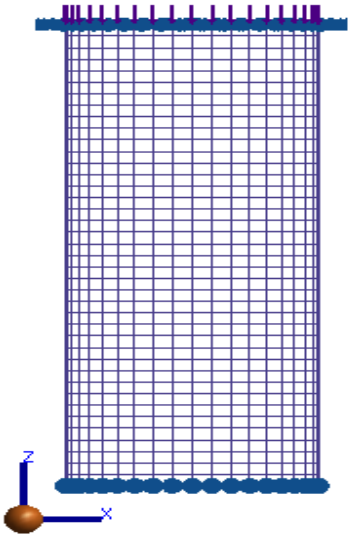
BUCKLING ANALYSIS OF CYLINDRICAL SHELL



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

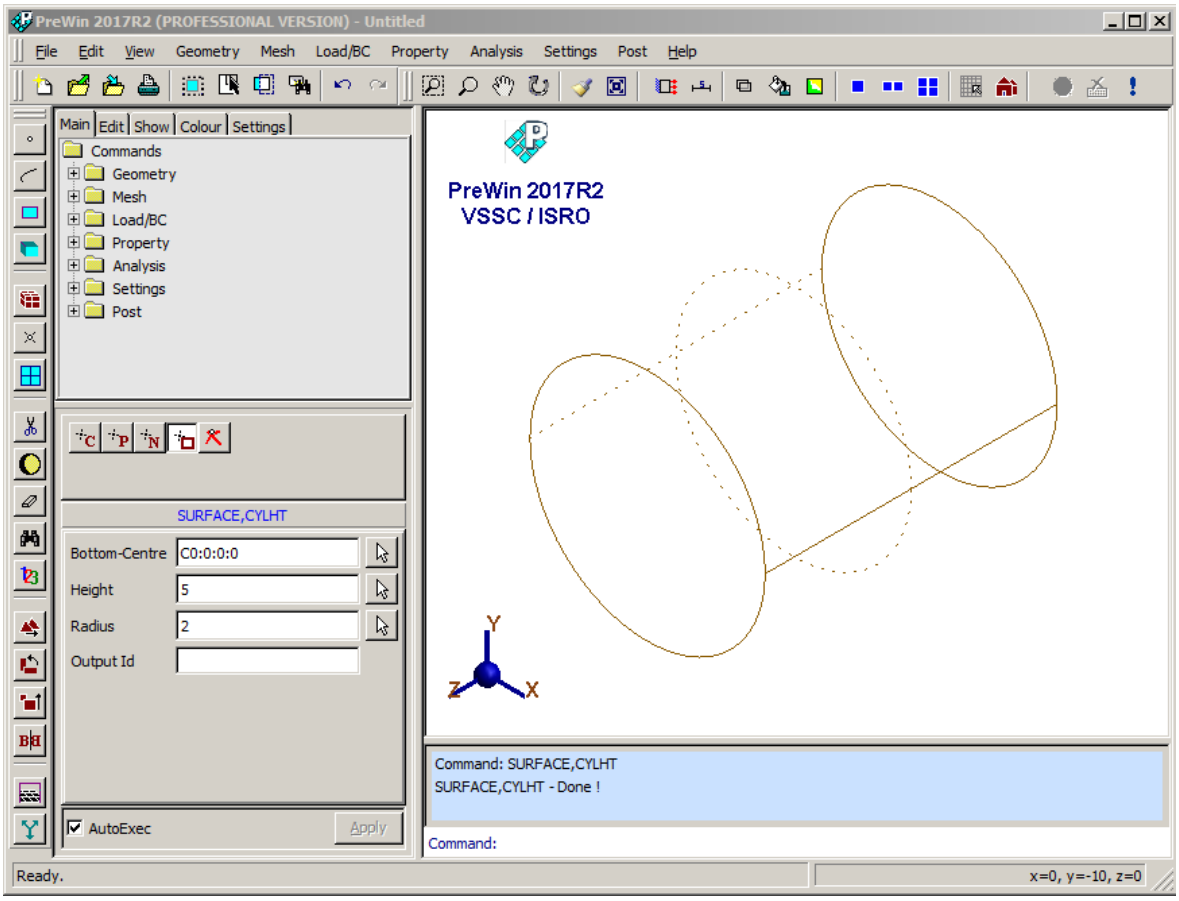
Command : SURFACE, CYLHT

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

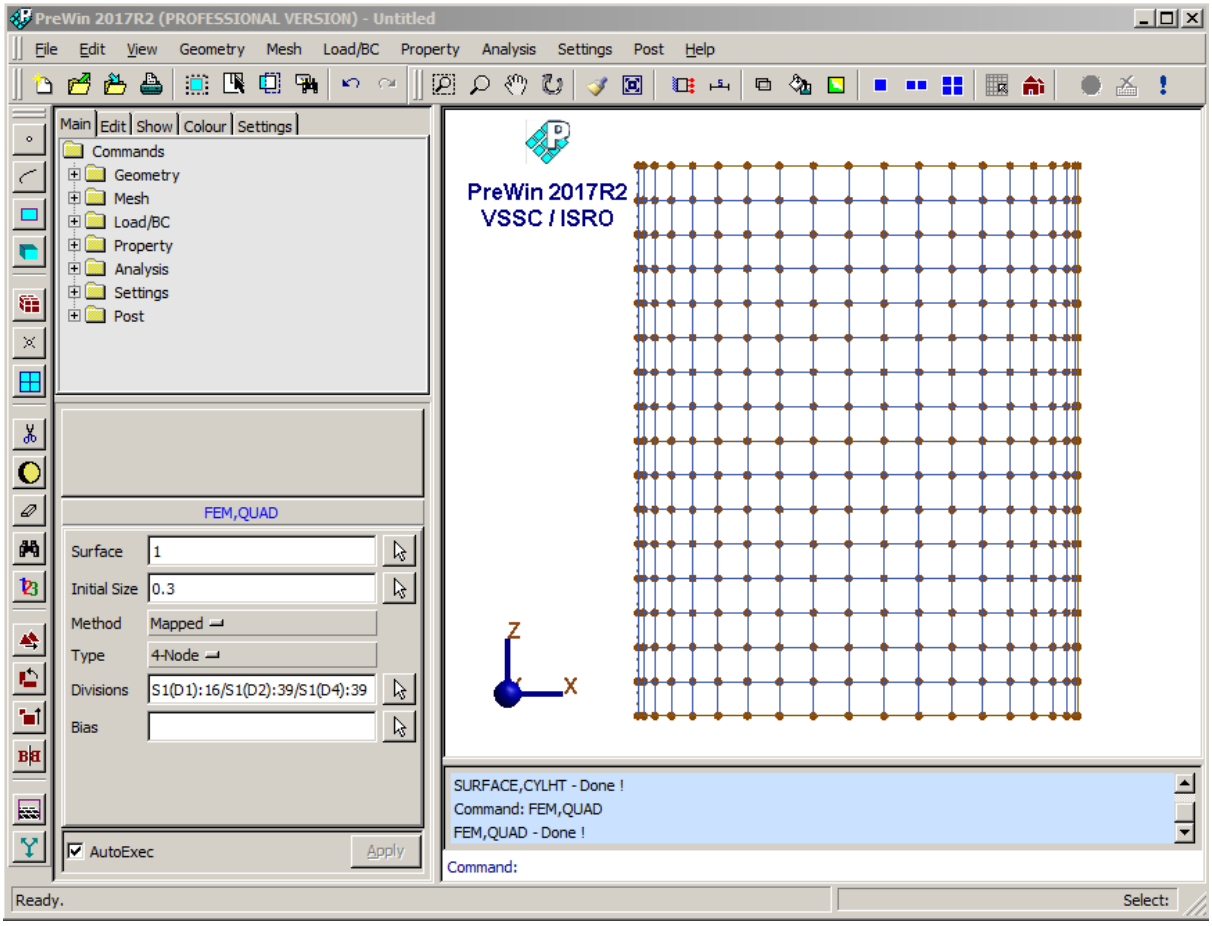
Command : FEM, QUAD

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

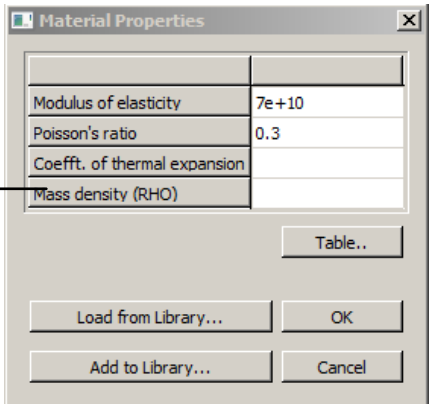
4. Specify material properties

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 :

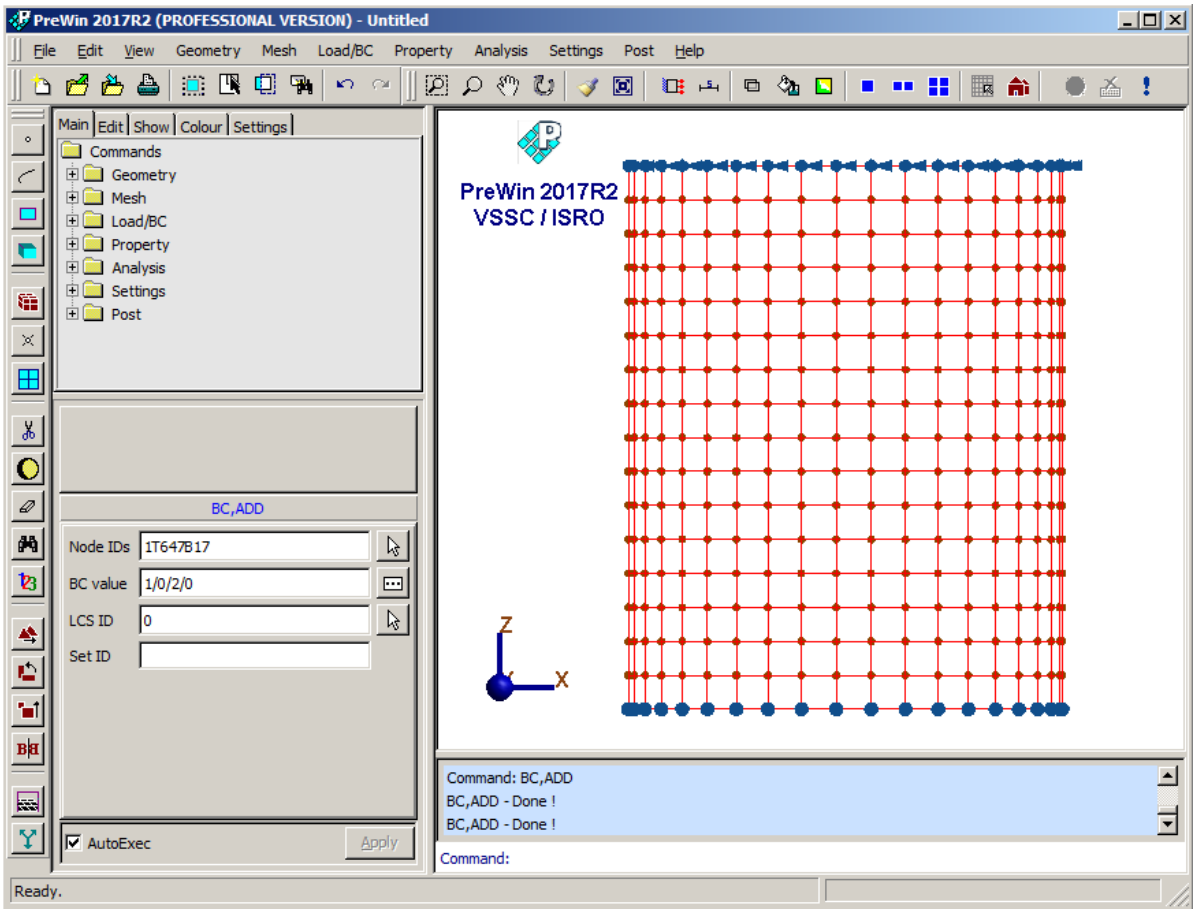
Node IDs	17T663B17
BC value	1/0/2/0/3/0/4/0/5/0/6/0
LCS ID	0

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

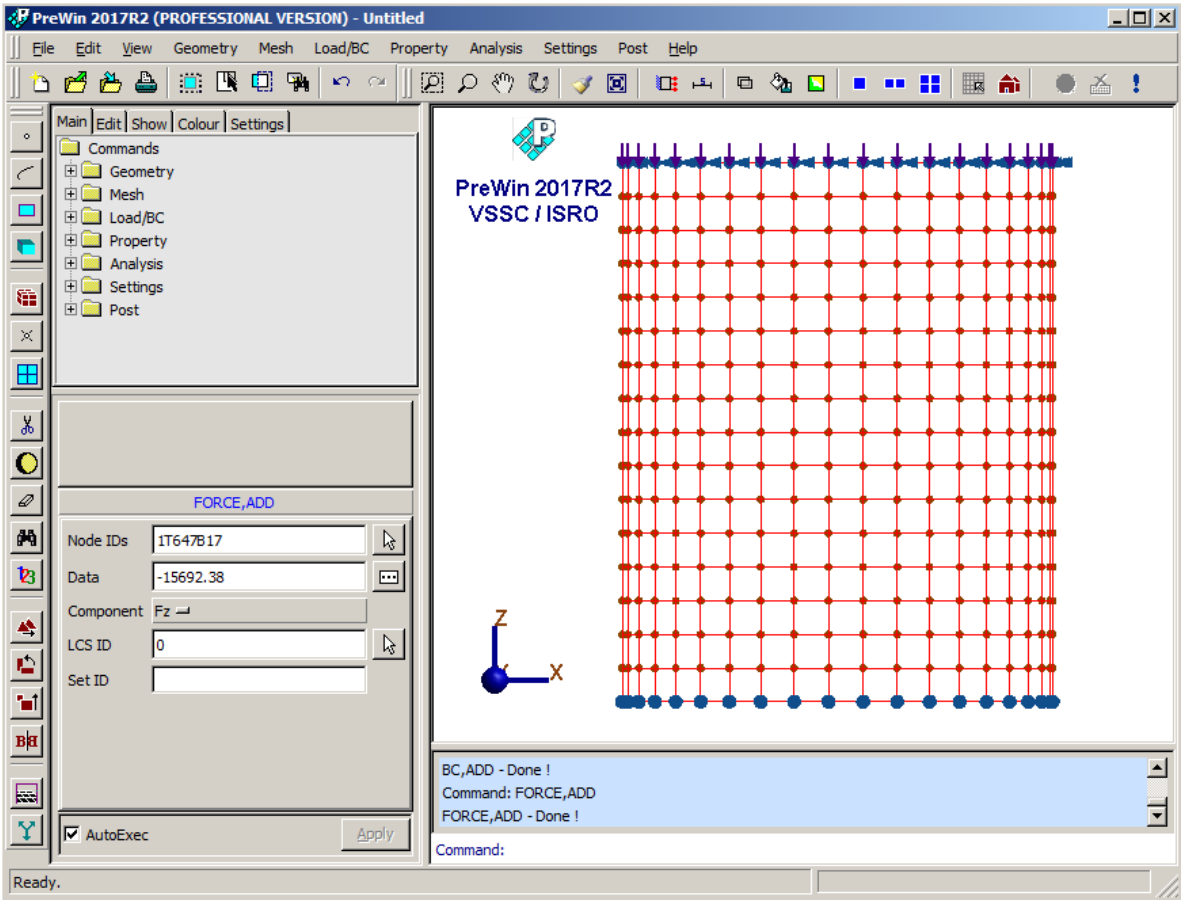
Command : FORCE, ADD

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

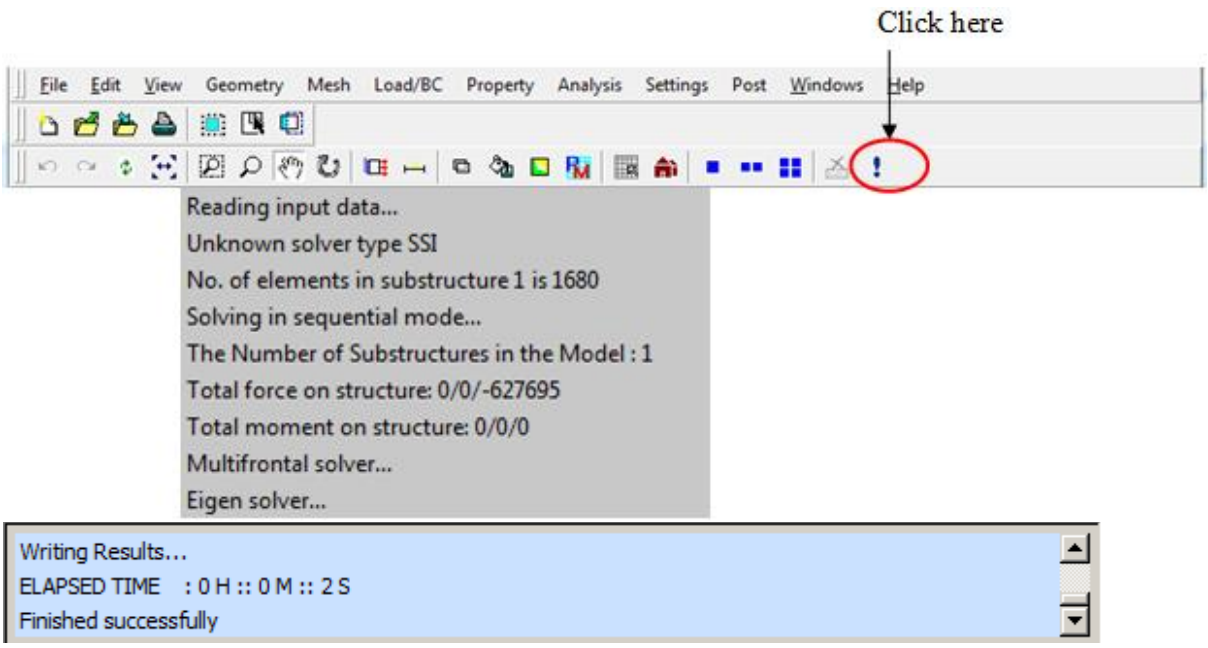
11. Save the project model

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*

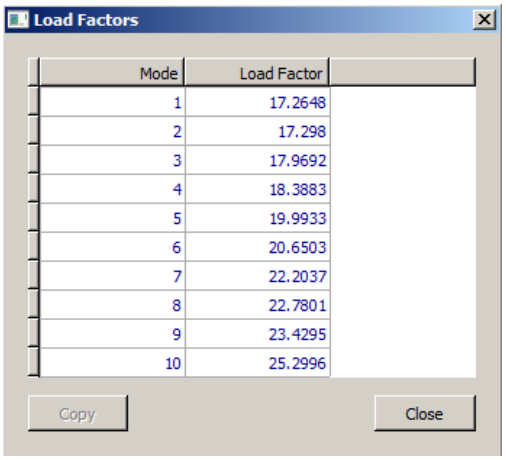
13.Perform post processing

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

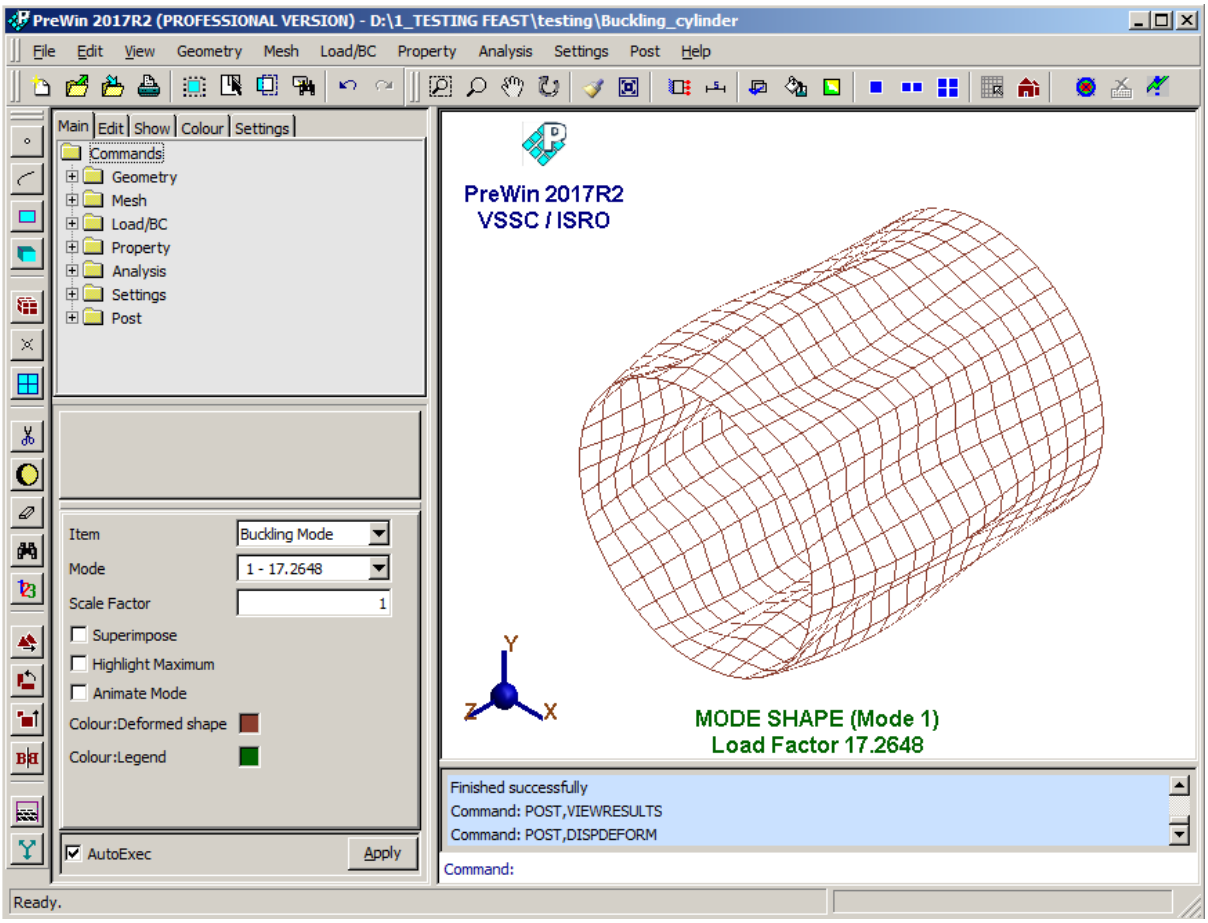
Command : POST,DISPDEFORM

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