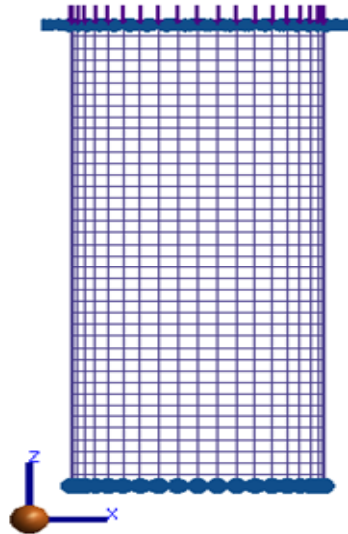


## BUCKLING ANALYSIS OF CYLINDRICAL SHELL



### Geometric Properties

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

### Boundary condition

Fixed at bottom surface  
 $U_x=U_y=0$  at top surface

### Material properties

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

### Loading condition

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

### Element Type

QUAD4 shell element

## PROCEDURE

### STEP

#### 1. Create a Cylindrical surface

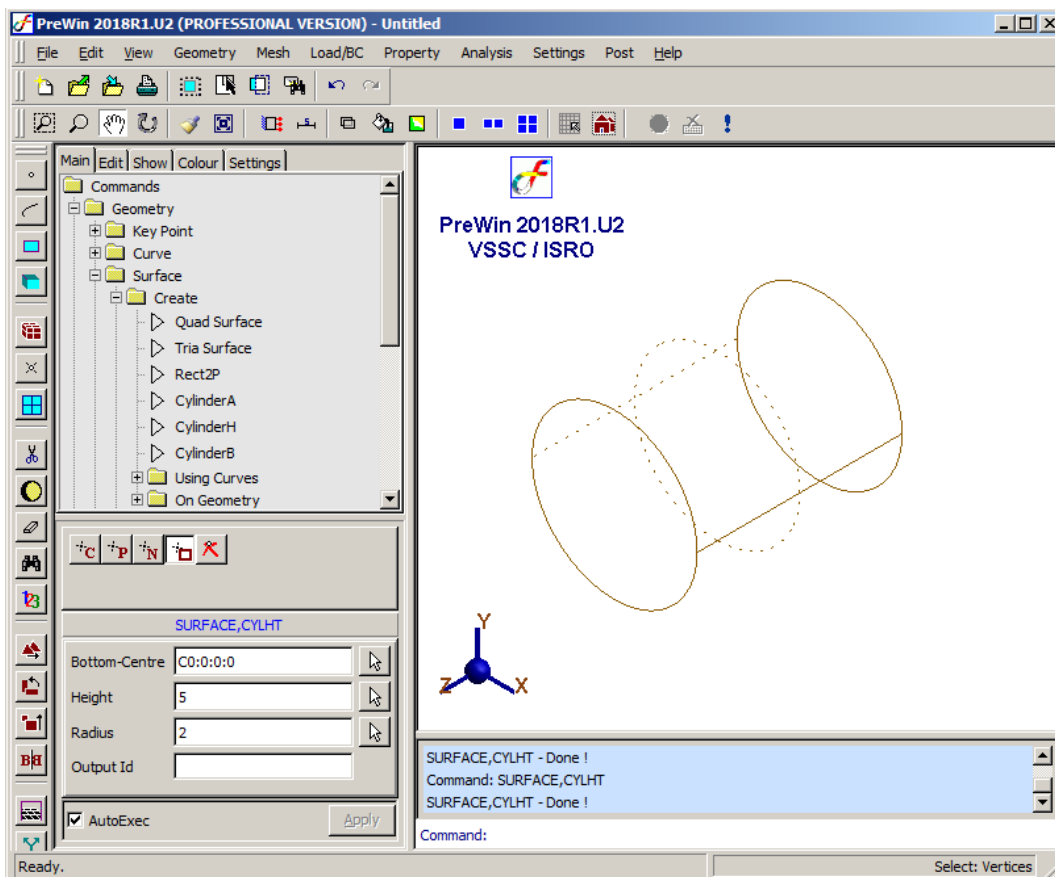
Command : SURFACE, CYLHT

Menu : Geometry → Surface → Create → CylinderH

Parameters

<b>Centre</b>	C0:0:0:0
<b>Height</b>	5
<b>Radius</b>	2

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



\*Now change to bottom view

## 2. Meshing the surface using quadrilateral elements

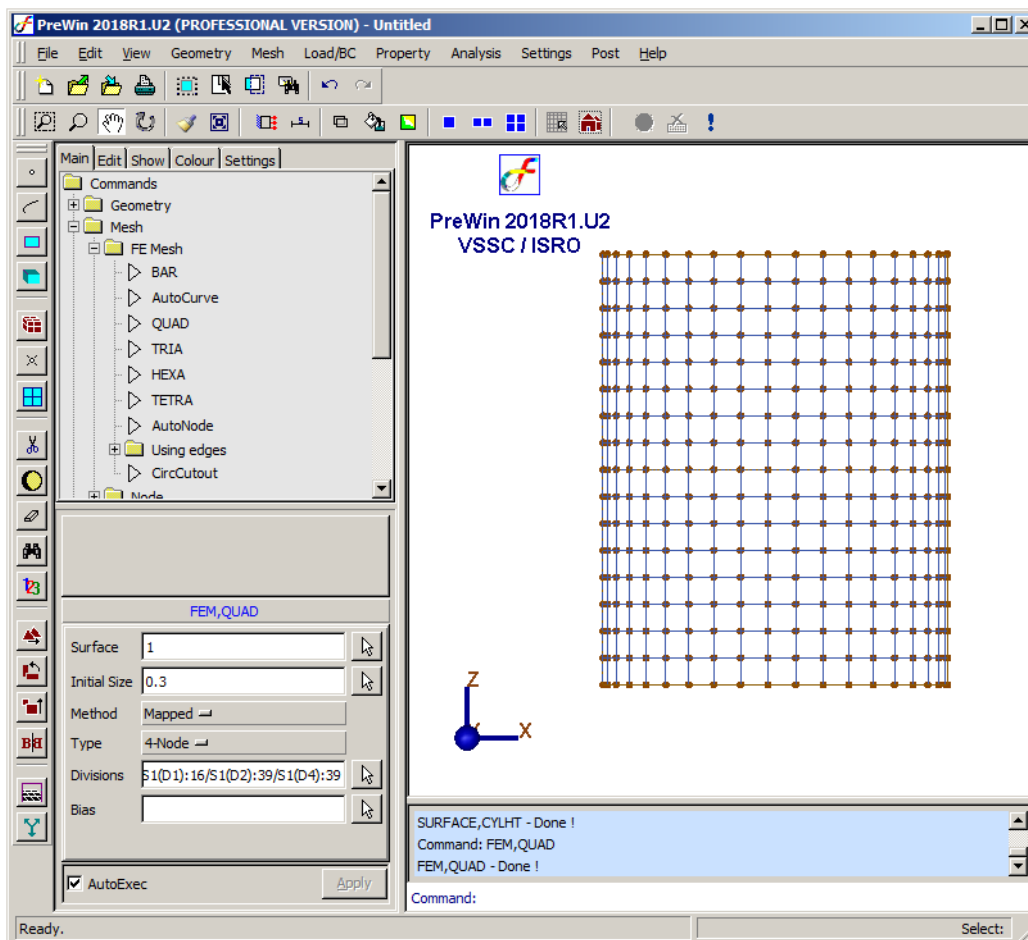
Command : FEM, QUAD

Menu : Mesh → FE Mesh → QUAD

Parameters :

<b>Surface IDS</b>	1
<b>Initial Size</b>	Give an approximate initial value by typing or picking two edge points
<b>Method</b>	Mapped
<b>Type</b>	4 Node
<b>Divisions</b>	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 : 

<b>Surface IDS</b>	1
--------------------	---

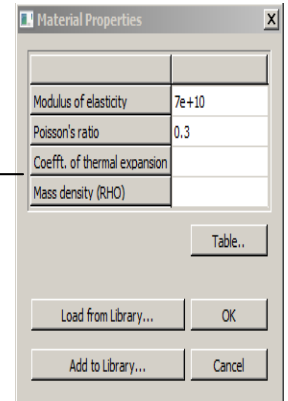
#### 4. Specify material properties

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic

Parameters : 

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



#### 5. Specify thickness

Command : THICKNESS, ADD

Menu : Property → Physical → Thickness → Add

Parameters : 

<b>Element IDs</b>	All
<b>Thickness</b>	0.005
<b>Set ID</b>	1

#### 6. Specify Displacement Boundary Conditions

##### a. Specify boundary conditions at bottom edges

Command : BC, ADD

Menu : Load/BC → Displacement BC → Add

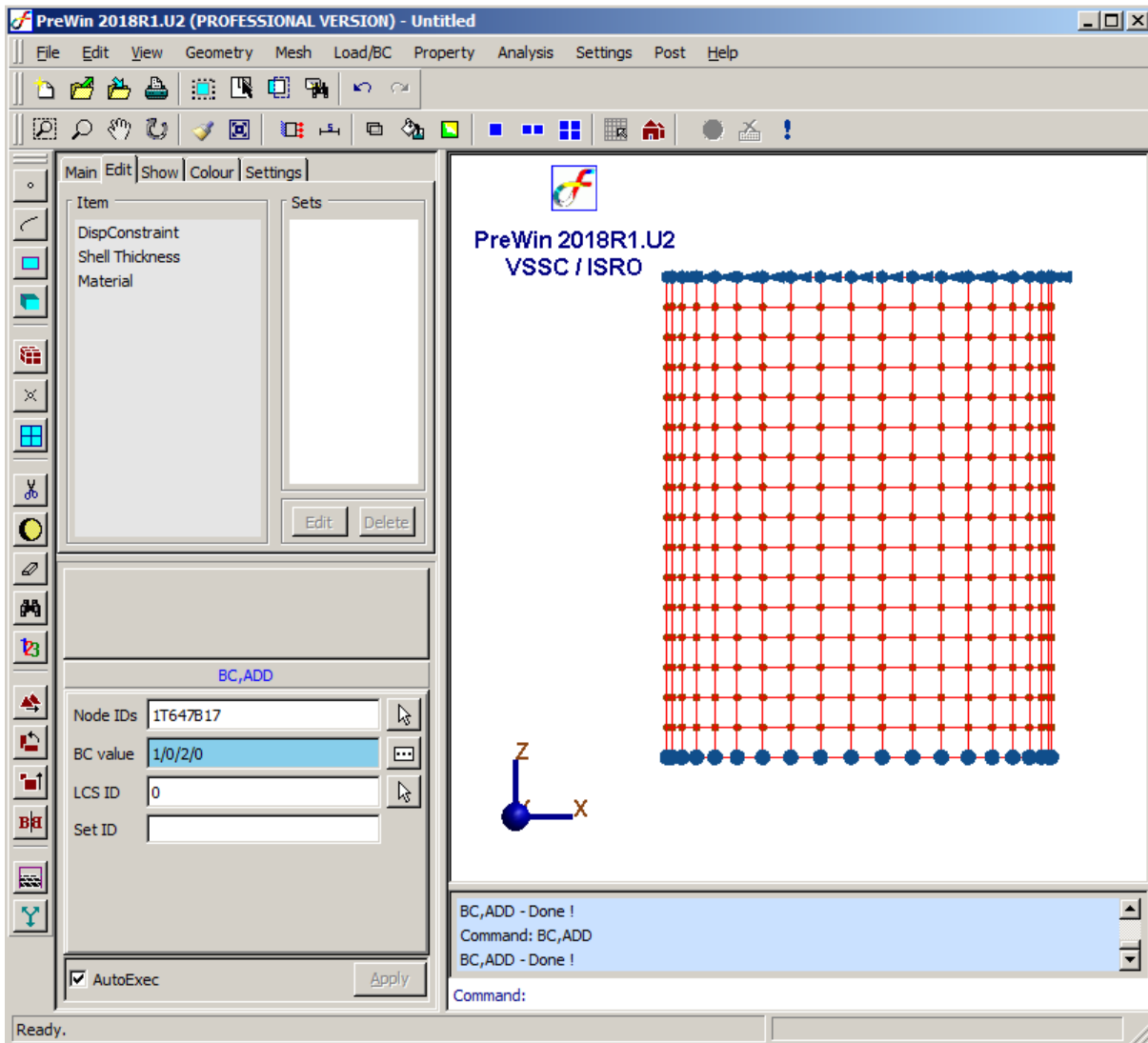
Parameters : 

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

**b. Specify boundary conditions at top edges**

Parameters :

<b>Node IDs</b>	1T647B17
<b>BC value</b>	1/0/2/0
<b>LCS ID</b>	0



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

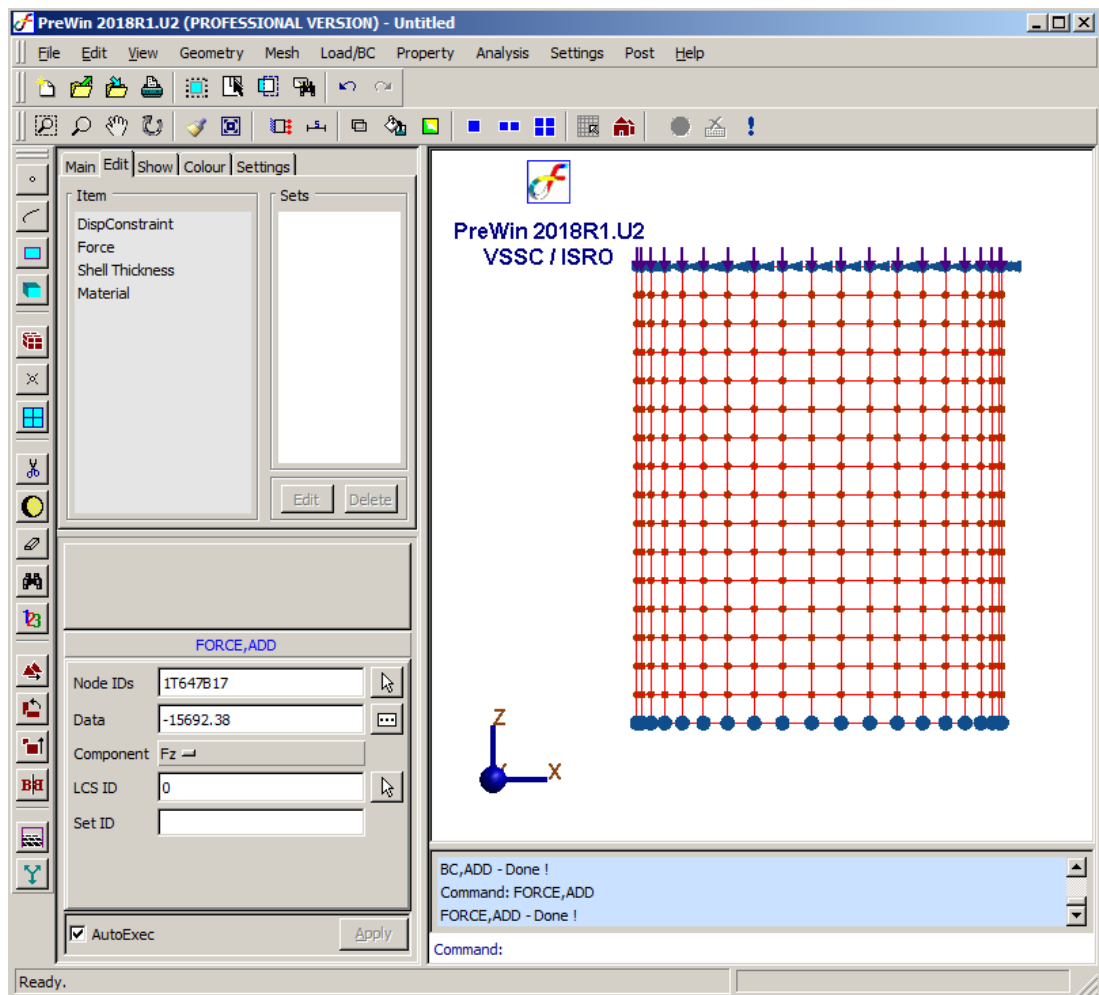
### 7. Specify Point load along top circumference of cylinder

Command : FORCE, ADD

Menu : Load/BC → Point Load → Add

Parameters :

<b>Node IDs</b>	Select top circumference nodes 1T647B17
<b>Data</b>	-15692.38
<b>Component</b>	Fz
<b>LCS ID</b>	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 :

<b>Analysis Type</b>	Buckling
----------------------	----------

## 9. Set the analysis options

Command : ANOPTION, SET

Menu : Analysis → Analysis Options

Parameters :

<b>Linear Solver</b>	MultiFrontal
<b>Eigean solver</b>	Lanczos

## 10. Specify Static generate data

Command : BUCKLING, ADD

Menu : Analysis → Buckling → General → Add

Parameters

<b>No of Eigen values</b>	10
<b>Stress Output</b>	Yes

:

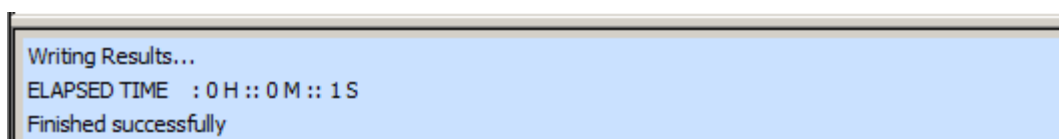
## 11. Save the project model

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.

Mode	Load Factor
1	17.2648
2	17.9692
3	18.3883
4	19.9933
5	20.6505
6	22.4511
7	23.8098
8	26.844
9	27.3237
10	28.0623

#### b. Buckling Mode Shape

Command : POST,DISPDEFORM

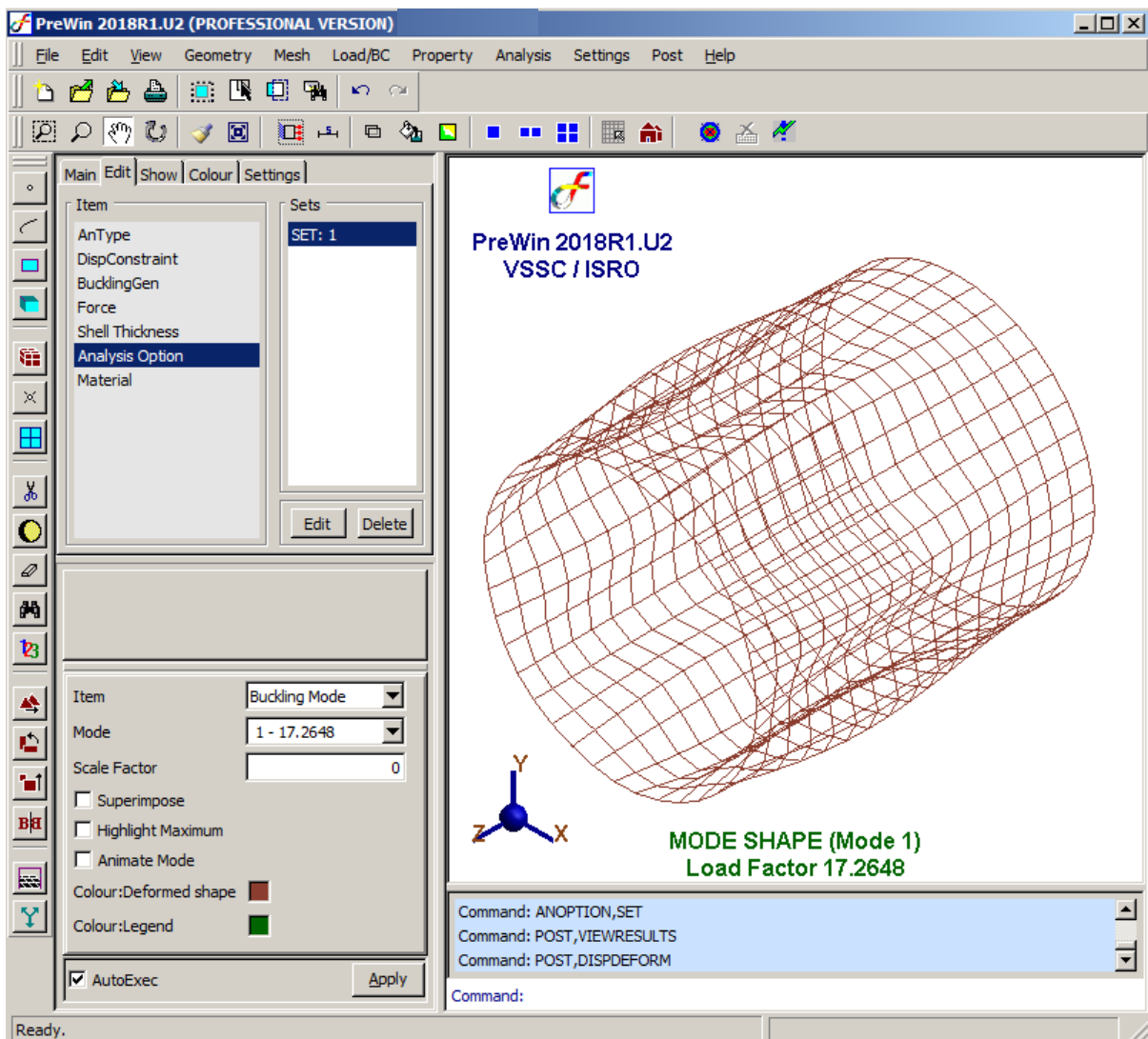
Menu : Post → Deformed Shape → Buckling Mode

Parameters :

<b>Item</b>	Buckling Mode
<b>Mode</b>	1 - 17.2648
<b>Scale Factor</b>	1



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



c. Output can be seen in \*.OUT file