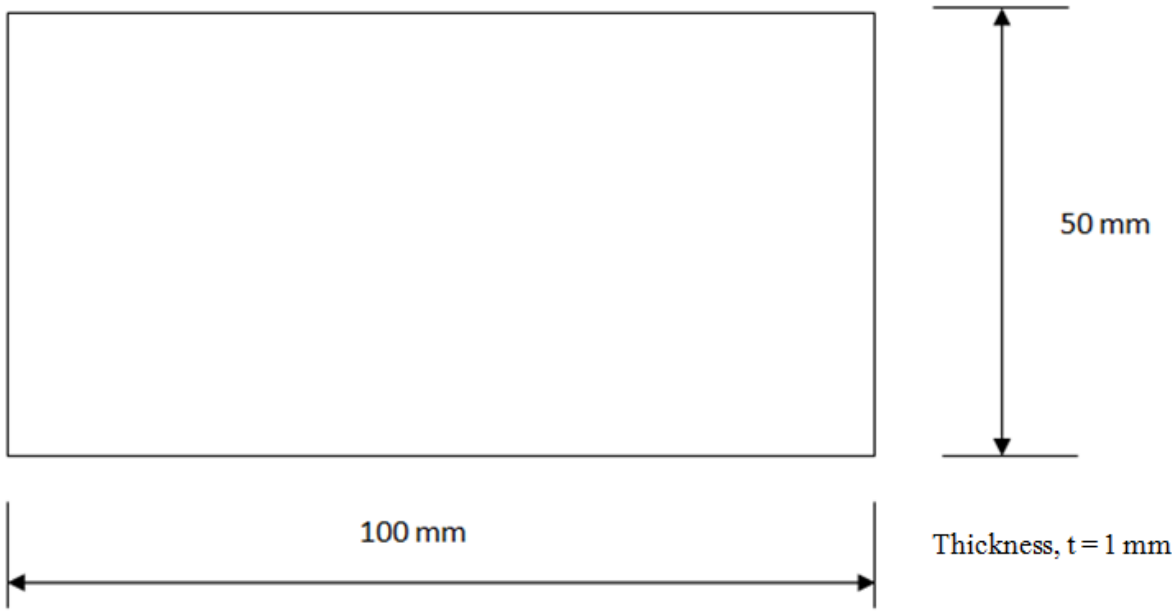
FREE VIBRATION ANALYSIS OF A COMPOSITE RECTANGULAR CANTILEVER PLATE



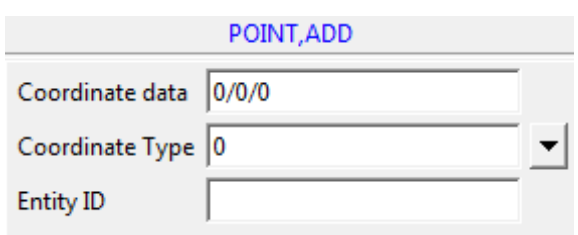
PROCEDURE

1. **Create keypoints**

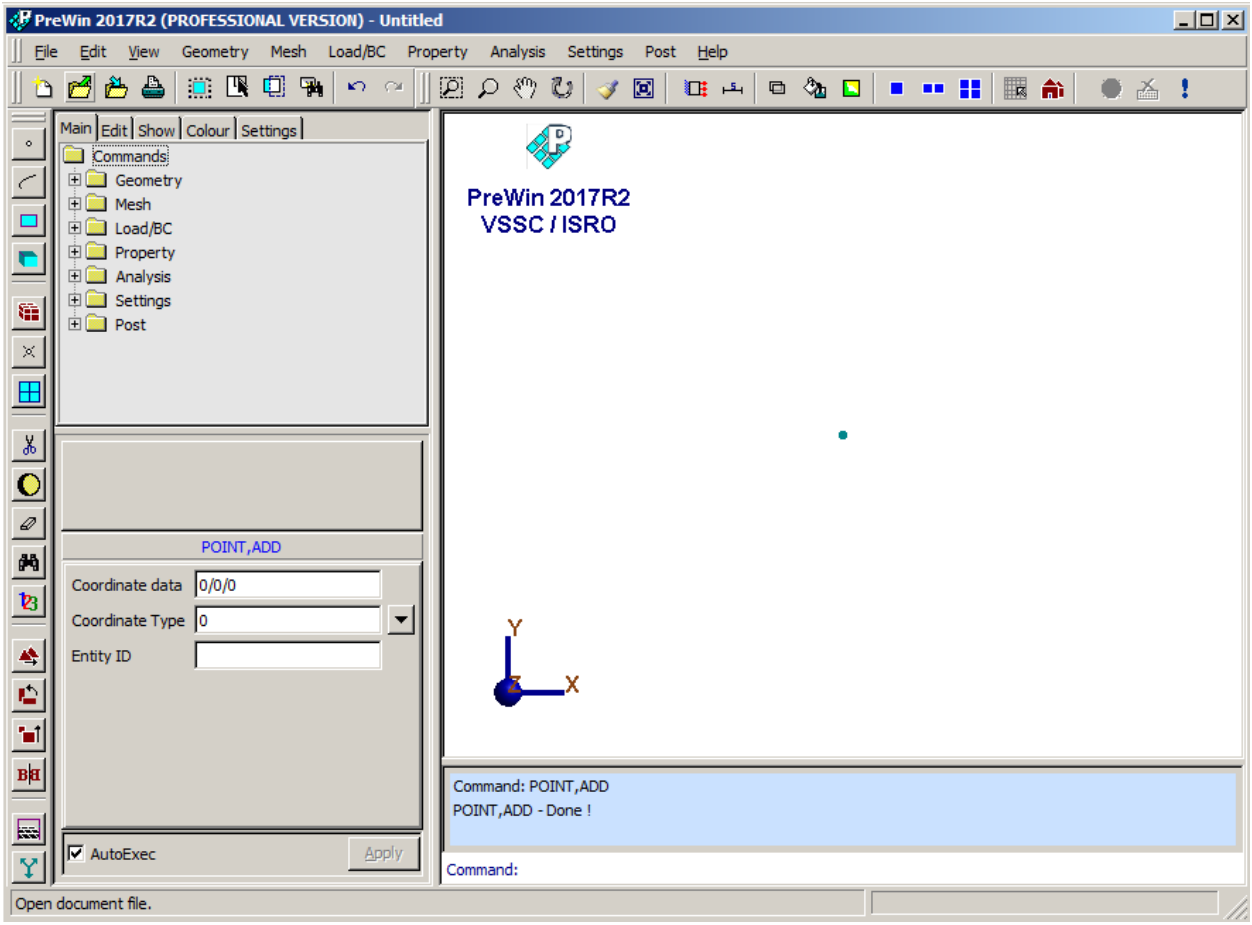
Command : POINT,ADD

Menu : Geometry → Keypoint → Create → By X/Y/Z

Parameters :



At the end of the operation your screen should look like this.



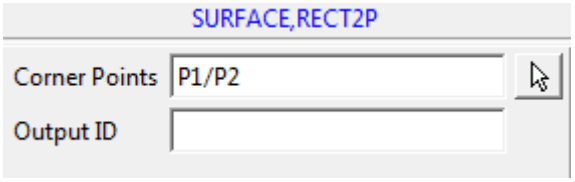
Similarly create key points at (100/50/0)

2. Create surface

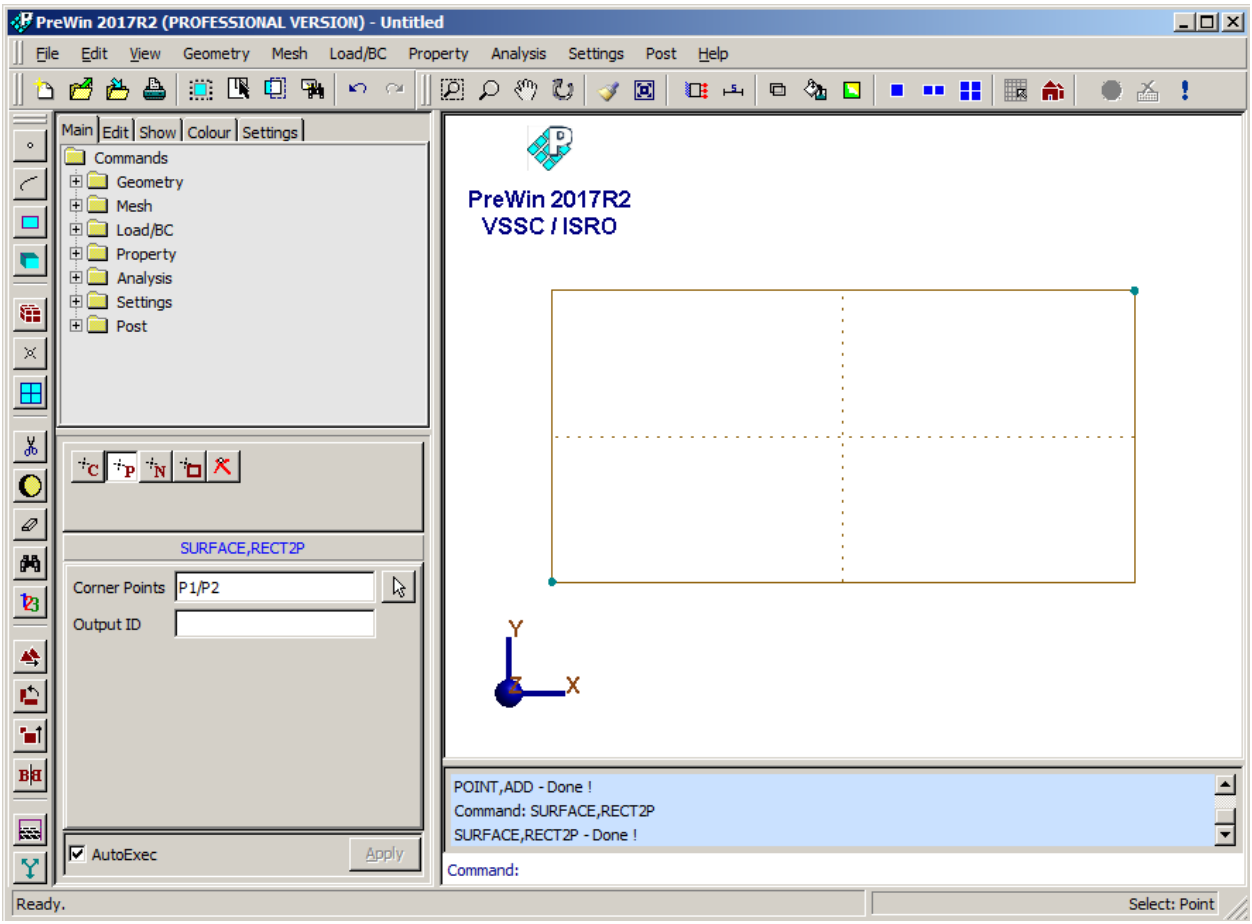
Command : SURFACE,RECT2P

Menu : Geometry → Surface → Create → Rect2P

Parameters :



At the end of the operation your screen should look like this.

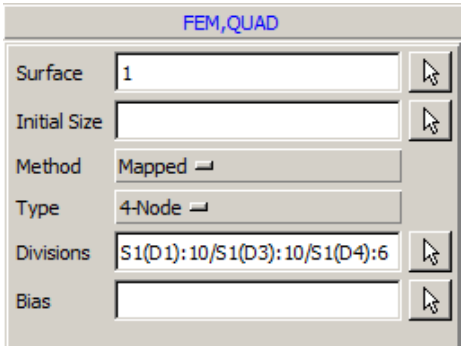


3. **Generate mesh**

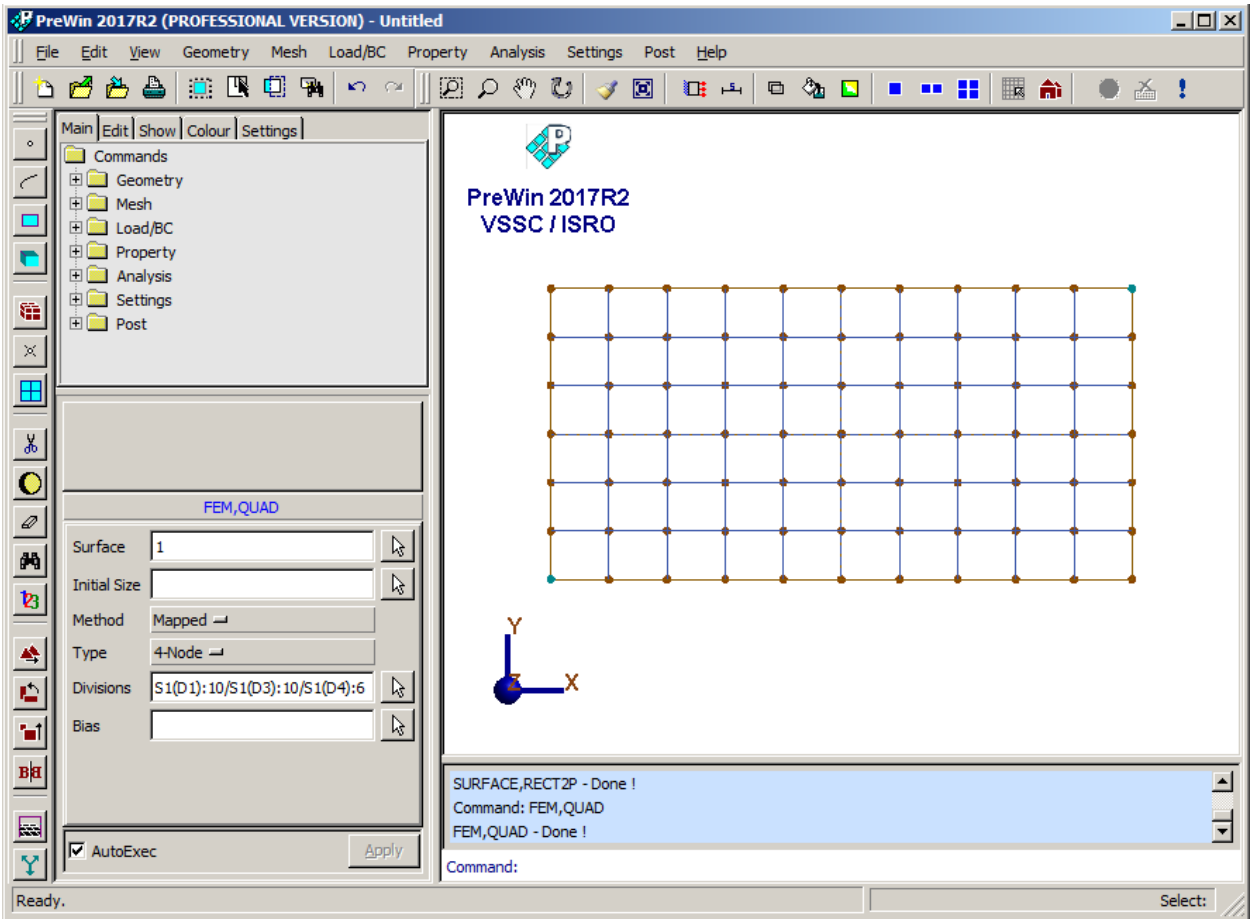
Command : FEM,QUAD

Menu : Mesh → FE Mesh → QUAD

Parameters :



At the end of the operation your screen should look like this.

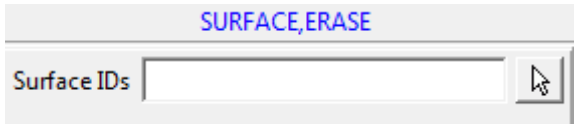


4. Erase surface

Command : SURFACE,ERASE

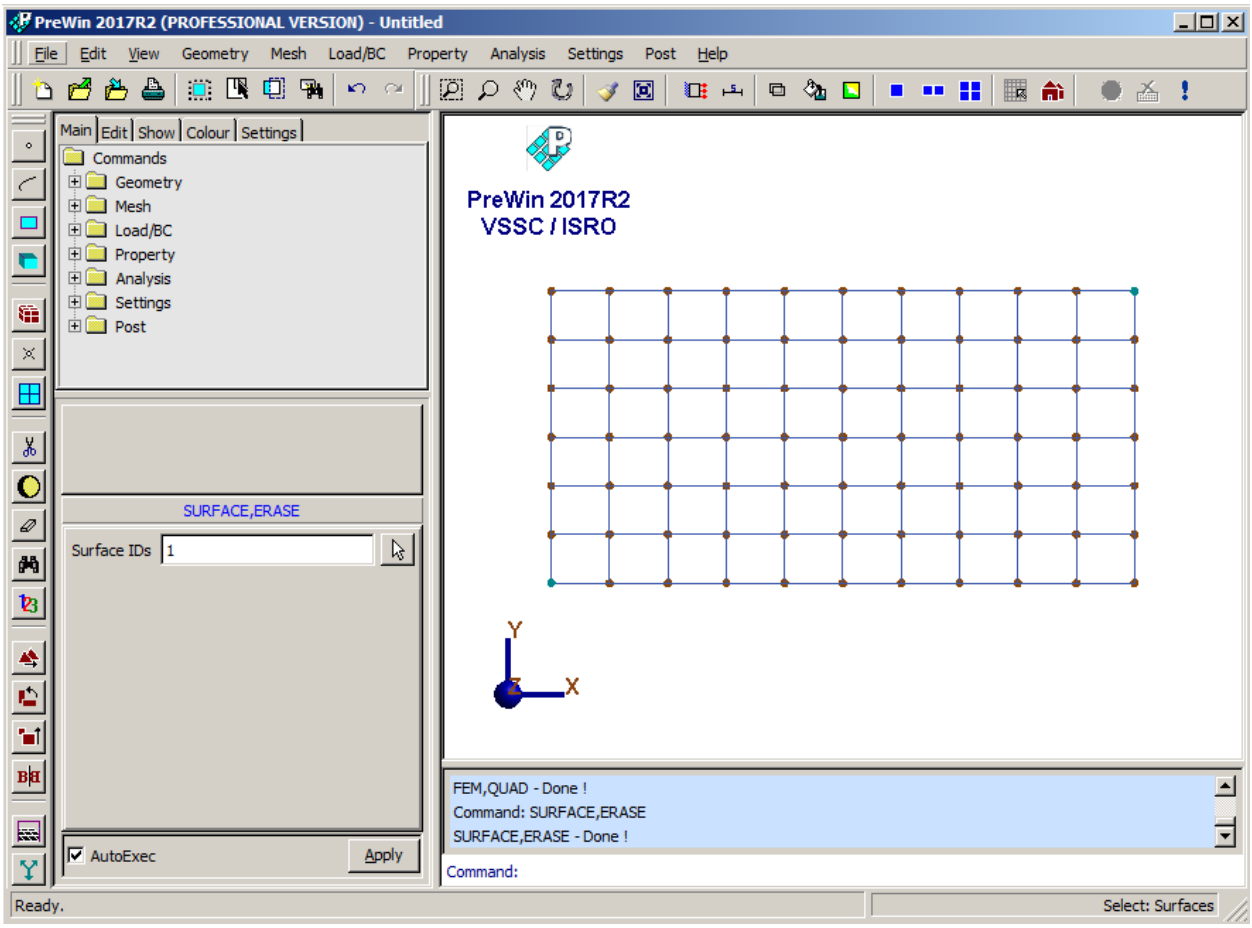
Menu : Geometry → Surface → Miscellaneous → Erase

Parameters :



Type in the surface ID or pick the surface after clicking the arrow in the surface ID box

At the end of the operation your screen should look like this.

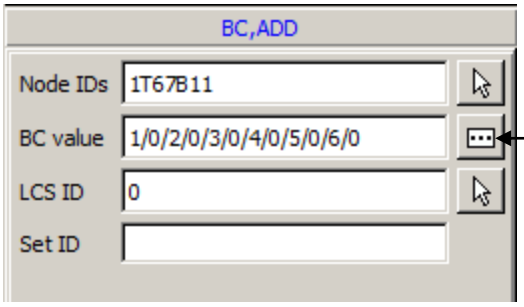


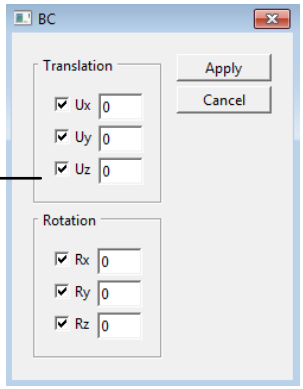
5. Apply boundary condition

Command : BC,ADD

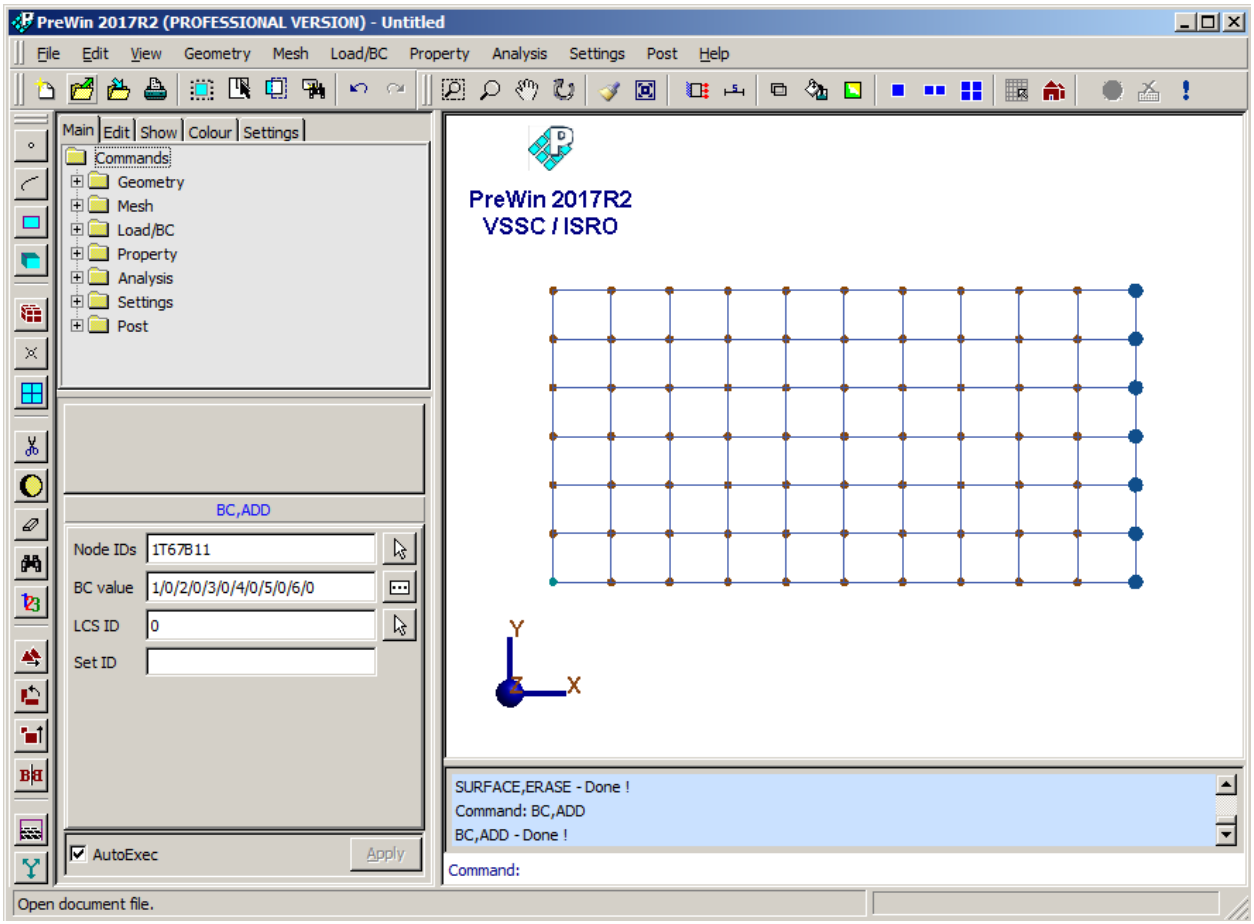
Menu : Load/BC → Displacement BC → Add

Parameters :





At the end of the operation your screen should look like this.

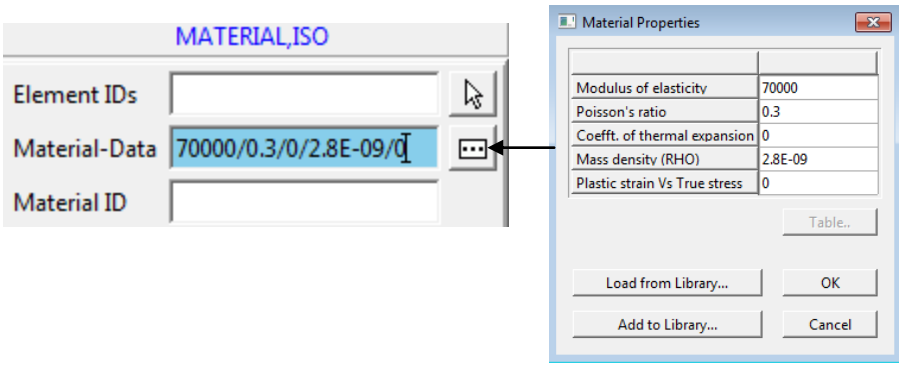


6. Create material property for different layers

Command : MATERIAL,ISO

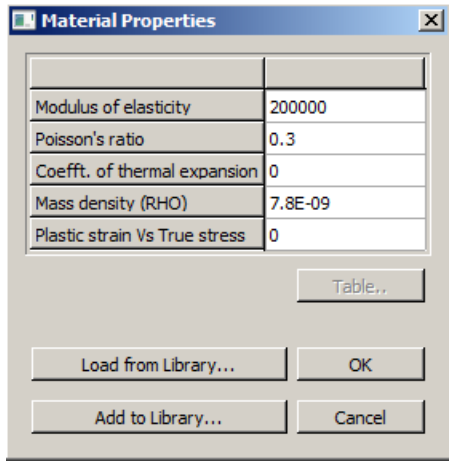
Menu : Property → Material → Isotropic → Add

Parameters :



**Do not specify any Element IDs here.

Similarly create another set of material property with following data.



Verify whether two set of material properties were defined in 'Edit' option.

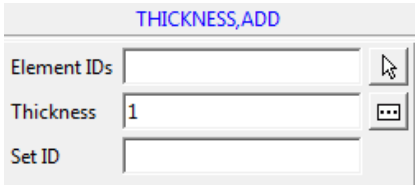
Material property can be load from library or can be typed in.

7. Create thickness for layers

Command : THICKNESS,ADD

Menu : Property → Physical → Thickness → Add

Parameters :



Here also do not specify the element ID's.

Create another set of thickness data with thickness value 2mm.

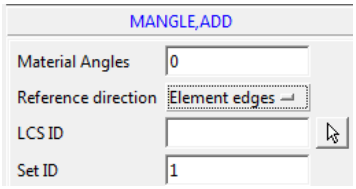
Verify whether two set of thickness values were defined in 'Edit' option .

8. Create material angle

Command : MANGLE,ADD

Menu : Property → Physical → Material angle → Add

Parameters :

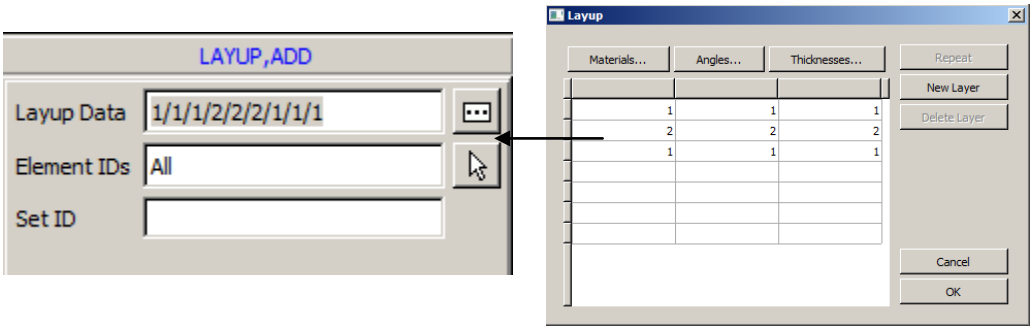


Create another set of material angle with value 90.

Verify whether two set of material angle values were defined in 'Edit' option.

9. Create layup details

Command : LAYUP,ADD
Menu : Property → Physical → Layup Details → Add
Parameters :



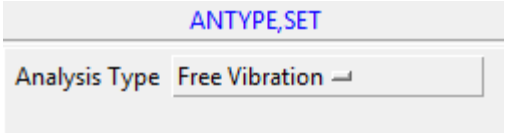
Now the composite layer definition is done.

10. Set Analysis Type

Command : ANTYPE,SET

Menu : Analysis → Analysis Type

Parameters :

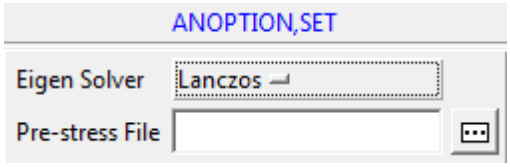


11. Set Analysis Option

Command : ANOPTION,SET

Menu : Analysis → Analysis Option

Parameters :

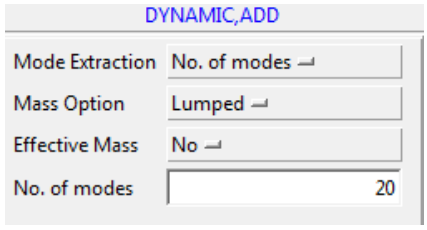


12. Set free vibration general data

Command : DYNAMIC,ADD

Menu : Analysis → Free Vibration → General → Add

Parameters :

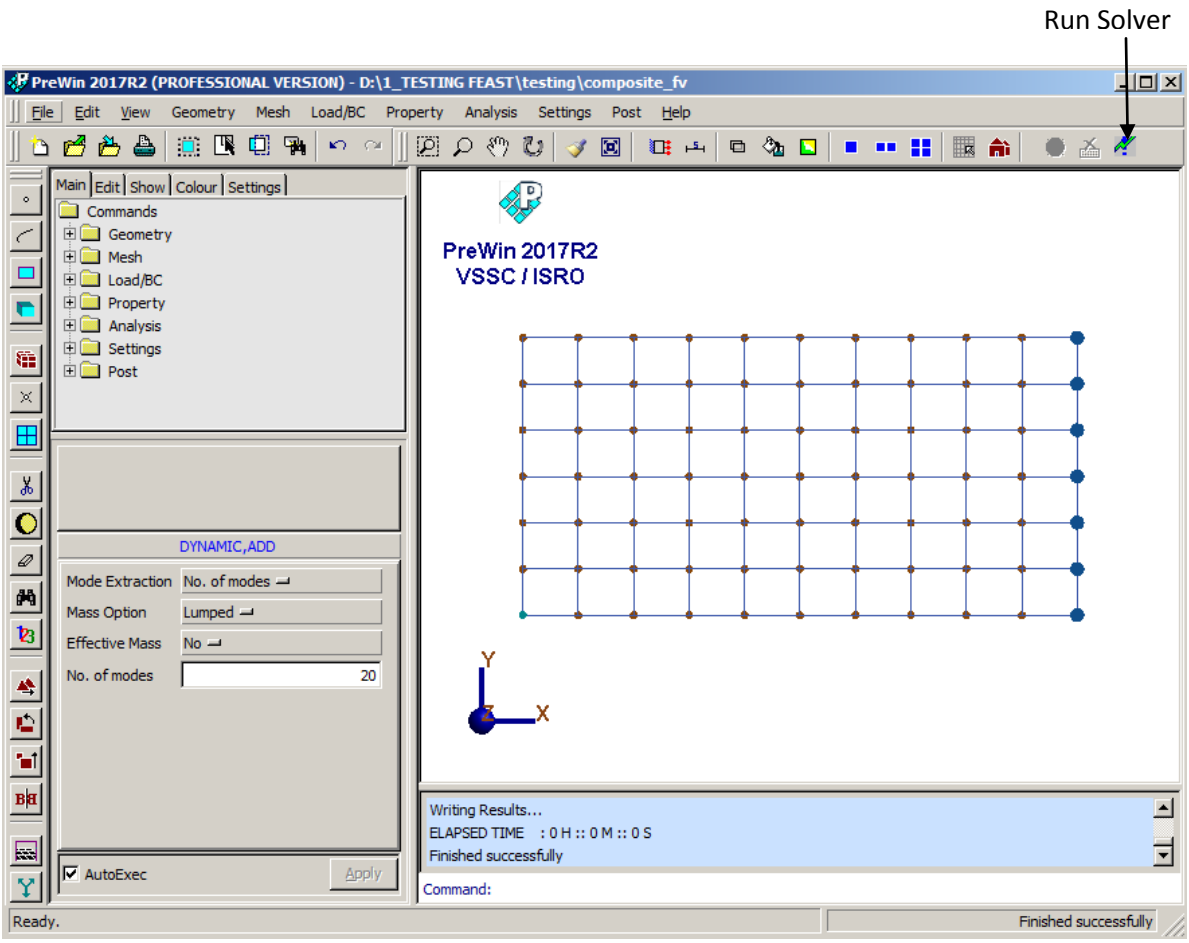


13. Save the project model

Menu : File → Save

14. Submit the job in to FEAST

Menu**: Analysis** → Run Solver



15. Perform Post Processing

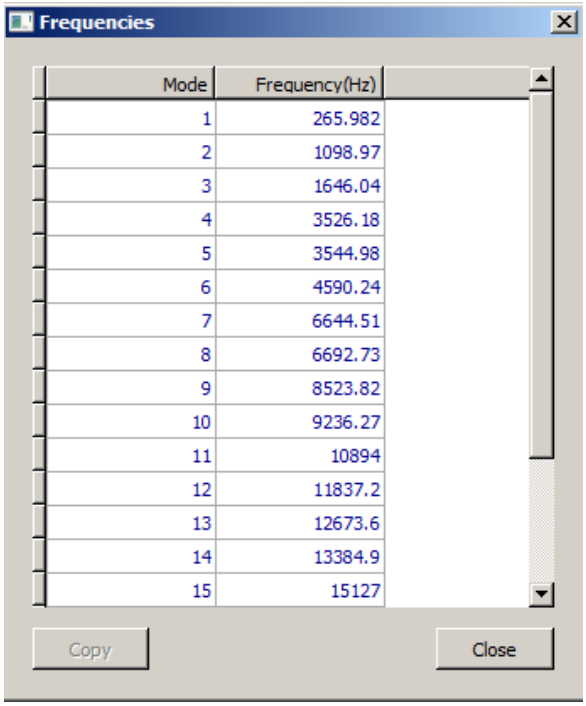
i) View result. Frequency

Command : POST,VIEWRESULTS

Menu : POST → View Result

Parameters :

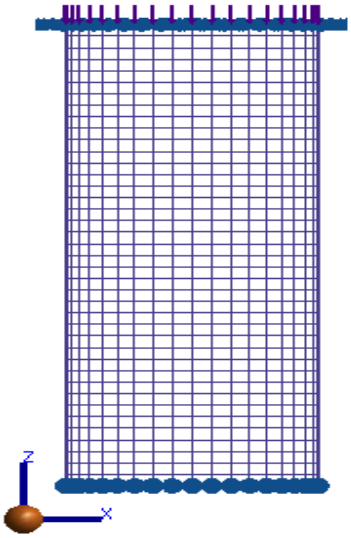
At the end of the operation your screen should look like this.



Mode	Frequency(Hz)
1	265.982
2	1098.97
3	1646.04
4	3526.18
5	3544.98
6	4590.24
7	6644.51
8	6692.73
9	8523.82
10	9236.27
11	10894
12	11837.2
13	12673.6
14	13384.9
15	15127

Output can be seen in *.OUT file

BUCKLING ANALYSIS OF CYLINDRICAL SHELL



Geometric Properties

Radius=2*m*
Height=5*m*
Thickness=0.005m

Boundary condition

Fixed at bottom surface
Ux=Uy=0 at top surface

Ma*terial properties*

E=7e+10 N/m^2
$\nu=0.3$

Loading condition

Force=-50KN/m on top edge of the cylinder

Element Type

QUAD4 shell element

PROCEDURE

STEP

1. Create a Cylindrical surface

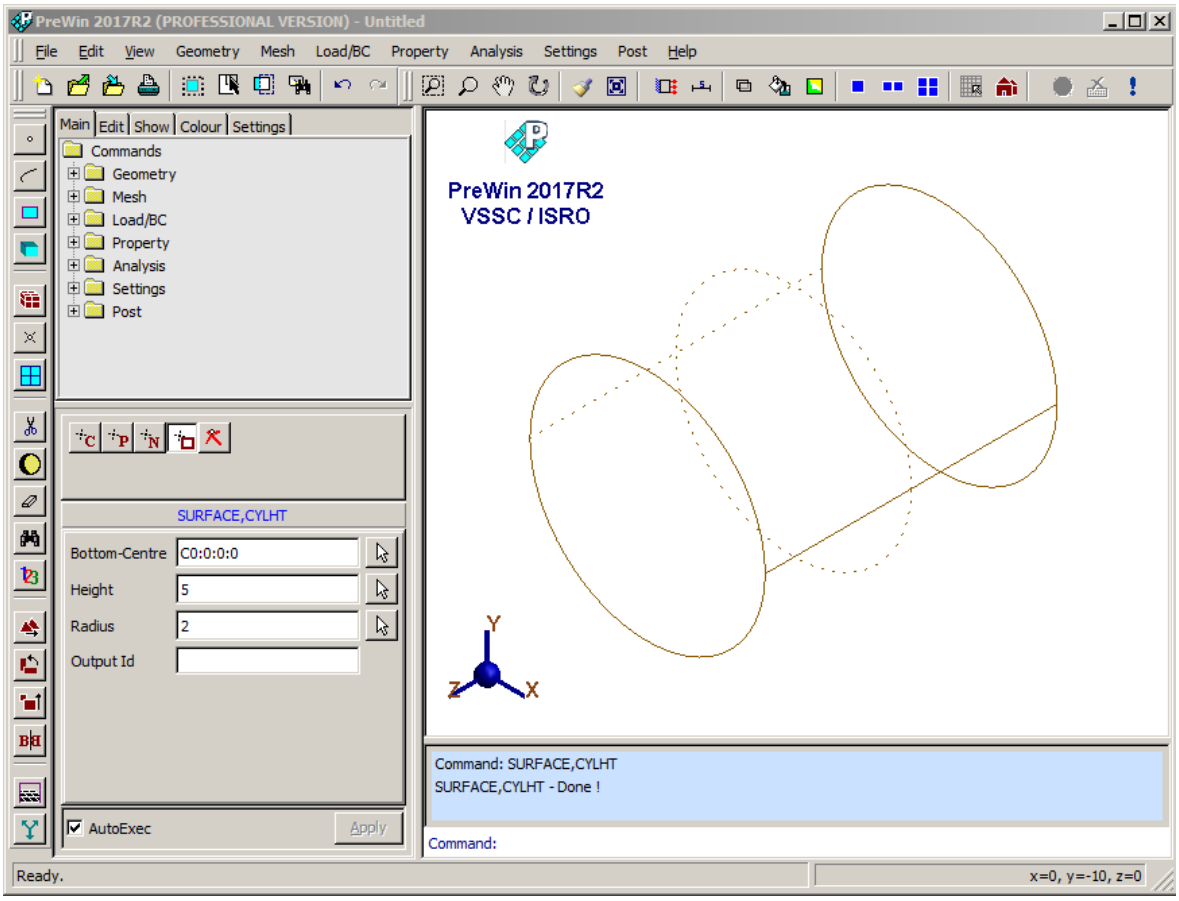
Command : SURFACE, CYLHT

Menu : Geometry → Surface →Create→CylinderH

Parameters

Centre	C0:0:0:0
Height	5
Radius	2

At the end of the above operations, your screen should look like this.



*Now change to bottom view

2. Meshing the surface using quadilateral elements

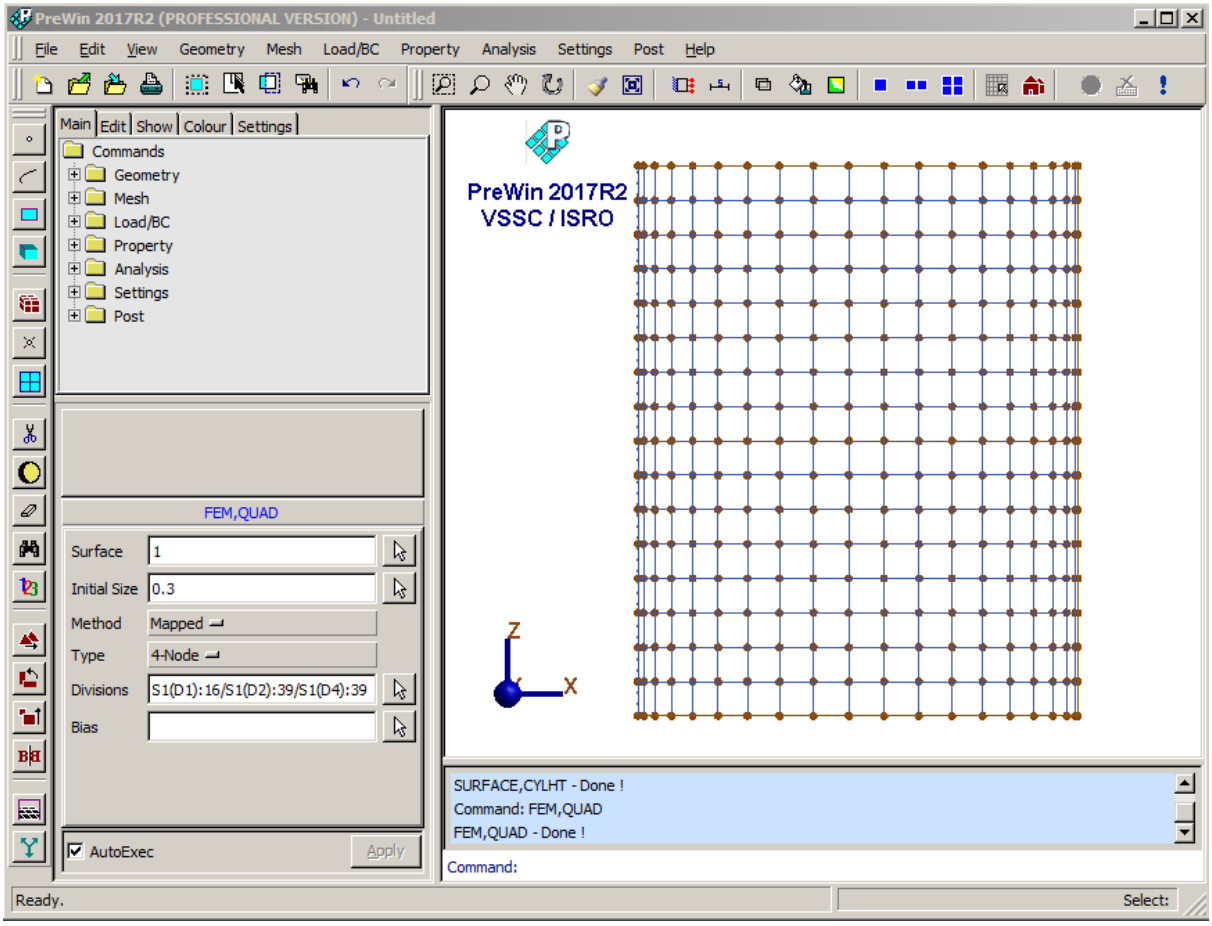
Command : FEM, QUAD

Menu : Mesh → FE Mesh → QUAD

Parameters :

Surface IDS	1
Initial Size	Give an approximate initial value by typing or picking two edge points
Method	Mapped
Type	4 Node
Divisons	S1(D1):16/S1(D2):39/S1(D4):39

At the end of the above operations, your screen should look like this.



3. Delete surfaces

Command : SURFACE, DELETE

Menu : Geometry → Surface → Miscellaneous → Delete

Parameters :

Surface IDS	1

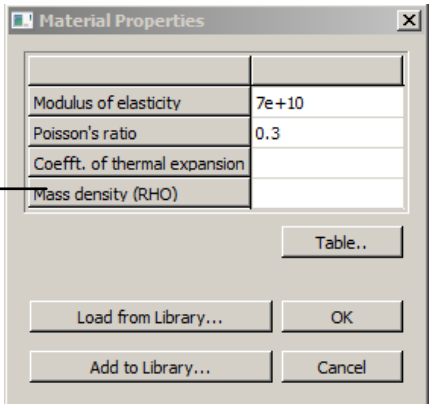
4. Specify material properties

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic → Add

Parameters :

Element IDs	All
Material Data	7e+10/0.3/0



5. Specify thickness

Command : THICKNESS, ADD

Menu : Property → Physical → Thickness → Add

Parameters :

Element IDs	All
Thickness	0.005
Set ID	1

6. Specify Displacement Boundary Conditions

a. Specify boundary conditions at bottom edges

Command : BC, ADD

Menu : Load/BC→ Displacement BC →Add

Parameters :

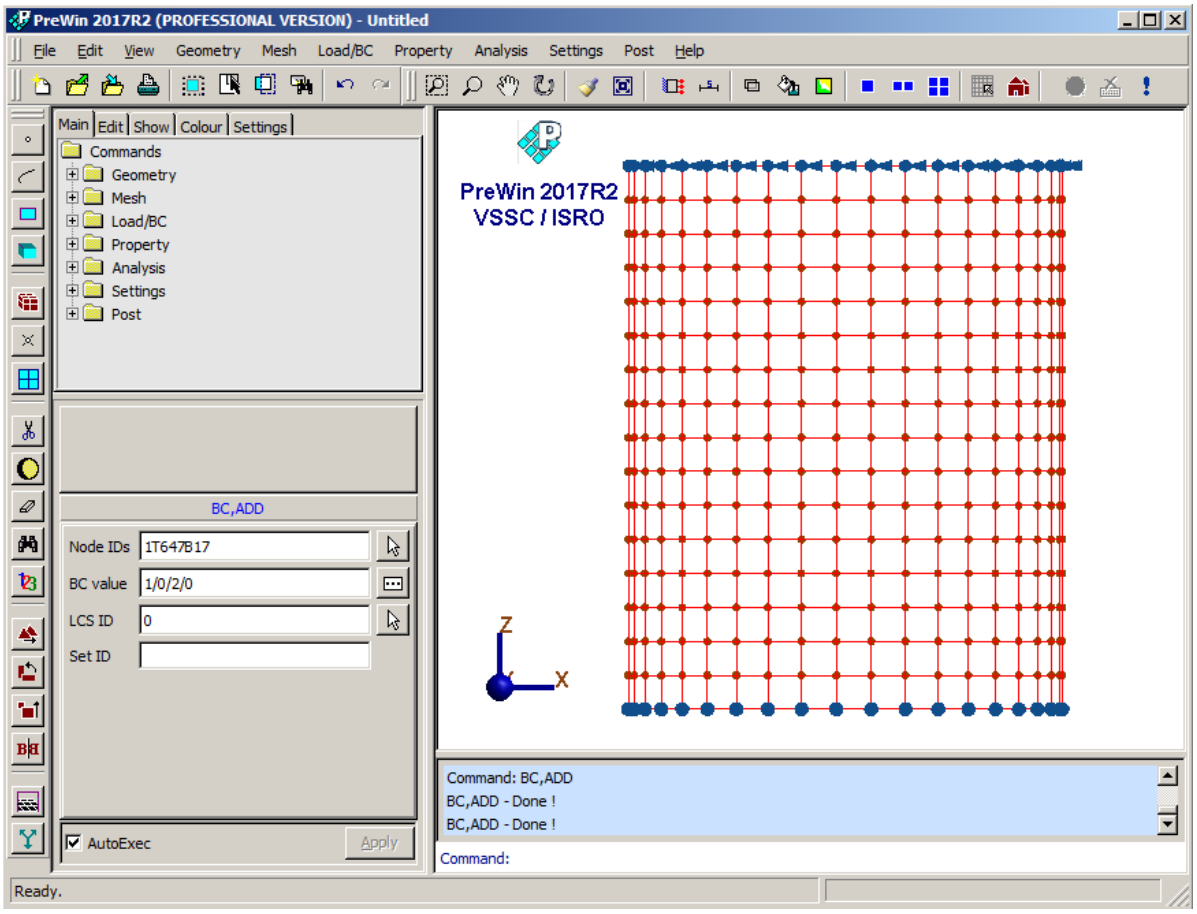
Node IDs	17T663B17
BC value	1/0/2/0/3/0/4/0/5/0/6/0
LCS ID	0

b. Specify boundary conditions at top edges

Parameters :

Node IDs	1T647B17
BC value	1/0/2/0
LCS ID	0

At the end of the above operations, your screen should look like this.



7. Specify Point load along top circumference of cylinder

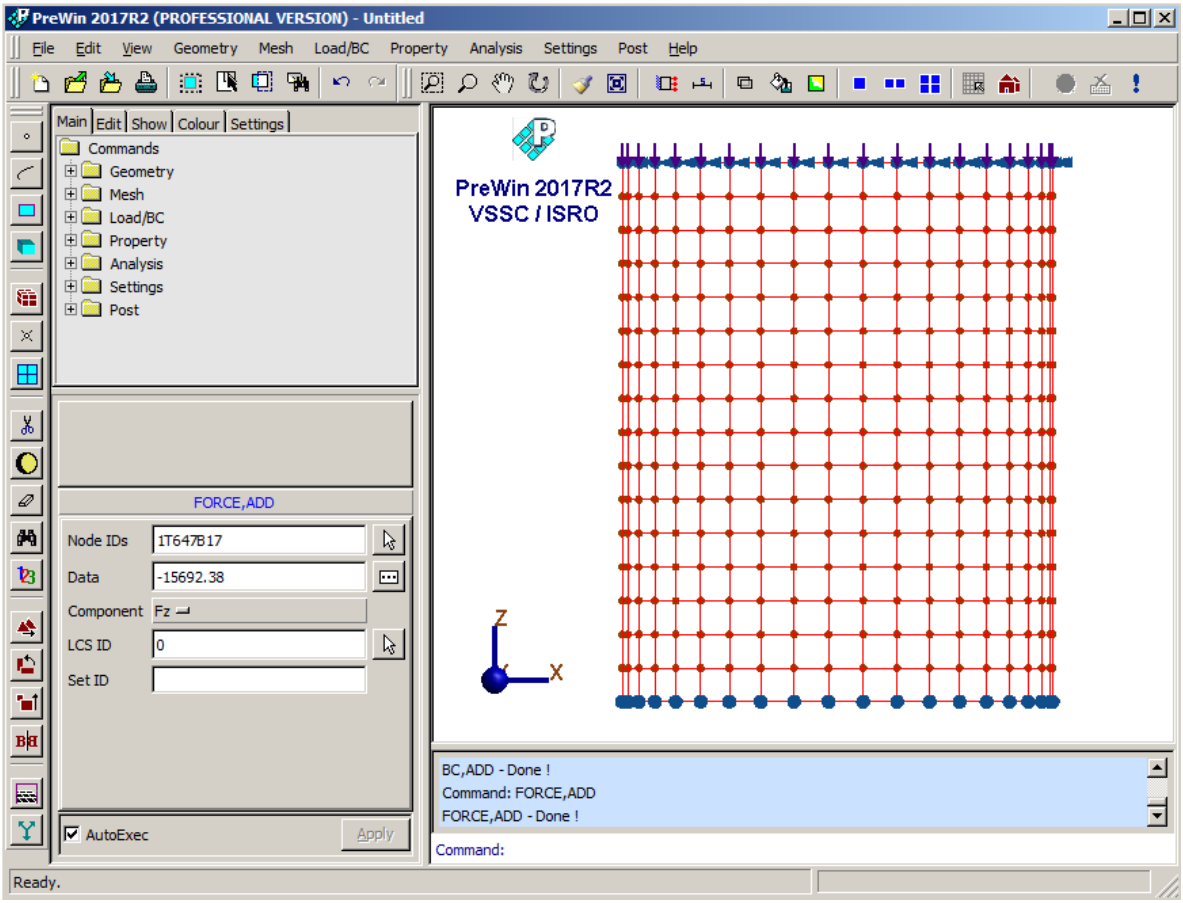
Command : FORCE, ADD

Menu : Load/BC→ Point Load→Add

Parameters :

Node IDs	Select top circumference nodes 1T647B17
Data	-15692.38
Component	Fz
LCS ID	1

At the end of the above operations, your screen should look like this.



8. Set the analysis type

Command : ANTYPE, SET

Menu : Analysis → Analysis Type

Parameters :

Analysis Type	Buckling

9. Set the analysis options

Command : ANOPTION, SET

Menu : Analysis → Analysis Options

Parameters :

Linear Solver	MultiFrontal

10. Specify Static generate data

Command : BUCKLING, ADD

Menu : Analysis →Buckling→General→Add

Parameters :

No of Eigen values	10
Stress Output	Yes

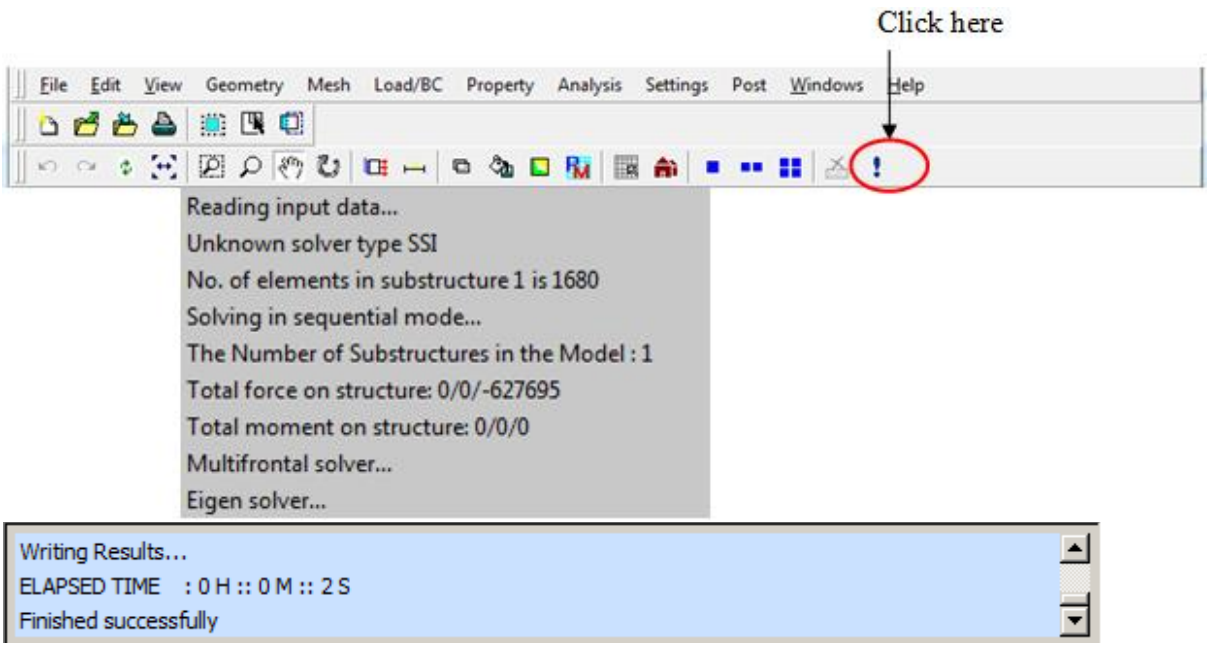
11. Save the project model

Menu : File → Save

12. Submit the job into FEAST

Menu : Analysis →Run solver

At the end of the above operations, your screen should look like this.



After the solution is completed the message *"successfully completed"* appears in the *message box*

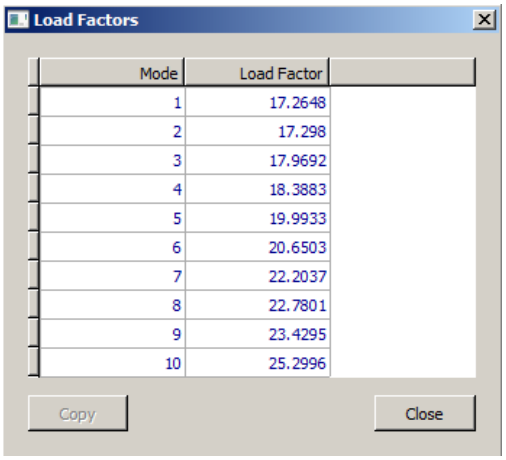
13.Perform post processing

a. Critical Load factor

Command : POST,VIEWRESULTS

Menu : Post → View results → Critical load

At the end of the above operations a table as shown below appears.



Load Factors

Mode	Load Factor
1	17.2648
2	17.298
3	17.9692
4	18.3883
5	19.9933
6	20.6503
7	22.2037
8	22.7801
9	23.4295
10	25.2996

Copy Close

b. Buckling Mode Shape

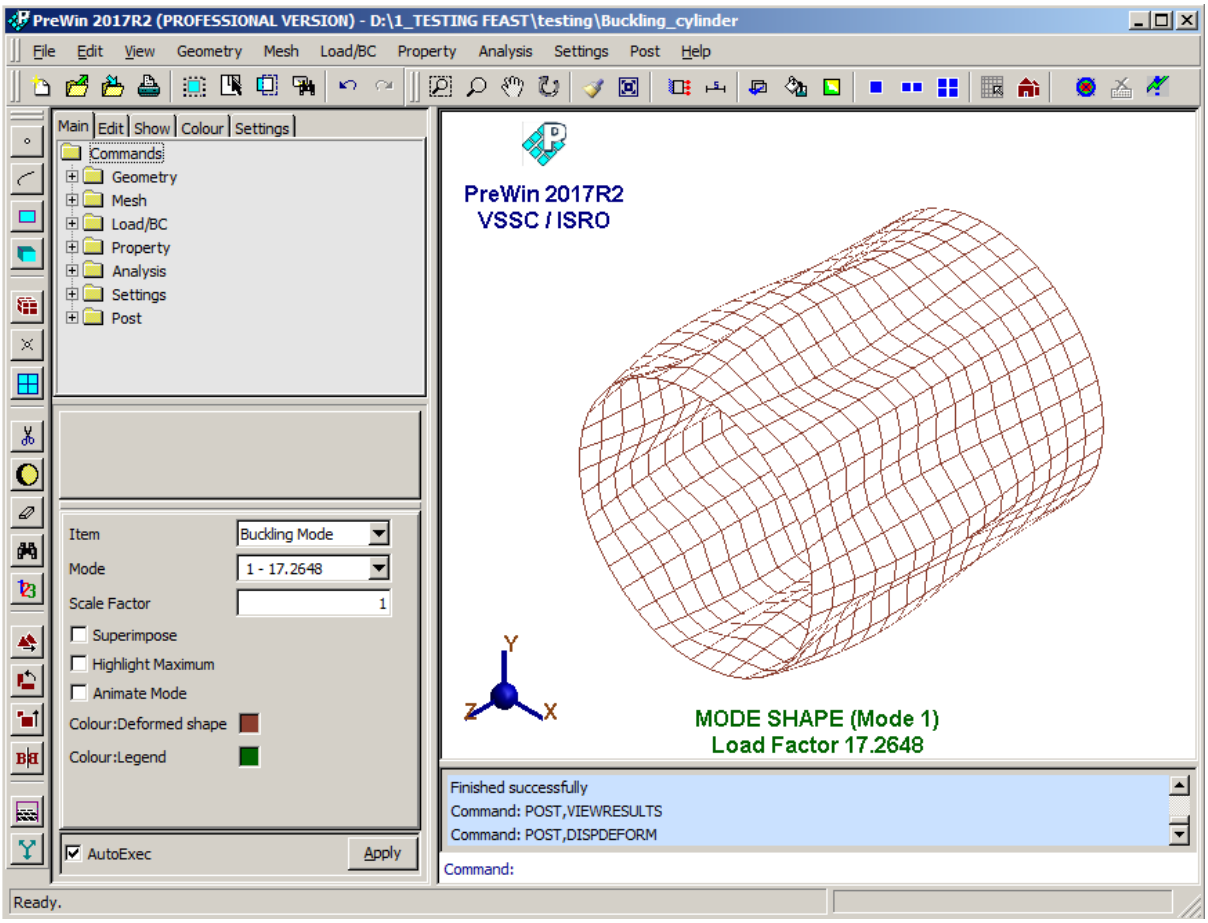
Command : POST,DISPDEFORM

Menu : Post → Deformed Shape→Buckling Mode

Parameters :

Item	Buckling Mode
Mode	1 - 17.2648
Scale Factor	1

At the end of the above operations, your screen should look like this



c. Output can be seen in *.OUT file