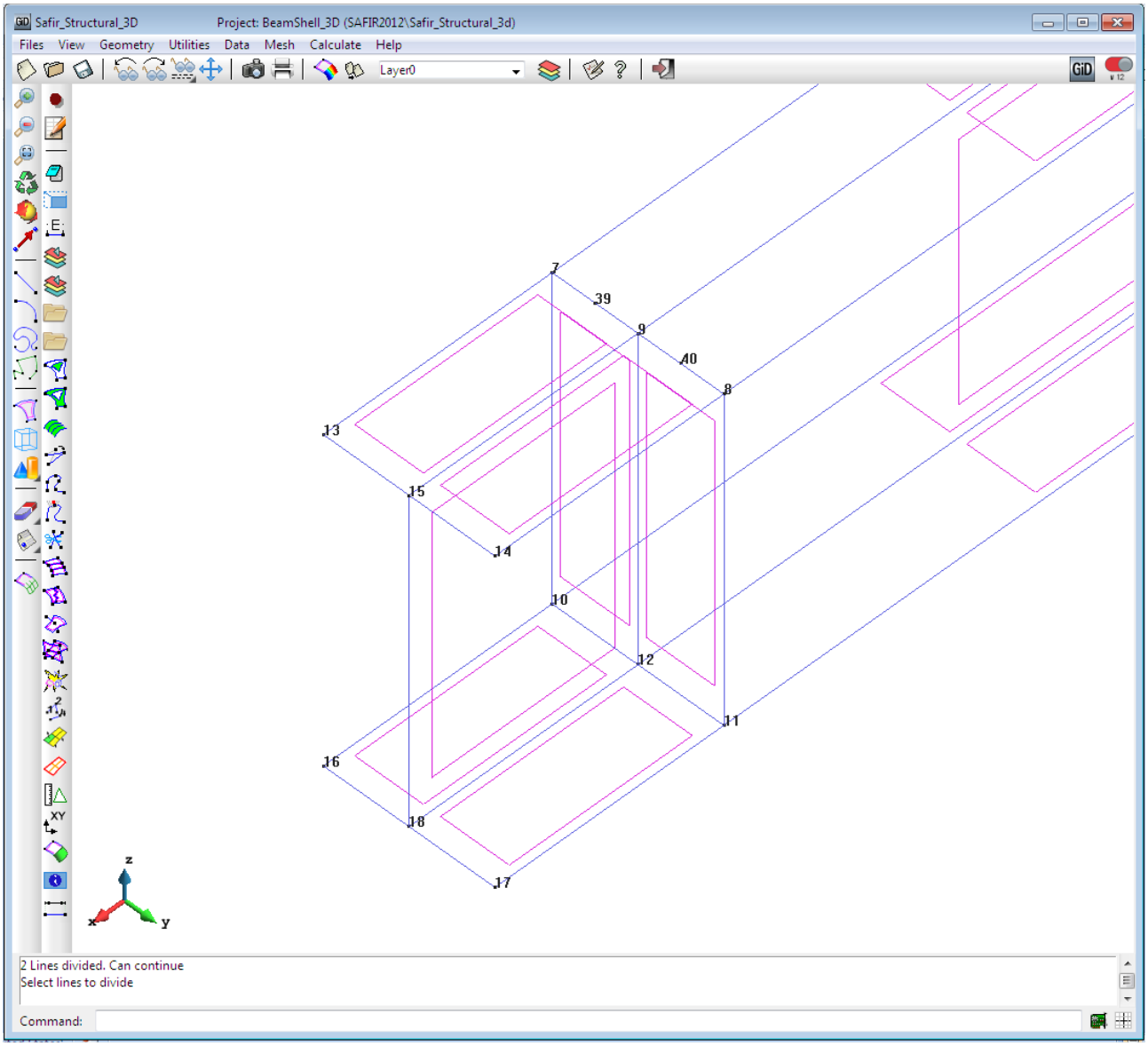
4. Define the loads

The loads will be applied at the end of the beam, at the level of the stiffener.

From the pull down menu select:

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

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

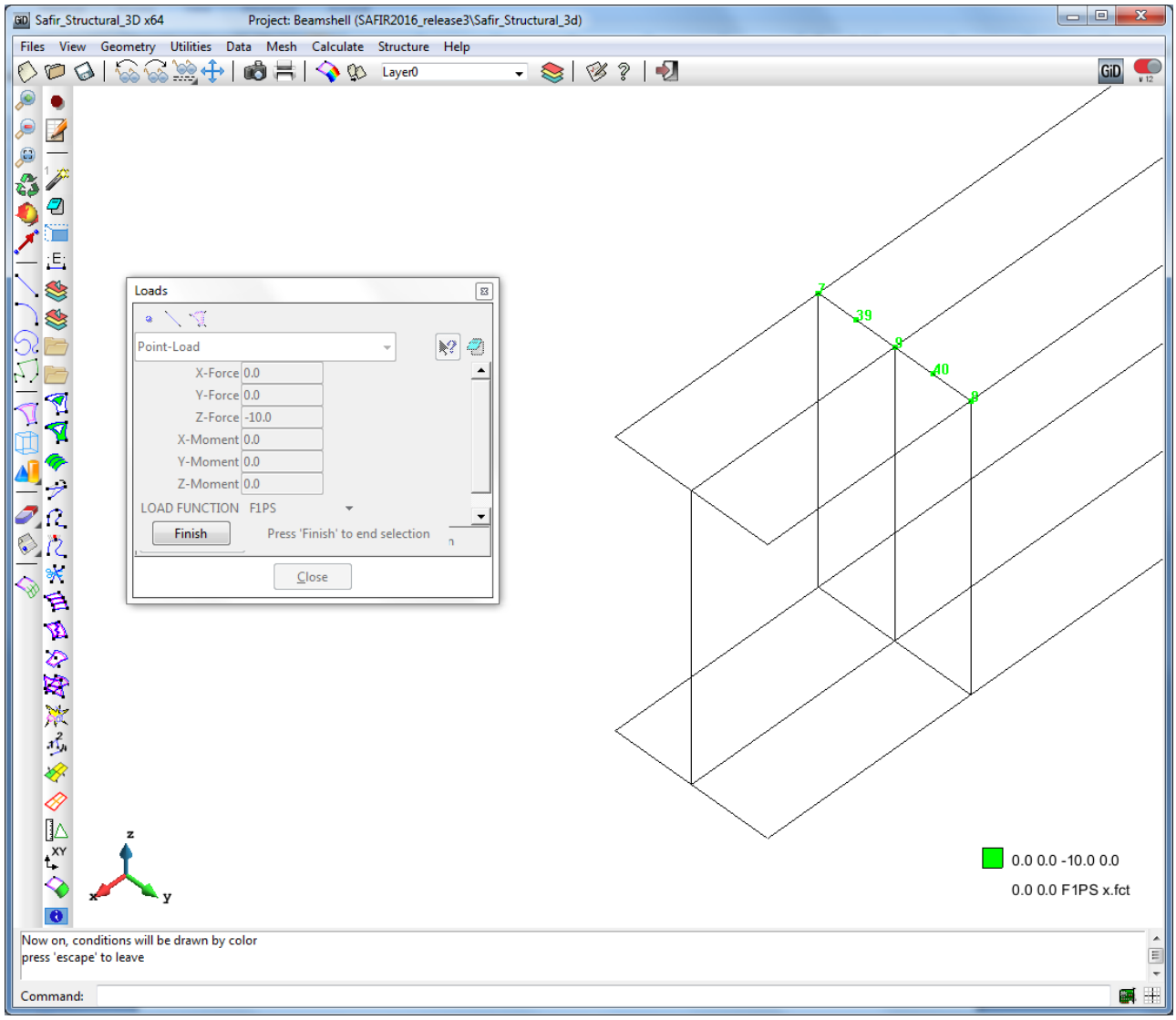


To define the loads, from the pull down menu select:

- ***Data->Loads***

Put a *Point-Load Z-Force = -10 N*

Select F1PS in the LOAD FUNCTION dialog box.



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

Press ***Finish***, then ***Close*** to leave this view mode.

5. Assign .TSH files

The objective is to assign the different *.tsh* files to the shell surface elements :

- flange of the column : *column_flange.tsh* (thickness = 0.019m)
- web of the column : *column_web.tsh* (thickness = 0.011m)
- flange of the beam : *beam_flange.tsh* (thickness = 0.0107m)
- web of the beam : *beam_web.tsh* (thickness = 0.0071m)
- stiffener : *stiffener.tsh* (thickness = 0.015m)

Select from the pull down menu:

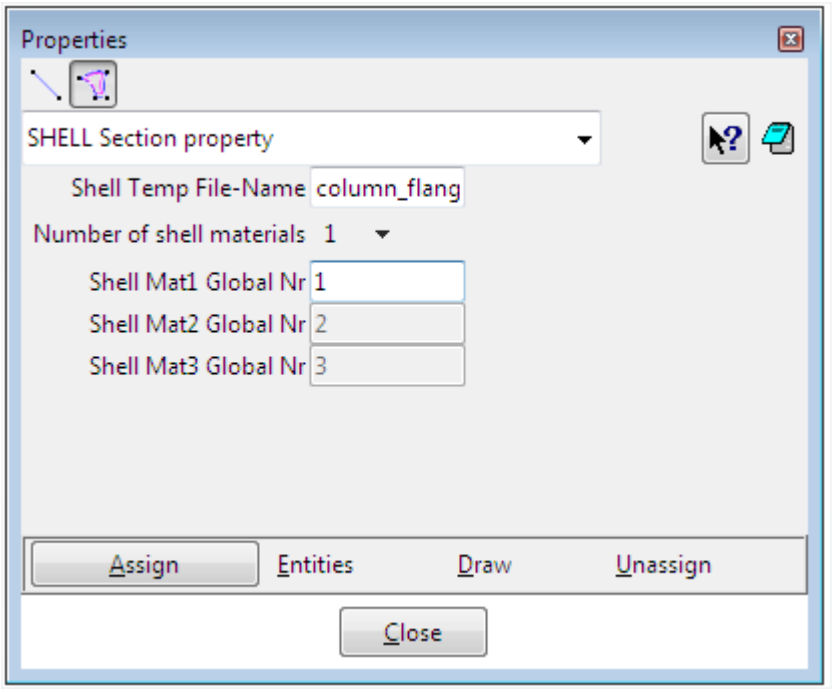
- ***Data-> Properties***

Select the *SHELL Section property* tab  and put:

As *Shell Temp File-Name*: ***column_flange.tsh***

As *Number of shell materials* : ***1***

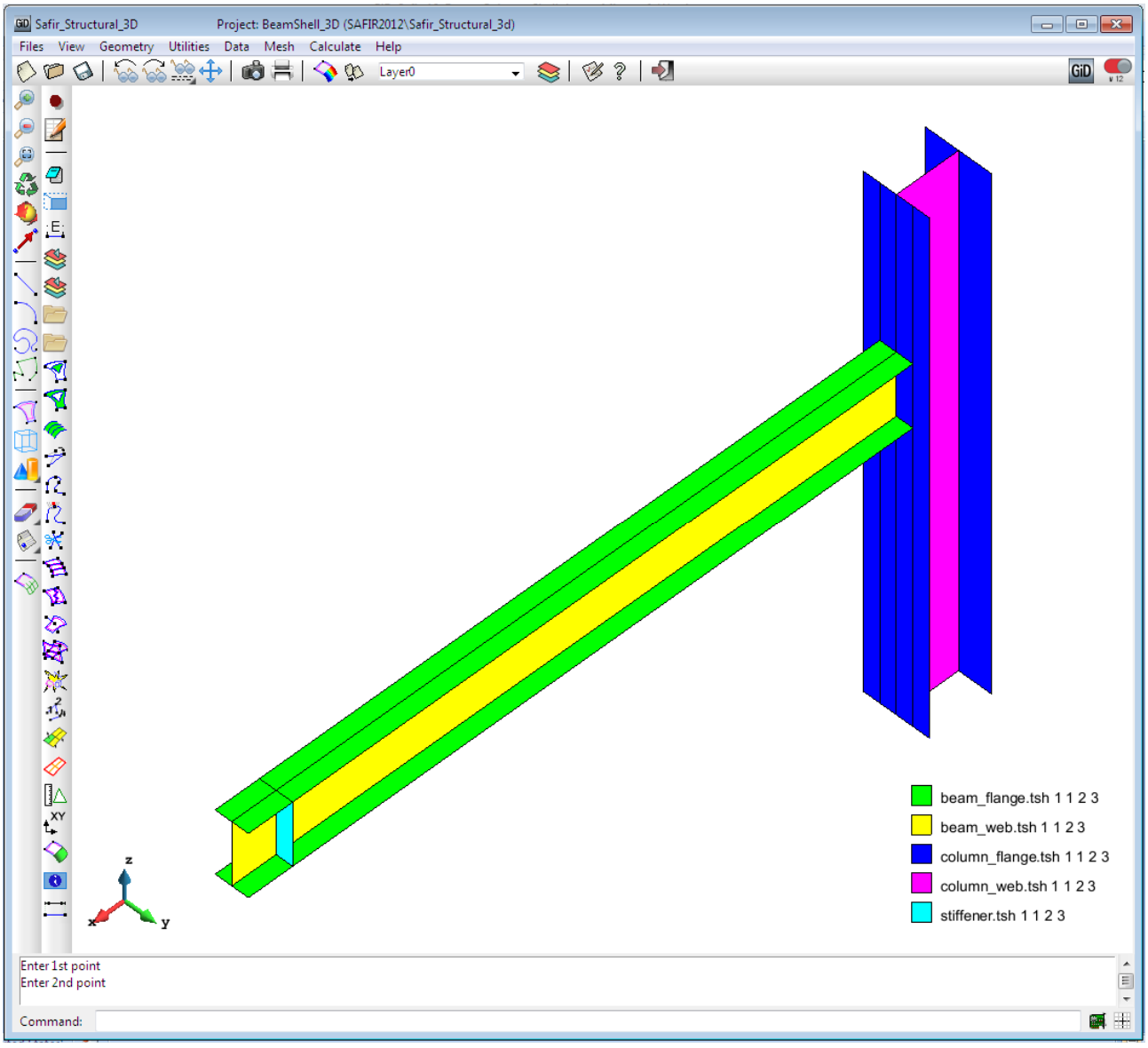
As *Shell Mat1 Global Nr*: ***1***



Click on ***Assign*** and select all column flanges corresponding surfaces and press ***Finish.***

Do the same operation for *column_web.tsh*, *beam_flange.tsh*, *beam_web.tsh* and *stiffener.tsh* .

Press ***Finish***, then ***Cancel*** to close the window.



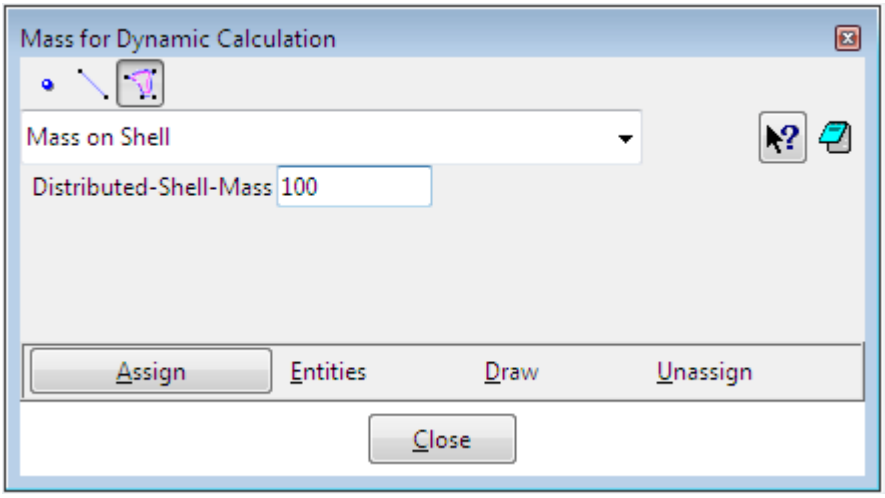
Refer to Exercise n°4 to create the different *.tsh* files.

6. Define the Mass

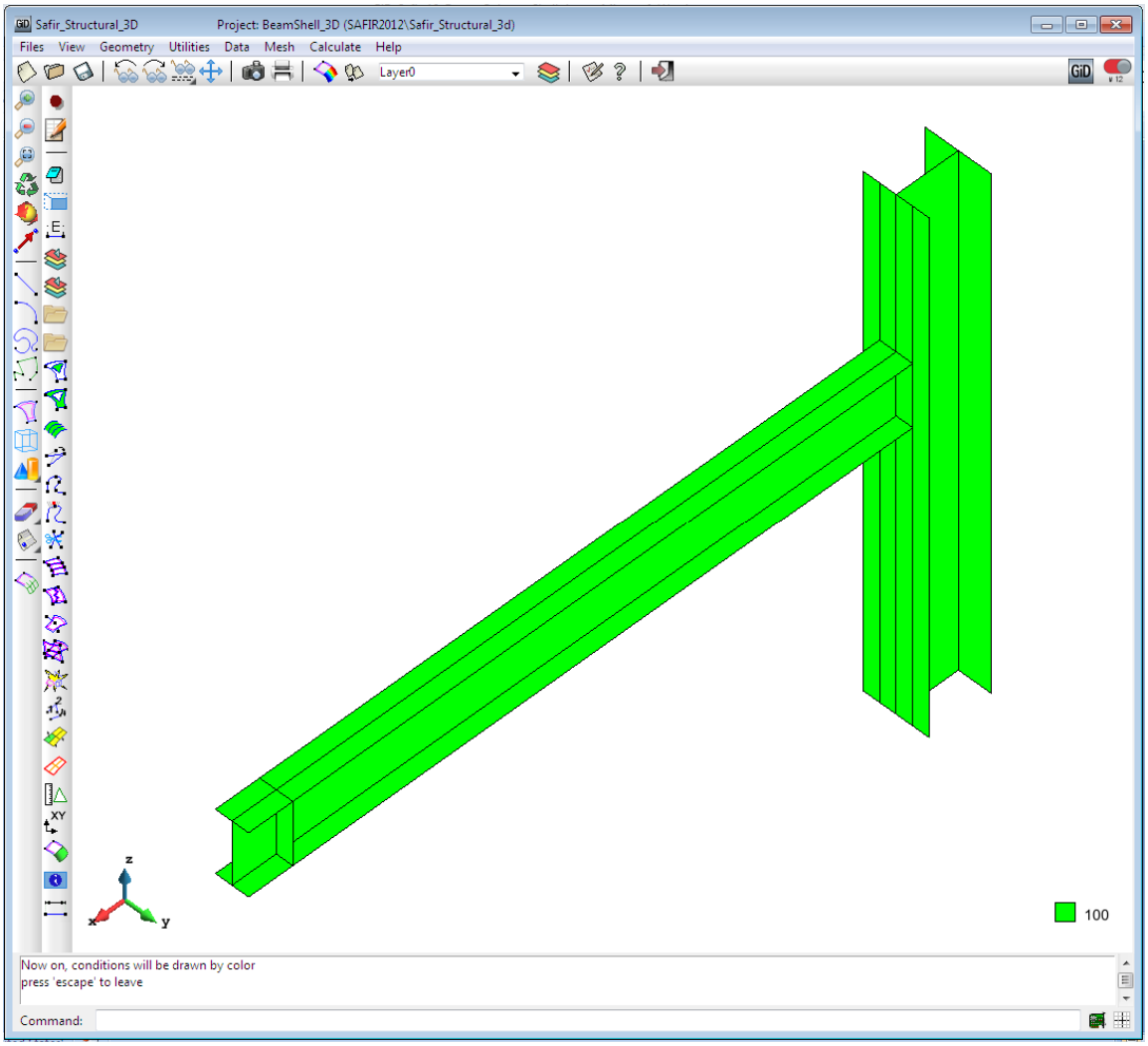
Select from the pull down menu:

- ***Data-> Mass***

Select the *Mass on Shell* tab  and fill as below.



Click on ***Assign*** and select all surfaces of the structure and press ***Finish***, then ***Close.***

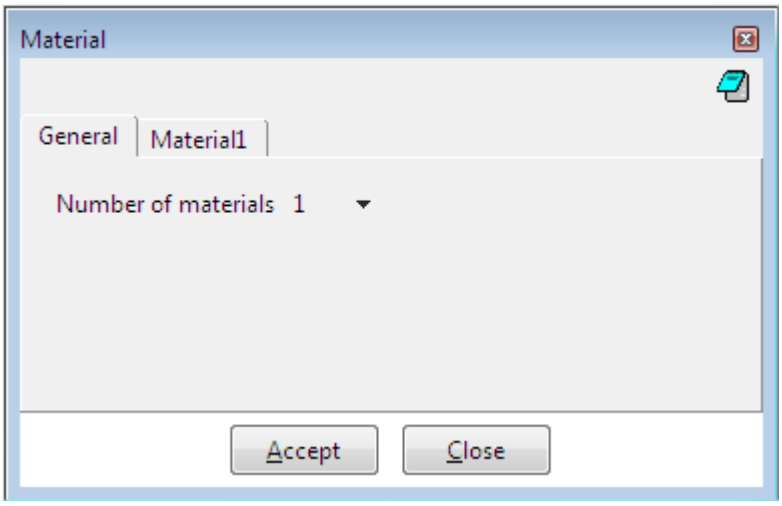


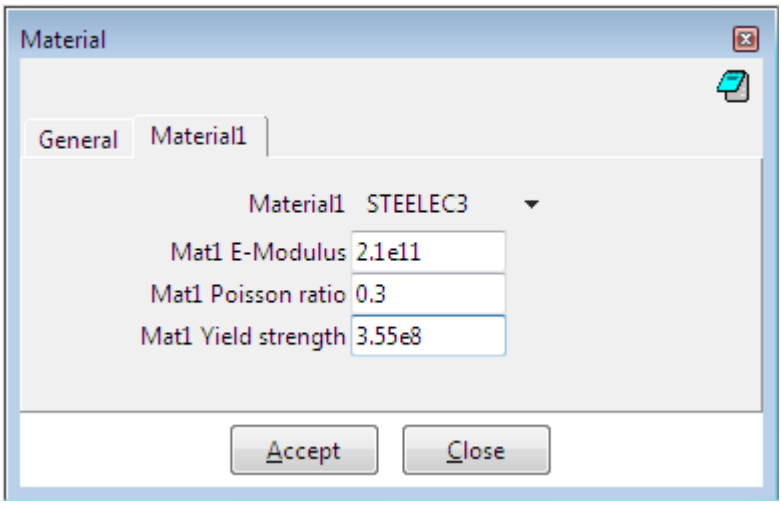
7. Define the materials

To define the material of the shell elements, select from the pull down menu:

- ***Data->Material***

In the *General* tab, put ***1*** as *Number of materials*; in the *Material1* tab, fill as shown below.



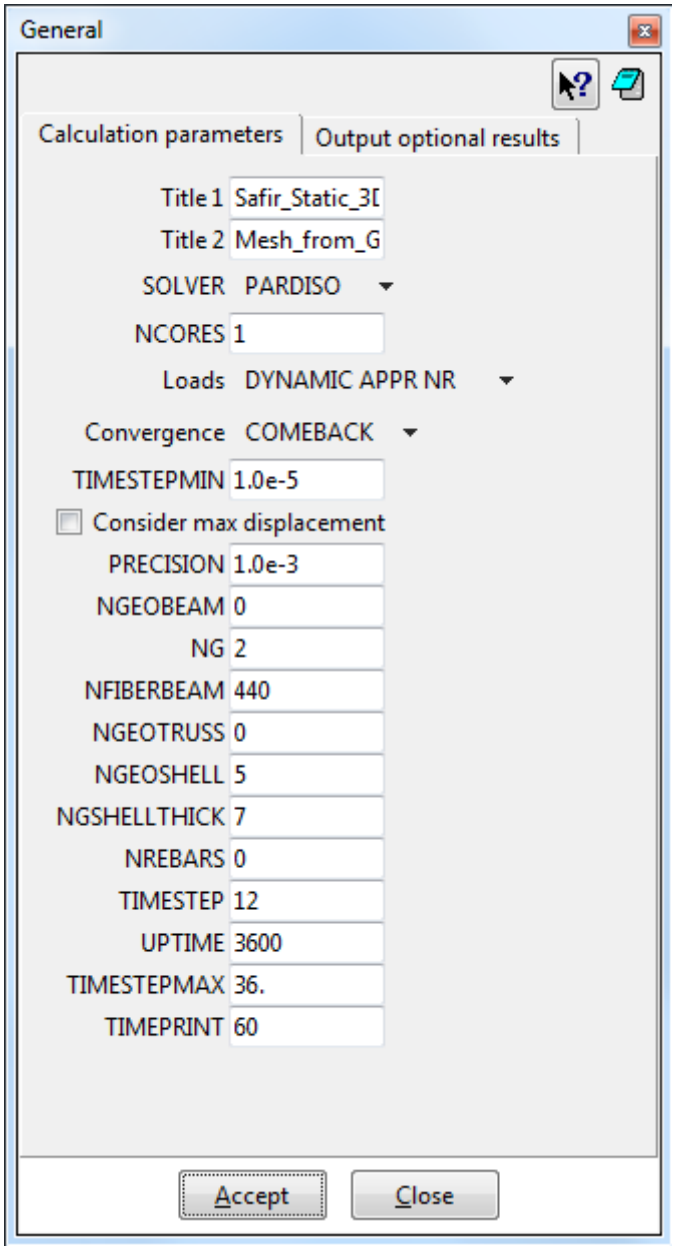


8. Define the general data

Select from the pull down menu:

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

And fill as shown below :



- ***NGEOSHELL= 5*** (= number of *.tsh* files)
- ***NGSHELLTHICK = 7*** (Number of divisions/integration points)
- ***TIMSTEP, UPTIME, TIMEPRINT,…*** as needed

Click on ***Accept*** to save your modifications.

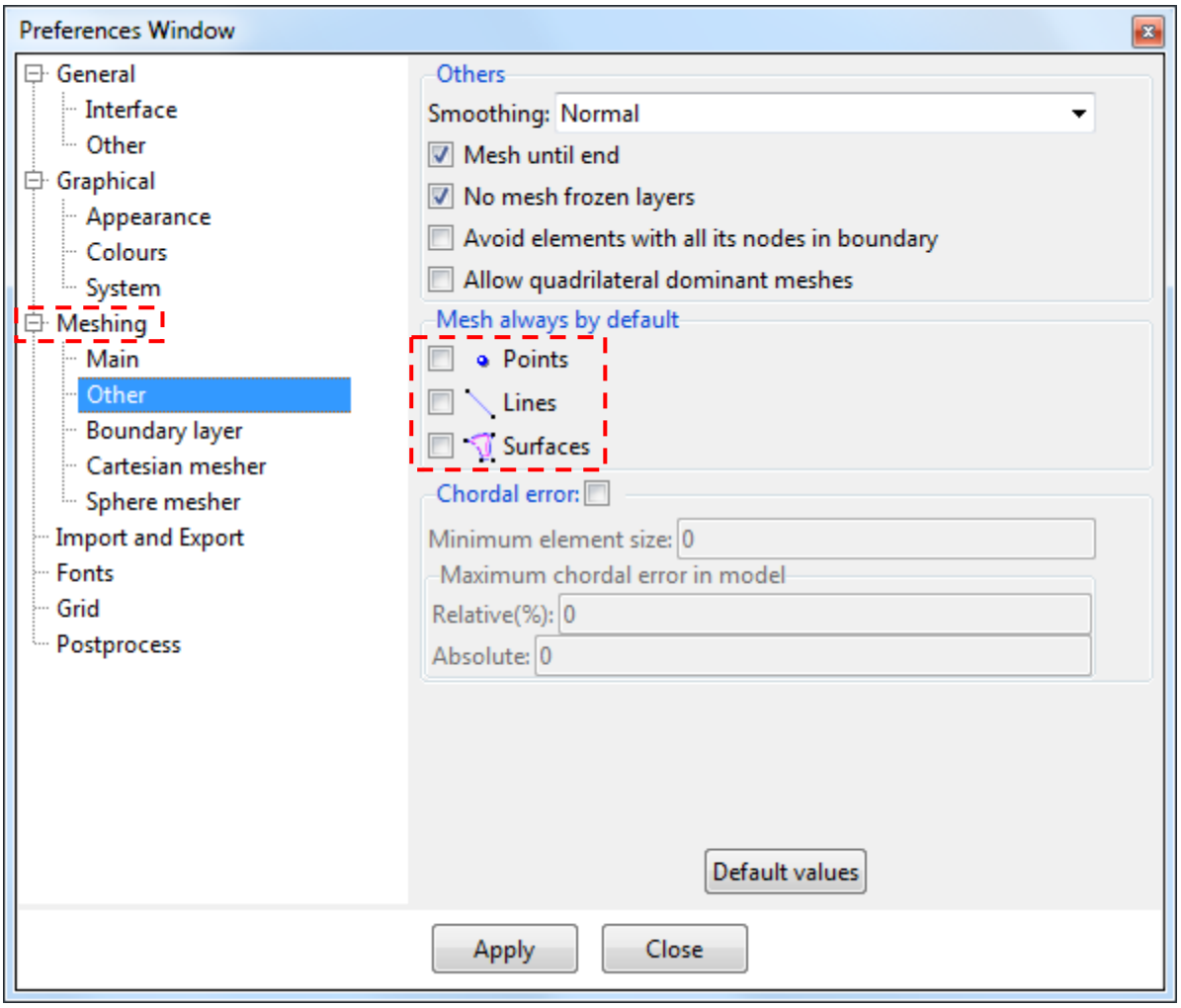
9. Generate the mesh

Before meshing, you need to specify the default meshing criteria.

Select from the pull down menu

- ***Utilities->Preferences***

Then go to *Meshing / Other / Mesh always by default* and check that all options are disabled as shown in the following figure.



Then select from the pull down menu:

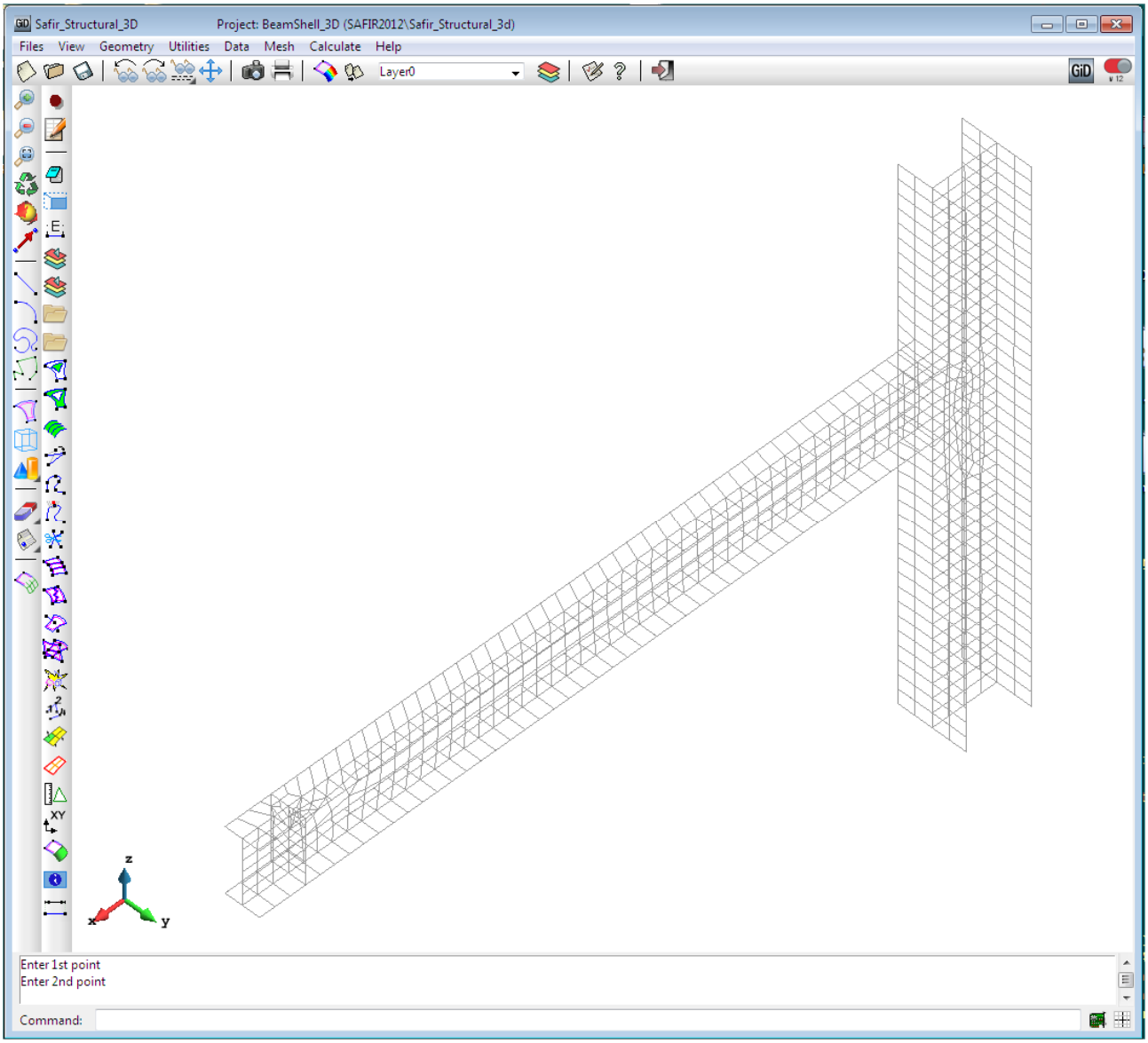
- ***Mesh->Element Type->Quadrilateral***

Select the whole structure.

Then select from the pull down menu:

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

Enter the element size of 0.064

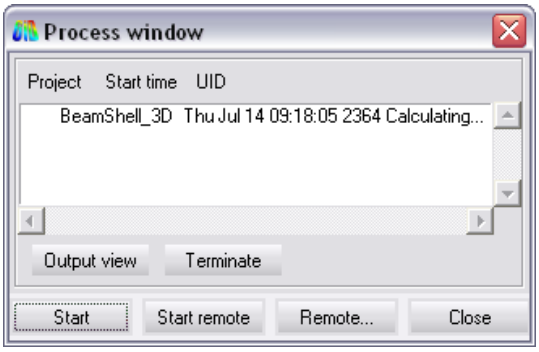


10. Start the calculation:

Before starting the calculation, don't forget to copy all the .tsh files (column_flange, column_web, beam_flange, beam_web and stiffener) into the BeamShell.gid diretory.

Then select from the pull down menu:

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



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