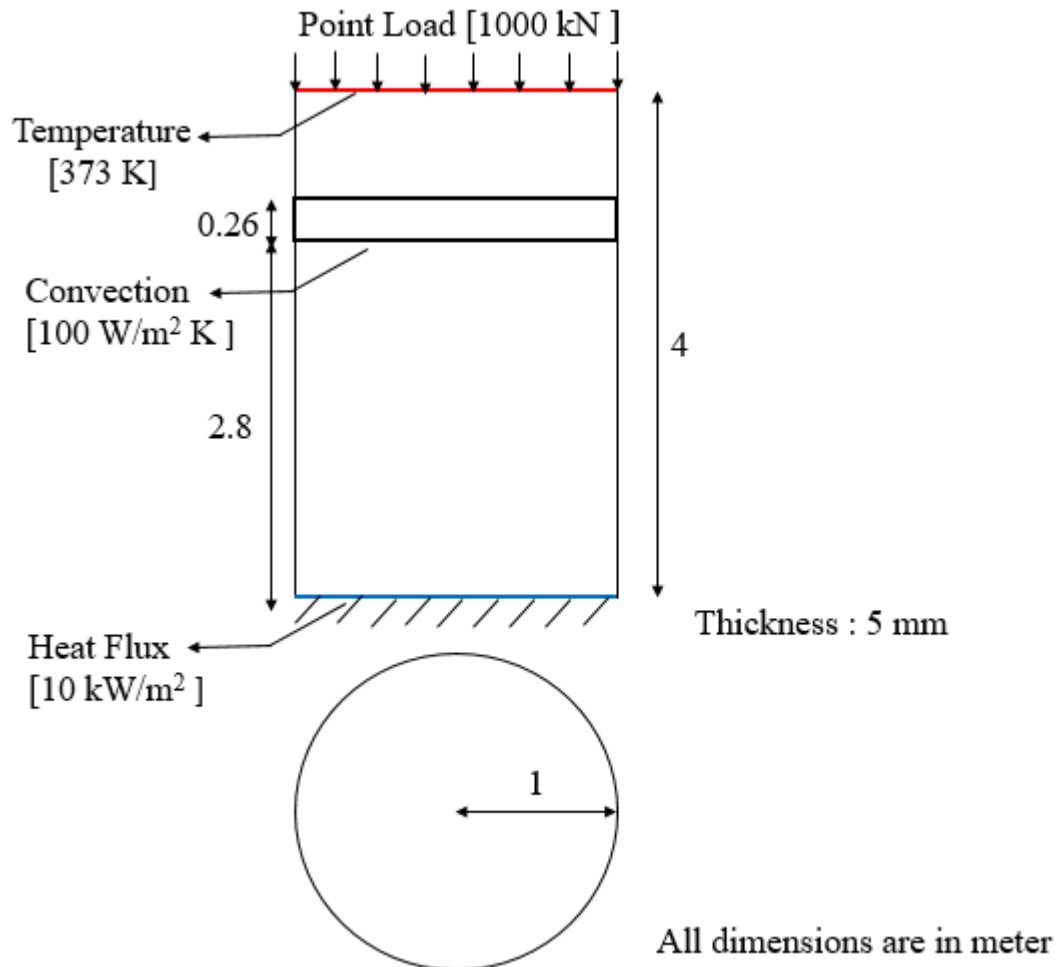




HEAT TRANSFER AND THERMOSTRUCTURAL ANALYSIS OF A CYLINDER



An aluminium cylinder of 4m height, 5mm thickness and 1m radius is considered for this tutorial. The outer surface of the cylinder is adiabatically sealed and a portion of the cylinder 26 cm at a height 2.8 m is exposed to the surrounding which has a convective coefficient of 100 W/m² K at 300 K. A constant heat flux 10 kW/m² is acting at the bottom, whereas the top edge of the cylinder is maintained at 373 K.

A structural load of 1000 kN is acting at the top edge of the cylinder and the bottom of the cylinder is assumed to be fixed. The objective of this analysis is to find the maximum stress due to the thermal environment along with the mechanical loads.



PROCEDURE

1. Create Keypoints

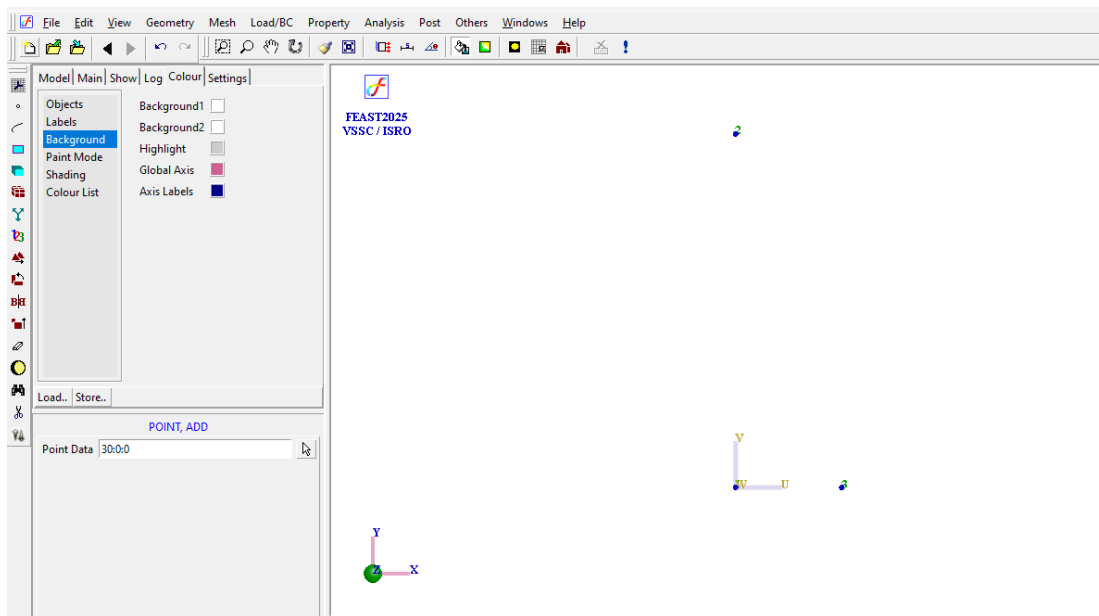
Command: POINT, ADD

Menu : Geometry → Keypoint → Create → ADD



Similarly create key points at (0:4:0) and (1:0:0)

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



2. Create Cylindrical Surface

Command: SURFACE, CYLAXIS

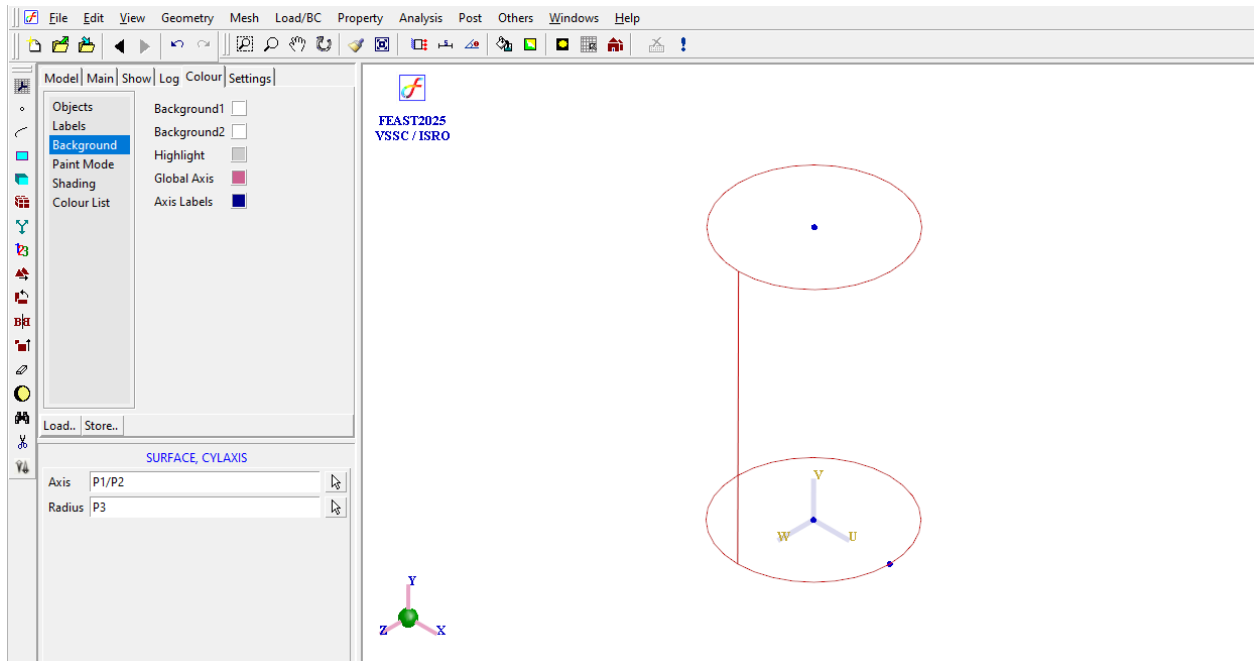
Menu : Geometry → Surface → Create → Cylaxis



Parameters:

SURFACE, CYLAXIS	
Axis	P1/P2
Radius	P3

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



3. Generate mesh

Command: MESH, QUAD

Menu : Mesh → MeshGen → Quad

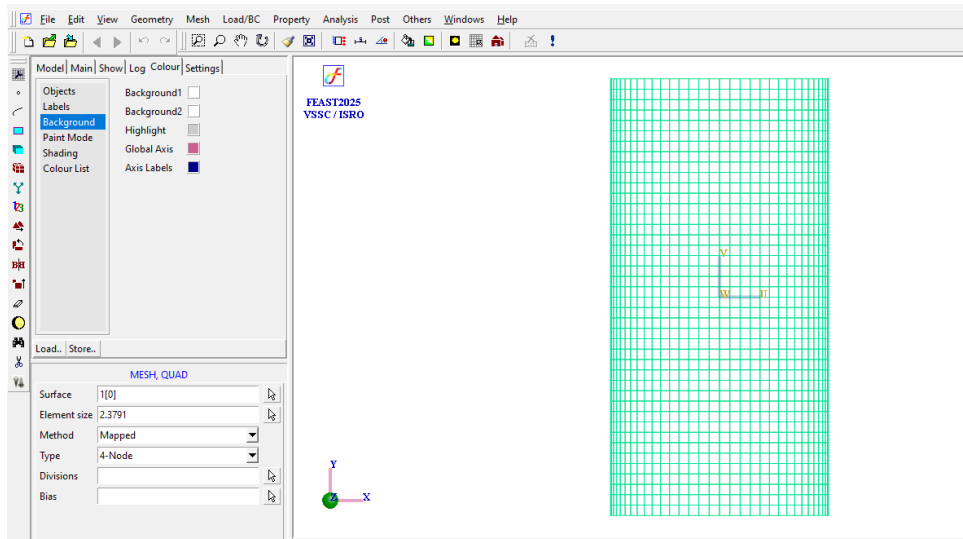
Parameters:

MESH, QUAD	
Surface	1[0]
Element size	0.130908
Method	Mapped
Type	4-Node
Divisions	
Bias	



Element size can be entered manually or by clicking two points on the geometry edge. Then click on the 'Divisions' command box, so that node divisions will be displayed on the geometry. The element subdivisions can be increased or decreased by left clicking or right clicking respectively.

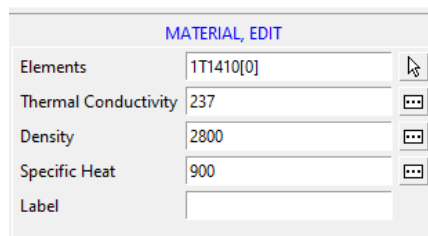
At the end of the operation your screen should look like this.



4. Apply Material Property

Command: MATERIAL, HTISOTROPIC

Menu : Property→Material →Thermal →Isotropic



Parameters:

5. Apply Thickness

Command: THICKNESS, ADD

Menu : Property→Physical →Thickness

Parameters:

THICKNESS, EDIT	
Elements	1T1410[0]
Thickness	0.005
Label	

6. Apply Convective Boundary Condition

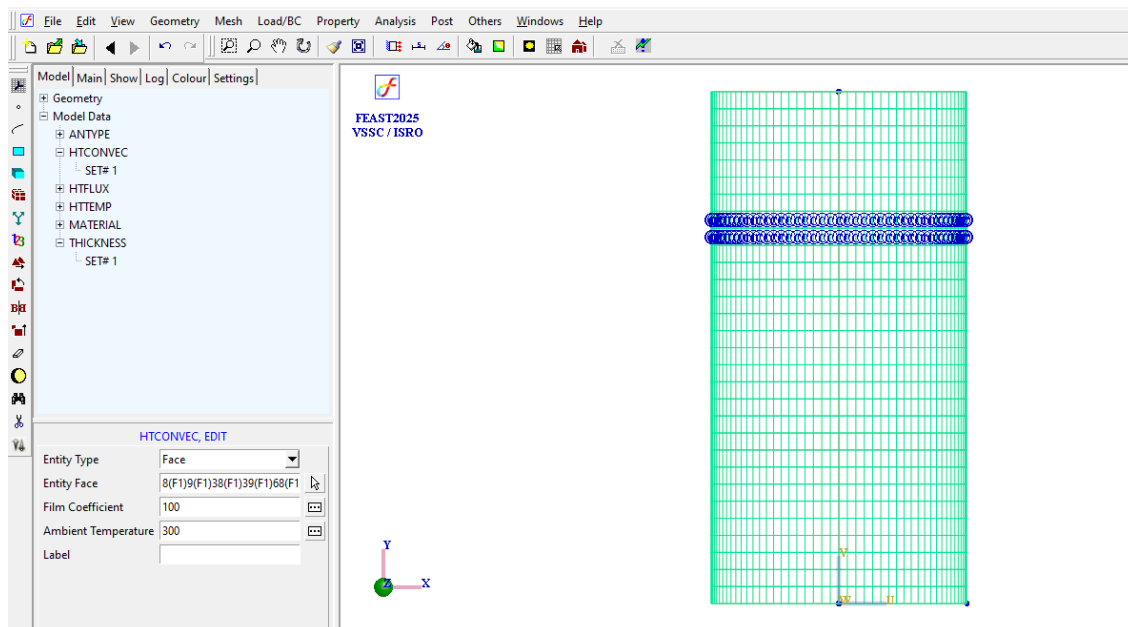
Command: HTCONVEC, ADD

Menu : Load/BC→Thermal→Convection

Parameters:

HTCONVEC, EDIT	
Entity Type	Face
Entity Face	8(F1)9(F1)38(F1)39(F1)68(F1)
Film Coefficient	100
Ambient Temperature	300
Label	

The selected face is look like below



7. Apply Heat Flux Boundary Condition

Command: HTFLUIX, ADD

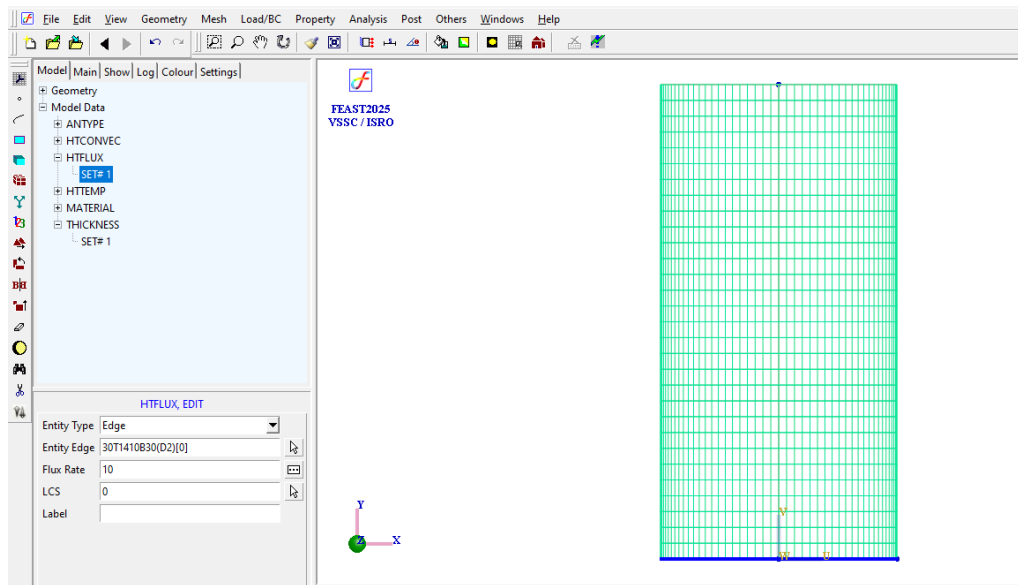
Menu : Load/BC→Thermal→Heat flux



HTFLUX, EDIT	
Entity Type	Edge
Entity Edge	30T1410B30(D2)[0]
Flux Rate	10
LCS	0
Label	

Parameters:

The defined edge is look like this,



8. Apply Temperature Boundary Condition

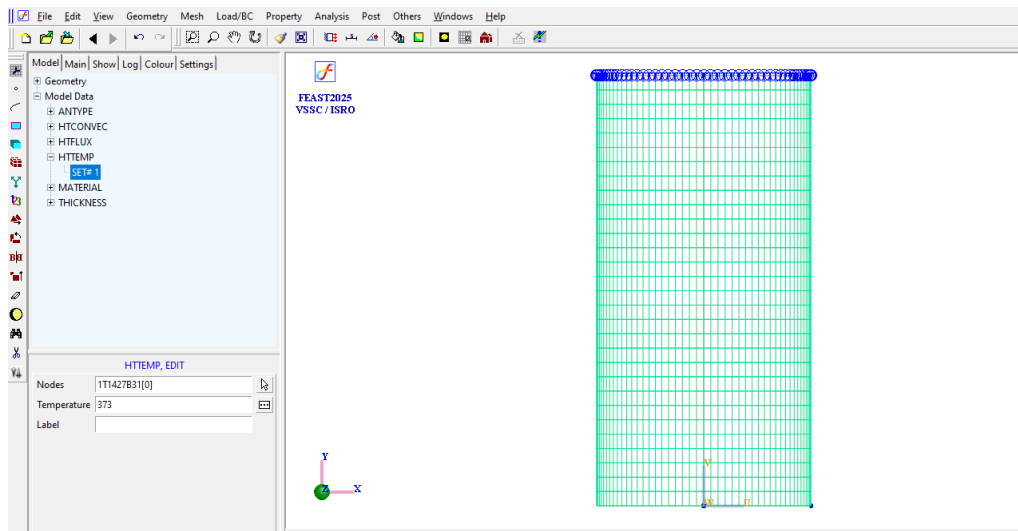
Command: HTEMP, ADD

Menu : Load/BC→Thermal→Temperature

HTEMP, EDIT	
Nodes	1T1427B31[0]
Temperature	373
Label	

Parameters:

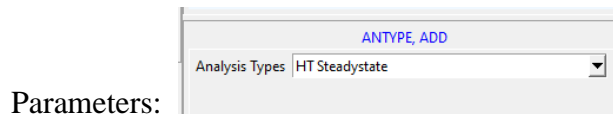
The defined nodes is look like this,



9. Set Analysis Type

Command: ANTYPE, SET

Menu : Analysis → Analysis Type



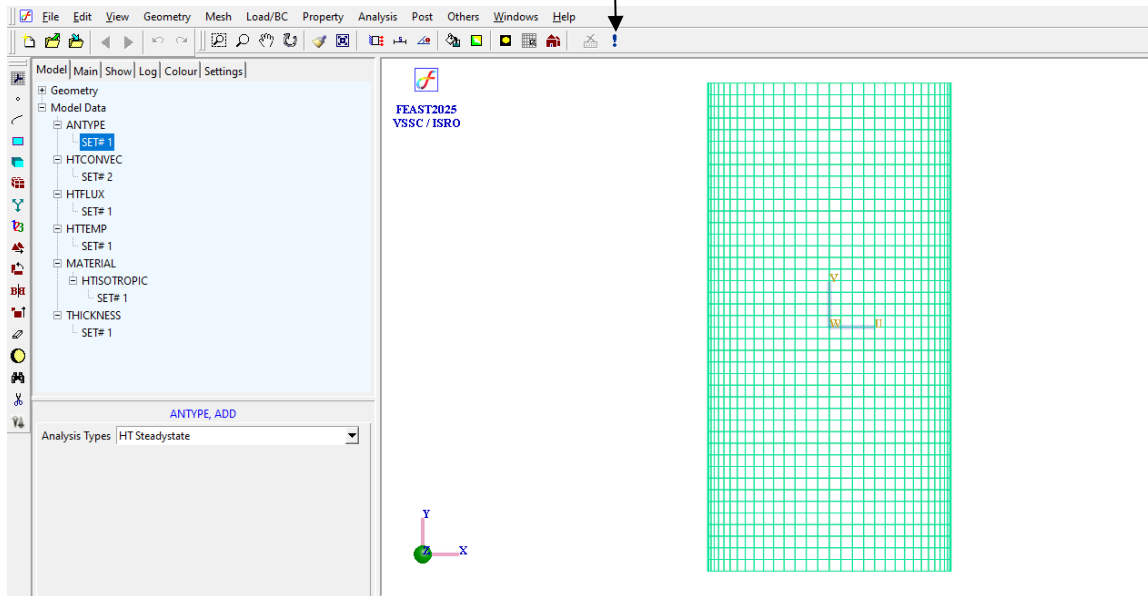
10. Save the project

Menu: File → Save



11. Activate solver

Click Here



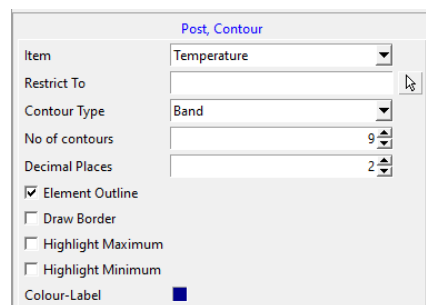
12. Perform Post Processing

i) Graph plots for displacement/ velocity/ acceleration

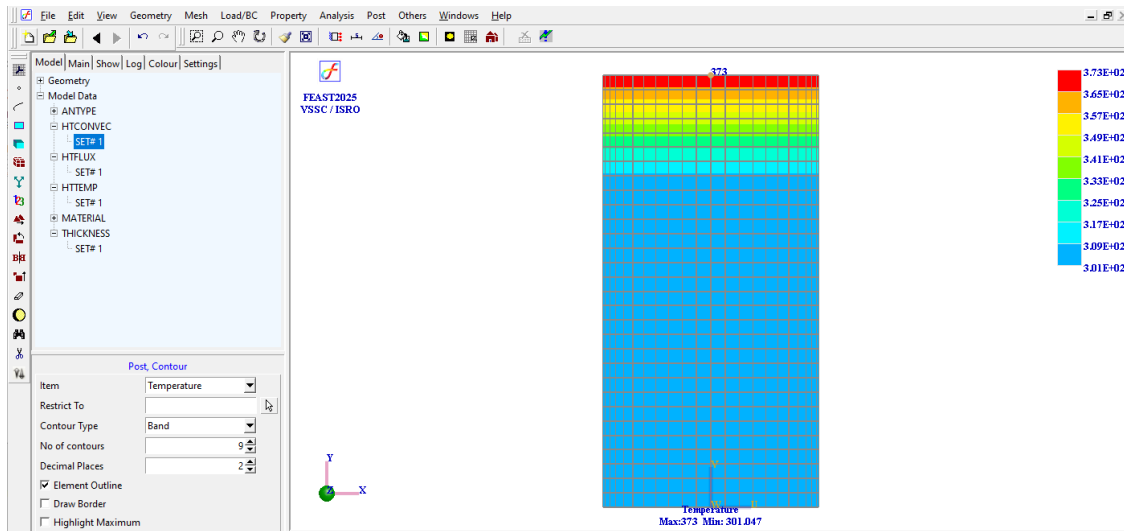
Command: POST, CONTOUR

Menu : Post→Contour

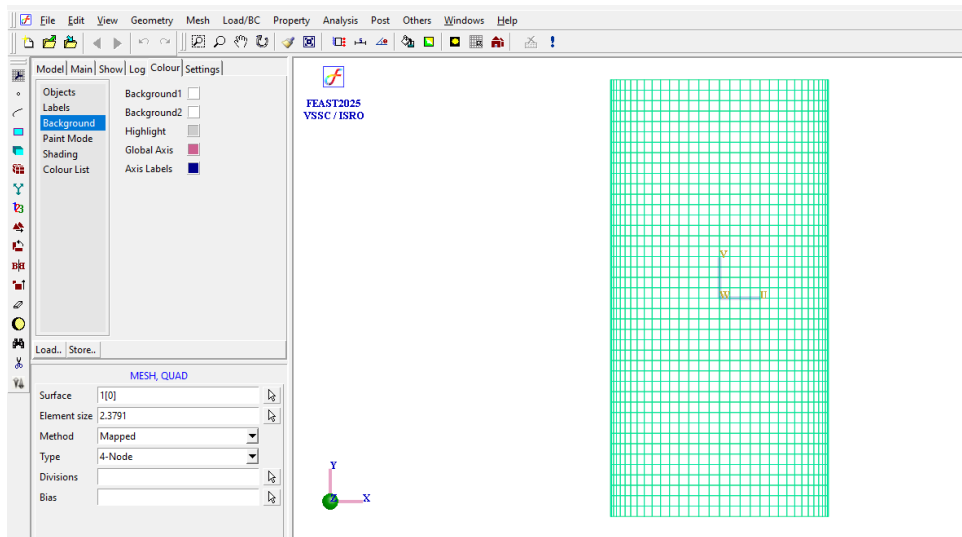
Parameters:



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



To perform thermo-structural analysis, the same model with load of 1000 kN at top edge of the cylinder and the procedure for this analysis is detailed below.



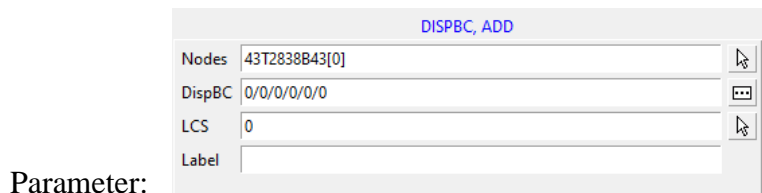


After meshing follow below steps,

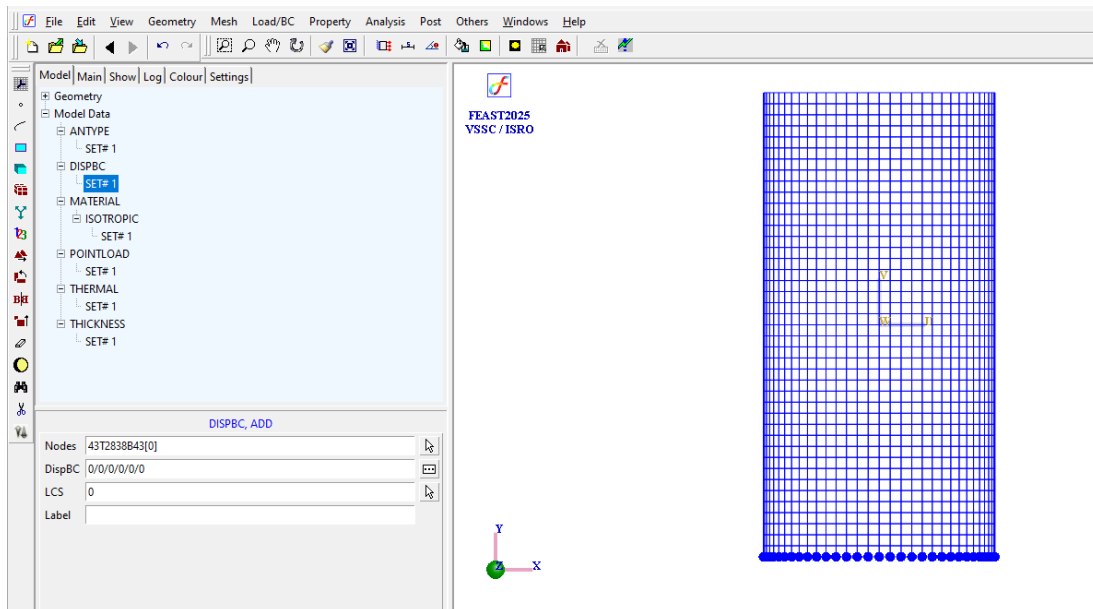
1. Apply Boundary Condition

Command: DISPBC, ADD

Menu : Load/BC→Structural→Displacement



At the end of the operation your screen look like this,



2. Apply Material Property

Command: MATERIAL, ISOTROPIC

Menu : Property→Material →Structural →Isotropic



MATERIAL, ISOTROPIC	
Elements	ALL
Young's Modulus	70E09
Nu	0.3
Density	2800
Alpha	
Label	

Parameter:

Thickness is same as applied in heat transfer analysis

3. Apply Point load

Command: POINTLOAD, ADD

Menu : LoadBC → Structural → Point load

POINTLOAD, EDIT	
Nodes	1T1427B31[0]
Magnitude	-1000e+3
Component	FY
LCS	
Label	

Parameter:

The node for point load is at the top end

4. Apply Thermal load

Command: THERMAL, ADD

Menu : LoadBC → Structural → Thermal load

THERMAL, ADD	
Nodes	ALL
Temperature Source	File
File name	E:\Thermostructural\cylinder-HT_1m_4m.OUT
Reference temperature	0
Label	

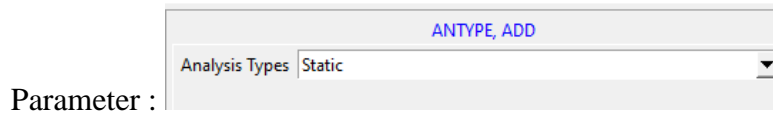
Parameter:

Select the *.out file from heat transfer analysis file location.

5. Set Analysis type

Command: ANTYPE, SET

Menu : Analysis → Analysis Type



After that follow steps from 10 and 11 from heat transfer analysis

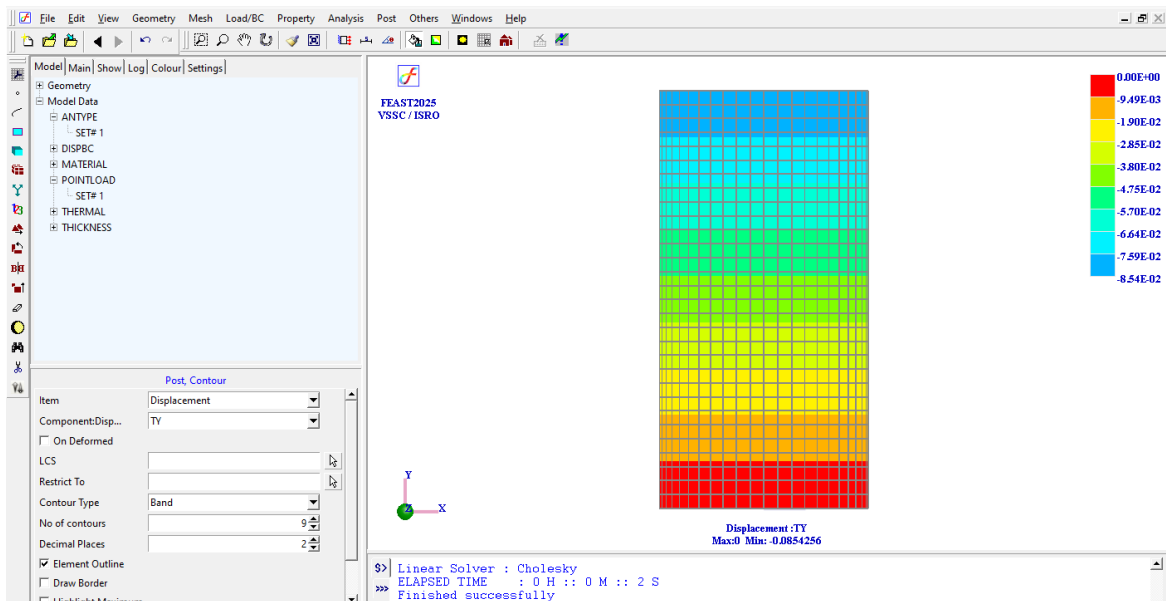
1. Perform Post processing

Command: POST, CONTOUR

Menu : Post→Contour

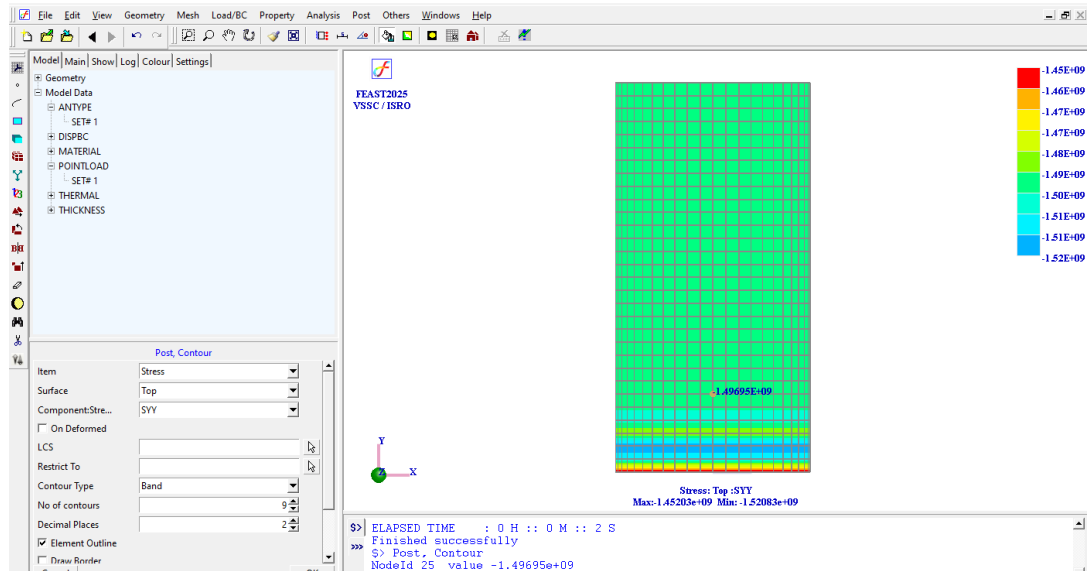


At the end of the operation your screen should look like this.



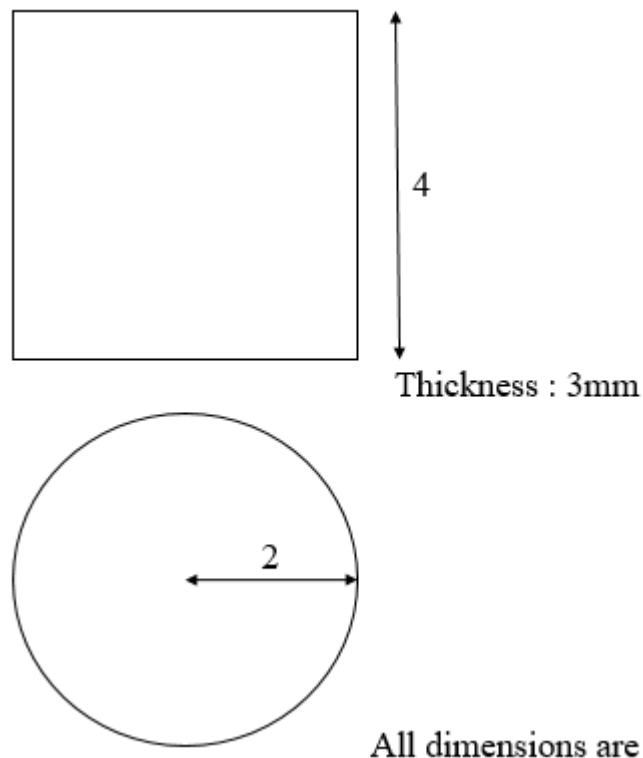


To find stress contour in y direction,





FREE VIBRATION ANALYSIS OF A CYLINDER



An aluminium cylinder of radius 2m and height 4m is constrained at the bottom through 24 bolted joints. The thickness of the cylinder is 3mm. A mass of 500 kg is lumped at a height of 5m from the cylinder top surface. The mass is connected with the top of the cylinder through 24 bolted joints. Initially a free vibration analysis is performed to estimate the natural frequencies and mode shapes of the system.

A 1g steady state lateral base excitation is applied at the bottom of the cylinder and the response at the mass is estimated. This is done by performing a frequency response base excitation analysis. The lateral base excitation is applied to the cylinder through a heavy inertial mass 50 times greater than the system mass. The inertial mass is connected to the cylinder bottom portion through 24 number of rigid links. 2% critical damping is assumed for all the modes.



PROCEDURE

2. Create Key points

Command: POINT, ADD

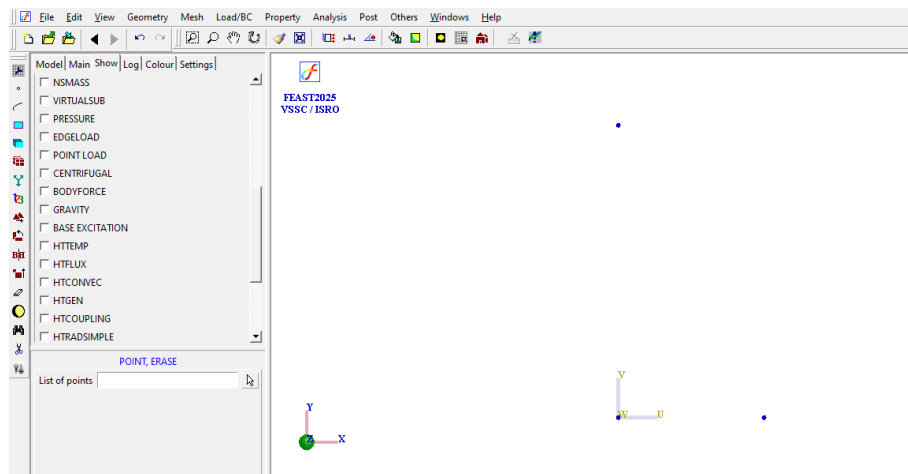
Menu : Geometry → Keypoint → Create → ADD



Parameters:

Similarly create key points at (0:4:0) and (2:0:0)

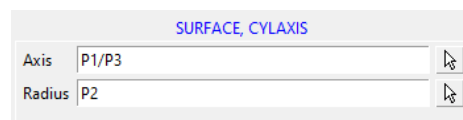
At the end of the operation your screen should look like this.



3. Create Cylindrical Surface

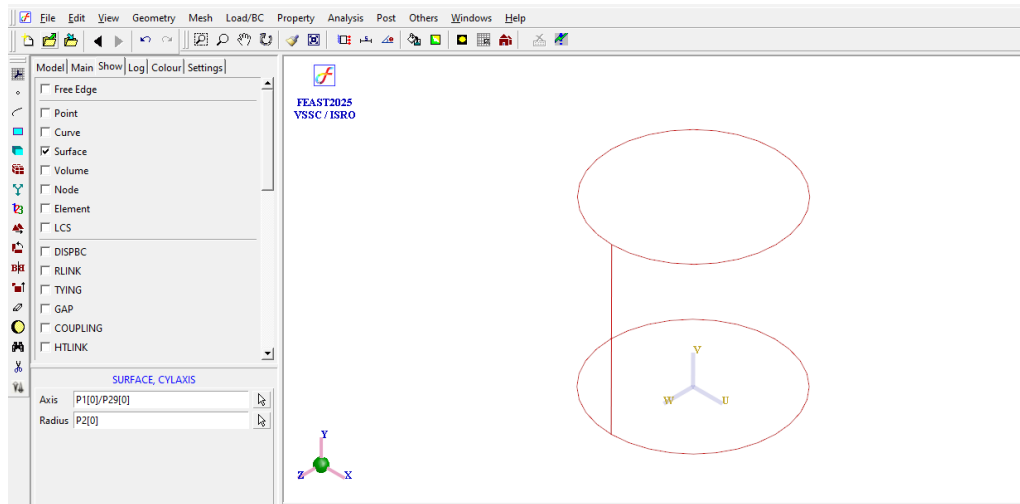
Command :SURFACE, CYLAXIS

Menu : Geometry → Surface → Create → Cylaxis



Parameters :

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

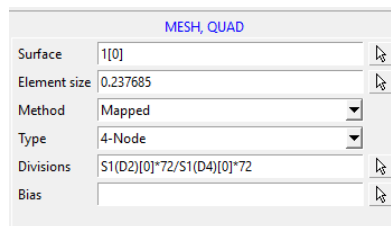


4. Generate mesh

Command : MESH, QUAD

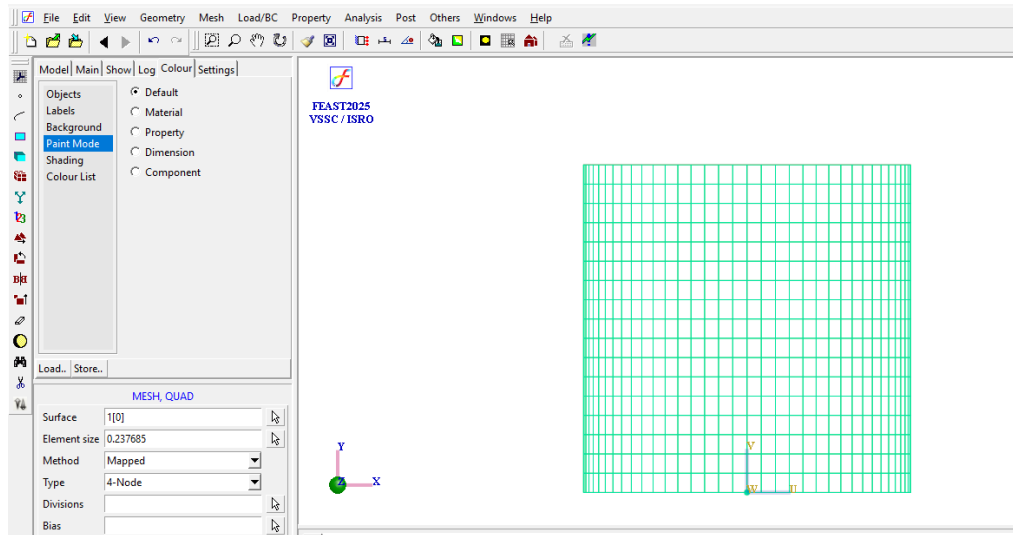
Menu : Mesh → MeshGen → Quad

Parameters :



Element size can be entered manually or by clicking two points on the geometry edge. Then click on the 'Divisions' command box, so that node divisions will be displayed on the geometry. The element subdivisions can be increased or decreased by left clicking or right clicking respectively.

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

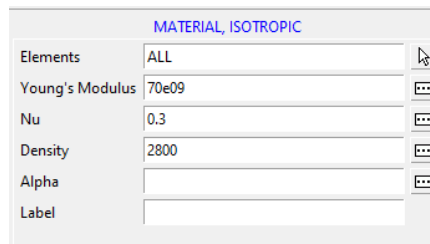


5. Apply Material Property

Command: MATERIAL, ISOTROPIC

Menu : Property → Material → Structural → Isotropic

Parameters:

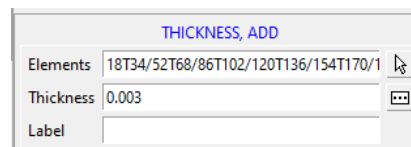


6. Apply Thickness

Command: THICKNESS, ADD

Menu : Property → Physical → Thickness

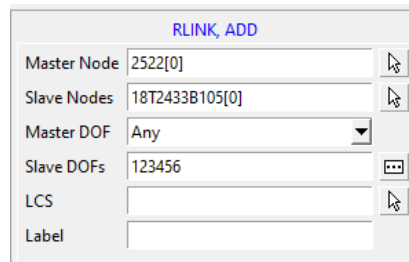
Parameters:



7. Create Rigid link

Command: RLINK, ADD

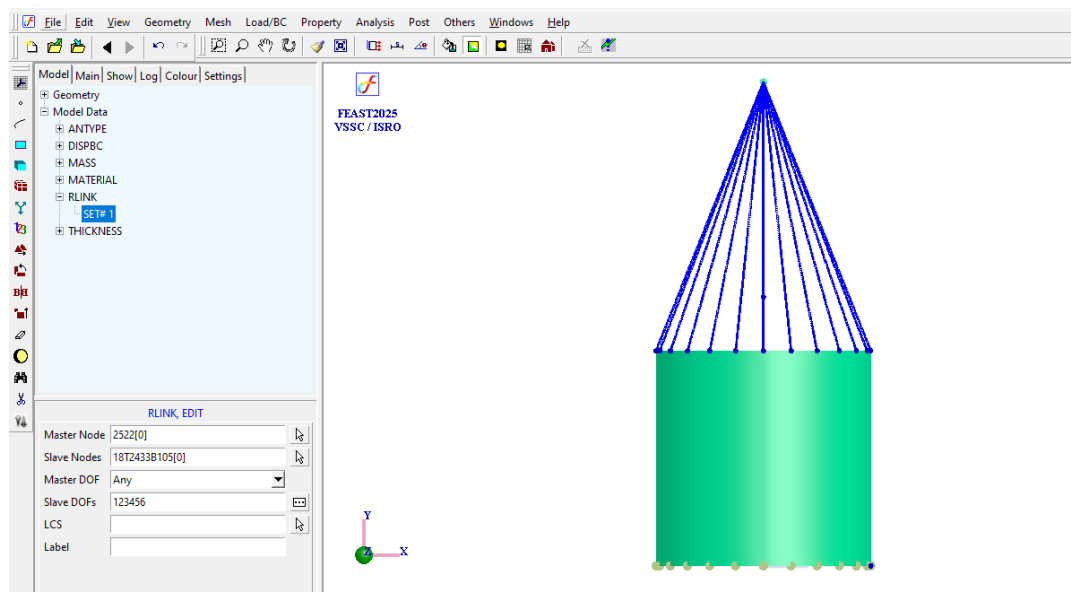
Menu : Load/BC → Structural → Rigid link



i. Parameters :

Create node at 0:9:0 by (Mesh →Node →Create→ Add) and it is taken as Master node remaining nodes at top edge is taken as slave nodes (Slave nodes should be 24. equally spaced between 72 nodes at bottom).

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



8. Apply Boundary Condition

Command: DISPBC, ADD

Menu : Load/BC→Structural→Displacement

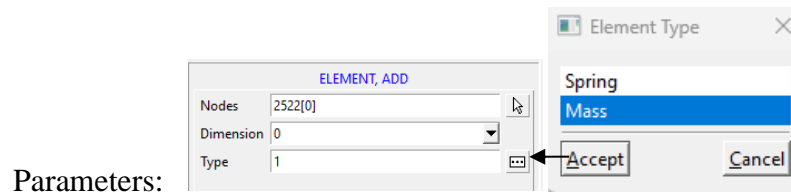


Parameters:

Apply DISPBC at bottom edge nodes corresponding to 24 rigid links

9. Set Mass Data

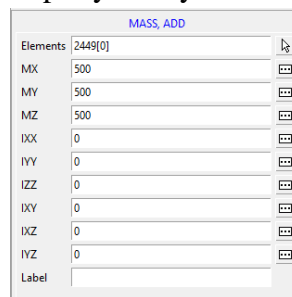
- i. Command: ELEMENT, ADD
Menu : Mesh → Element → Create → Add



Parameters:

Node id at location 0:9:0.

- ii. Command: MASS, ADD
Menu : Property → Physical → Mass



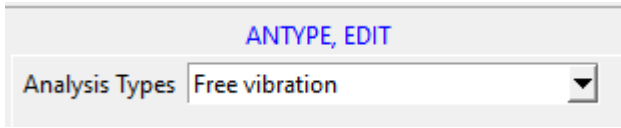
Parameters:

Mass of 500 is defined at top node id 0:9:0

10. Set Analysis Type

Command :ANTYPE,SET

Menu : Analysis → Analysis Type

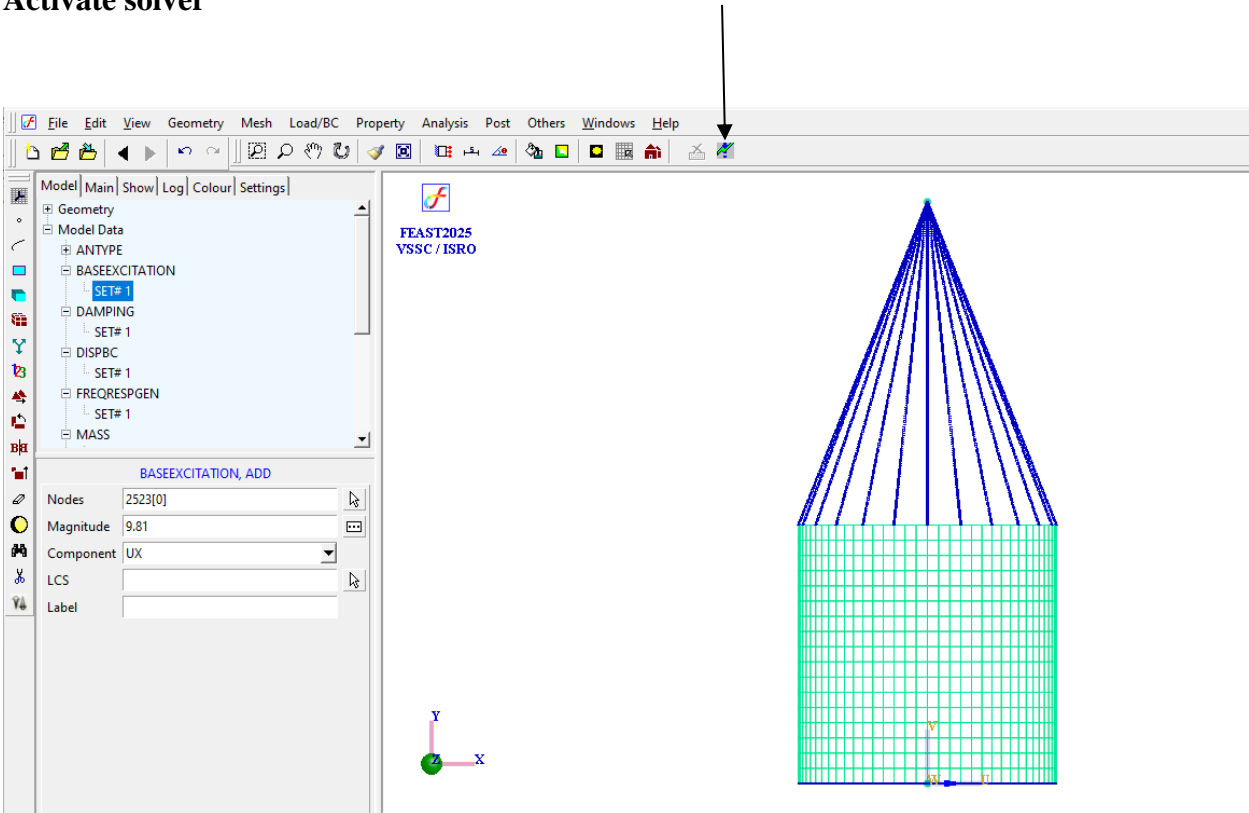


Parameters :

- 11. Save the project
Menu: File → Save

12. Activate solver

Click Here





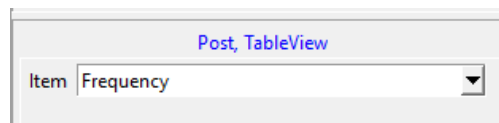
13. Perform Post Processing

I. To check natural frequencies

Command: POST, VIEWRESULT

Menu : Post → Table View

Parameters:

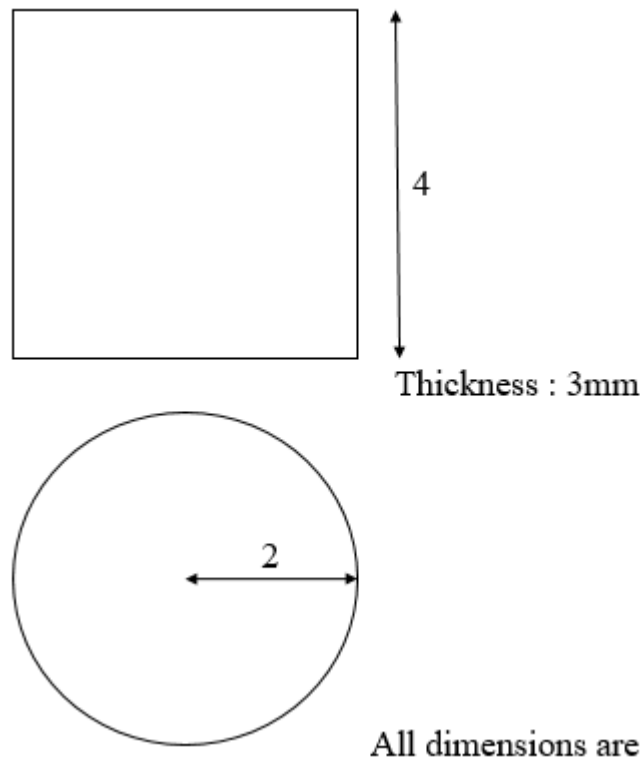


Frequency

Mode	Frequency(Hz)
1	23.4446
2	23.4446
3	23.8027
4	23.8027
5	25.203
6	25.203
7	25.7919
8	25.7919
9	28.1618
10	28.9817
11	28.9817
12	29.5078
13	29.5117
14	30.4276

Copy Close

FREQUENCY RESPONSE ANALYSIS OF A BASE EXCITED CYLINDER



An aluminium cylinder of radius 2m and height 4m is constrained at the bottom through 24 bolted joints. The thickness of the cylinder is 3mm. A mass of 500 kg is lumped at a height of 5m from the cylinder top surface. The mass is connected with the top of the cylinder through 24 bolted joints. Initially a free vibration analysis is performed to estimate the natural frequencies and mode shapes of the system.

A 1g steady state lateral base excitation is applied at the bottom of the cylinder and the response at the mass is estimated. This is done by performing a frequency response base excitation analysis. The lateral base excitation is applied to the cylinder through a heavy inertial mass 50 times greater than the system mass. The inertial mass is connected to the cylinder bottom portion through 24 number of rigid links. 2% critical damping is assumed for all the modes.

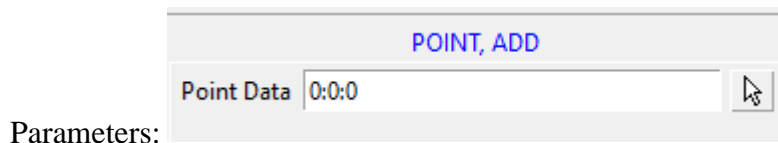


PROCEDURE

14. Create Key points

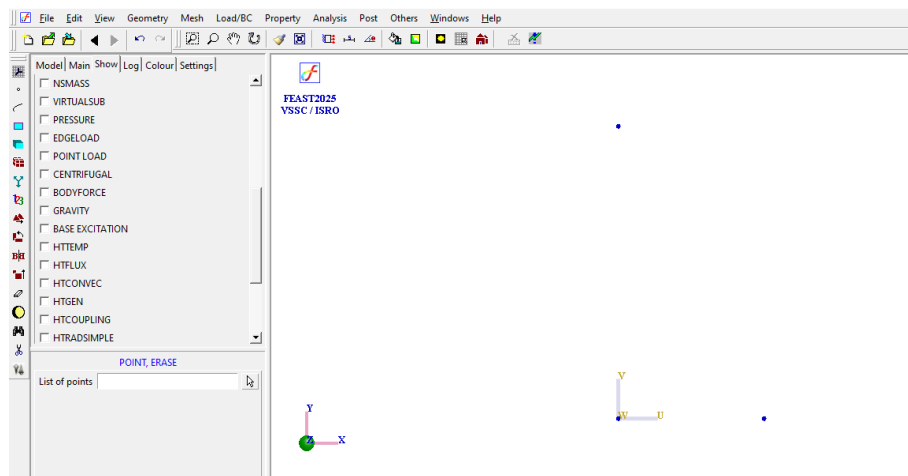
Command: POINT, ADD

Menu : Geometry → Keypoint → Create → ADD



Similarly create key points at (0:4:0) and (2:0:0)

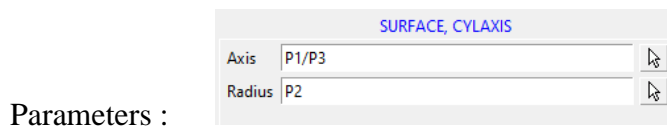
At the end of the operation your screen should look like this.



15. Create Cylindrical Surface

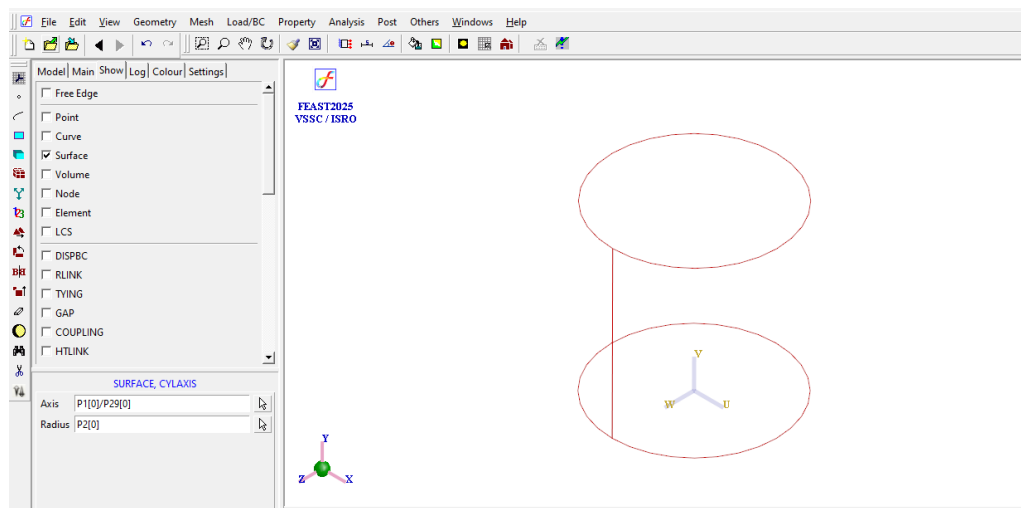
Command :SURFACE, CYLAXIS

Menu : Geometry → Surface → Create → Cylaxis





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

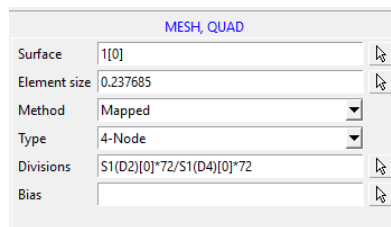


16. Generate mesh

Command : MESH, QUAD

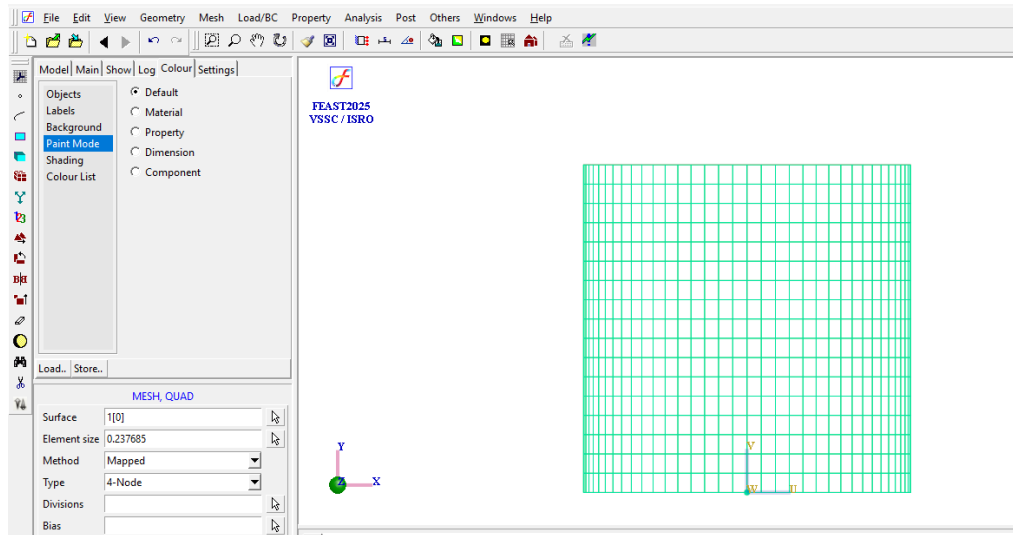
Menu : Mesh → MeshGen → Quad

Parameters :



Element size can be entered manually or by clicking two points on the geometry edge. Then click on the 'Divisions' command box, so that node divisions will be displayed on the geometry. The element subdivisions can be increased or decreased by left clicking or right clicking respectively.

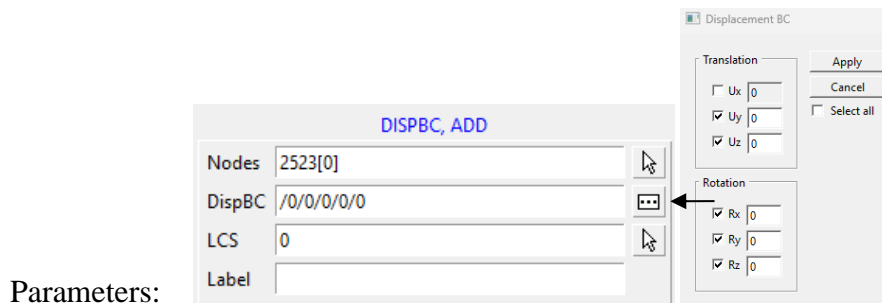
At the end of the operation your screen should look like this.



17. Apply Boundary Condition

Command: DISPBC, ADD

Menu : Load/BC→Structural→Displacement

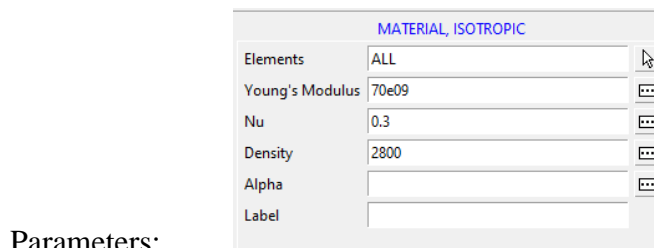


Apply DISPBC at location 0:0:0

18. Apply Material Property

Command: MATERIAL, ISOTROPIC

Menu : Property→Material →Structural →Isotropic





19. Apply Thickness

Command: THICKNESS, ADD

Menu : Property → Physical → Thickness

Parameters:

THICKNESS, ADD	
Elements	18T34/52T68/86T102/120T136/154T170/1
Thickness	0.003
Label	

20. Create Rigidlink

Command :RLINK,ADD

Menu : Load/BC → Structural → Rigidlink

ii. Parameters :

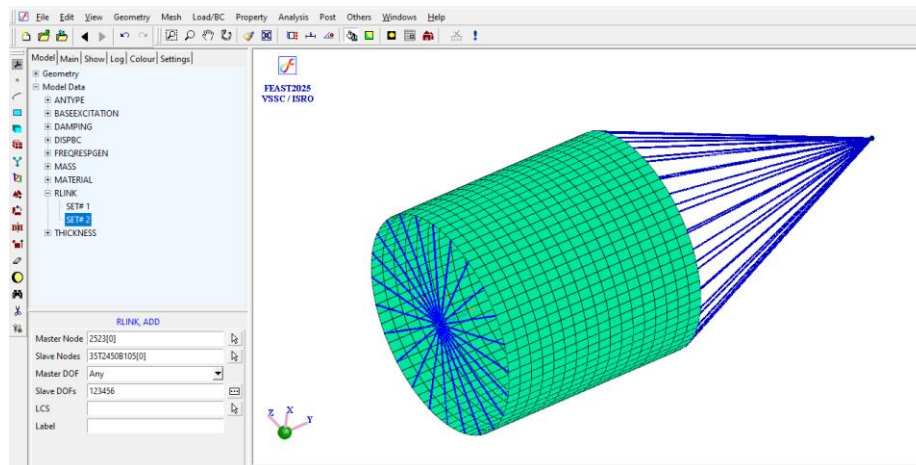
RLINK, ADD	
Master Node	2522[0]
Slave Nodes	18T2433B105[0]
Master DOF	Any
Slave DOFs	123456
LCS	
Label	

iii. Parameters :

RLINK, ADD	
Master Node	2523[0]
Slave Nodes	35T2450B105[0]
Master DOF	Any
Slave DOFs	123456
LCS	
Label	

Create node at 0:0:0 by (Mesh → Node → Create → Add) and it is taken as Master node remaining nodes at bottom edge is taken as slave nodes (Slave nodes should be 24. equally spaced between 72 nodes at bottom). Similarly create node at a height of 0:9:0 and create rigid links similar to the bottom edge.

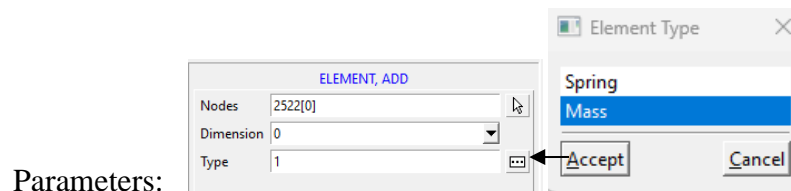
At the end of the operation your screen should look like this.



21. Set Mass Data

iii. Command: ELEMENT, ADD

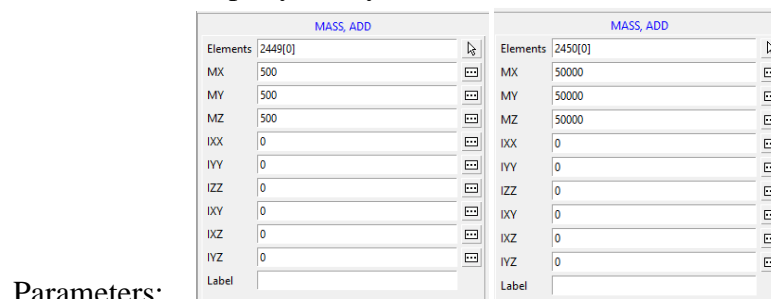
Menu : Mesh → Element → Create → Add



Node id at location 0:9:0. Similarly create mass element at 0:0:0

iv. Command: MASS, ADD

Menu : Property → Physical → Mass



Mass of 500 is defined at top node and mass of 50000 is defined at bottom mass location.

22. Set Base Excitation Data

Command: BASEEXCITATION, ADD

Menu : Load/BC → Structural → Base Excitation



BASEEXCITATION, ADD

Nodes: 2523[0]

Magnitude: 9.81

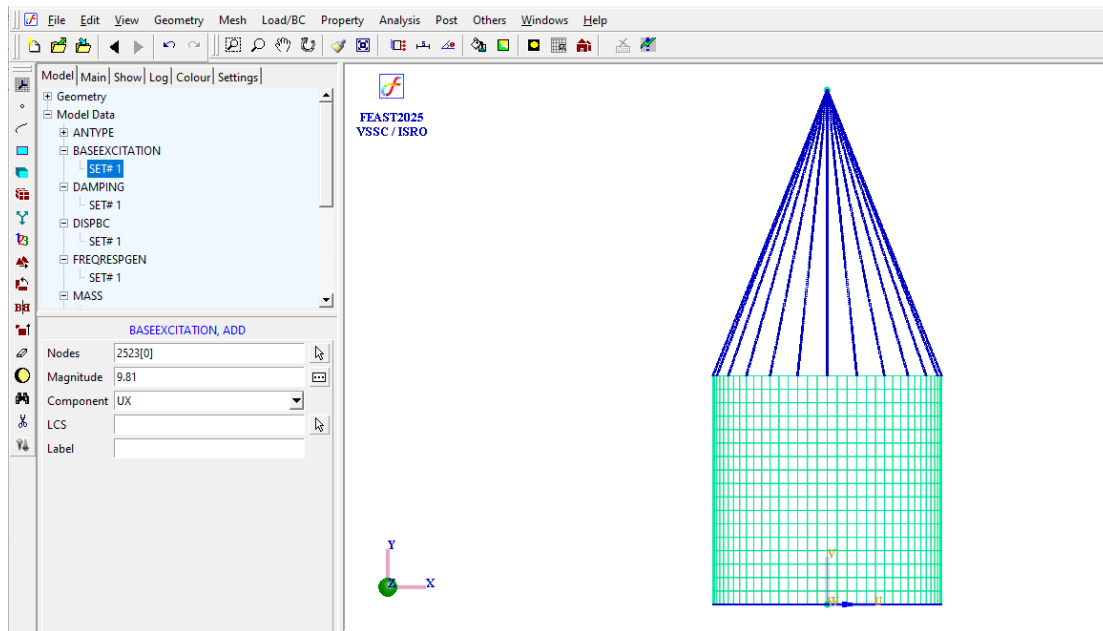
Component: UX

LCS:

Label:

Parameters:

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



23. Create Damping Data

Command: DAMPING, ADD

Menu : Analysis → Damping

DAMPING, ADD

Damping model: Modal Damping

Damping Factors: 5/0.02/100/0.02

Damping Factor

Frequency	Damping Factor
5	0.02
100	0.02

Buttons: Add Row, Insert Row, Delete Row, Copy, Paste, From file..., Cancel, Apply

Parameters:



24. Set Analysis Type

Command :ANTYPE,SET

Menu : Analysis→Analysis Type

Parameters :

ANTYPE,SET	
Analysis Type	Frequency Response ←

25. Set Frequency Response General Data

Command: FREQRESGEN, ADD

Menu : Analysis →Frequency Response →General

FREQRESGEN, ADD	
Response Extraction	Auto
Start Frequency	5
End Frequency	40
Finer Increment	0.1
Coarser Increment	1
Number of modes	20
Mass type	Lumped
Stress Output	NO
Nodes	ALL

Parameters:

Note :

In the node list the node IDs of the nodes at which the response to be extracted is specified

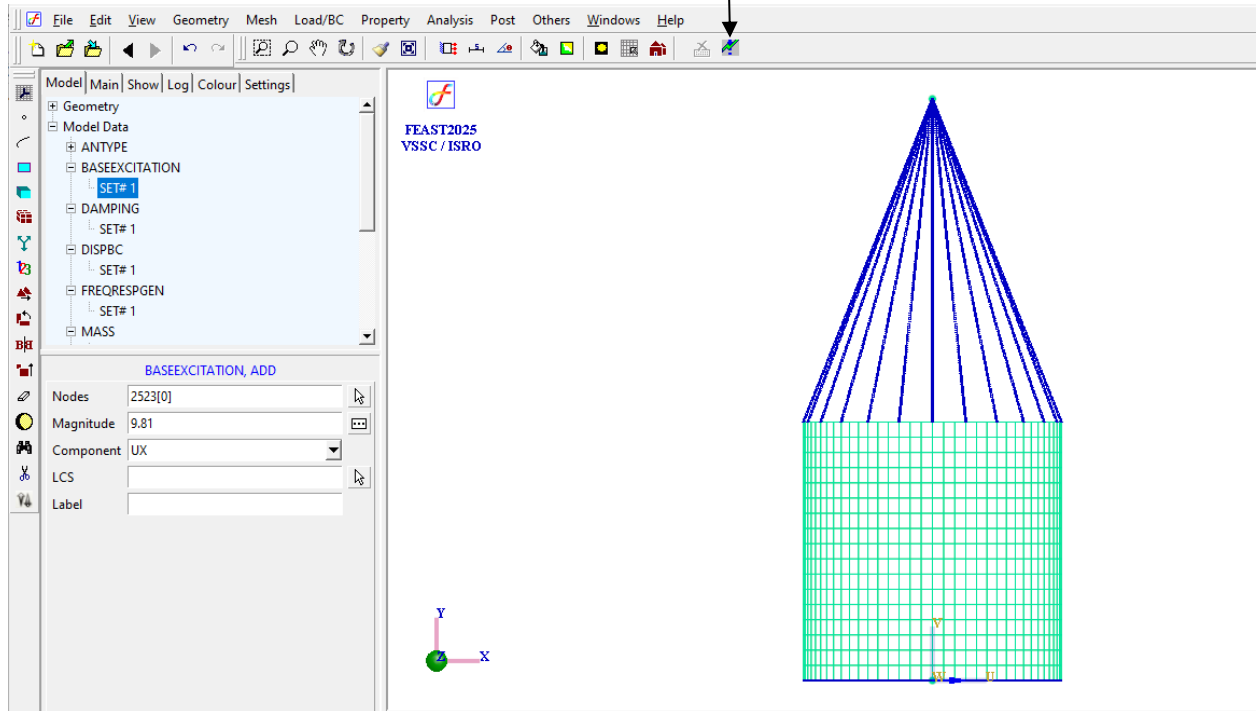
26. Save the project

Menu: File →Save



27. Activate solver

Click Here



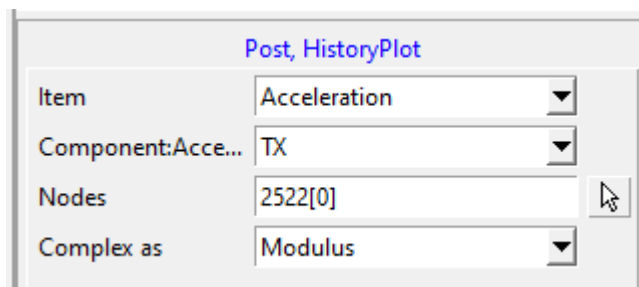
28. Perform Post Processing

I. Graph plots for displacement/ velocity/ acceleration

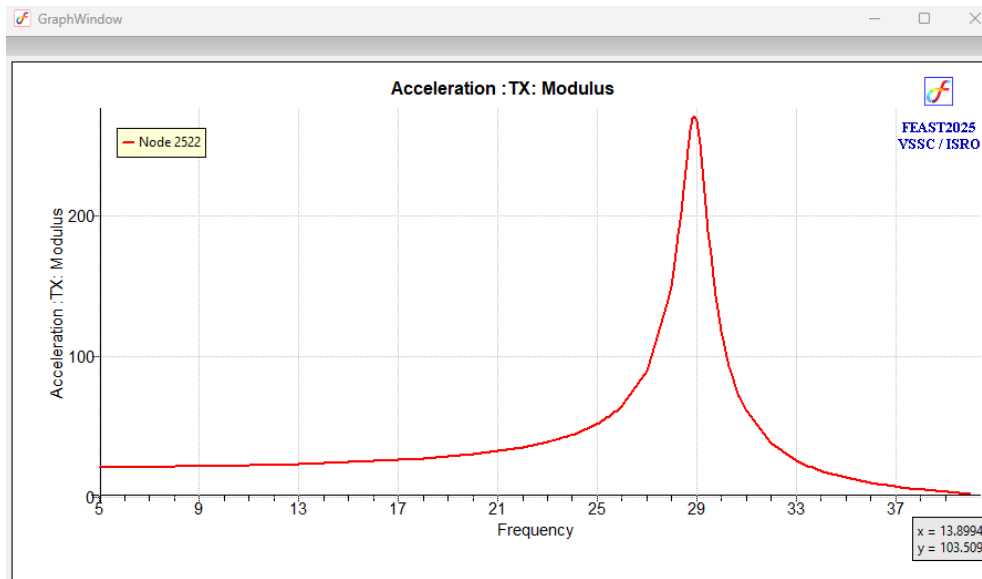
Command: POST, HISTORYPLOT

Menu : Post→History Plot

Parameters:



Select the top mass lumping node. At the end of the operation your screen should look like this.

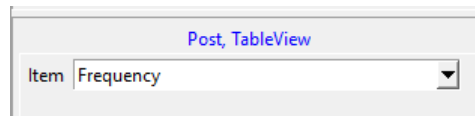


II. To check natural frequencies

Command: POST, VIEWRESULT

Menu : Post → Table View

Parameters:



Mode	Frequency(Hz)
1	23.427
2	23.427
3	23.7916
4	23.7916
5	25.1774
6	25.1774
7	25.7859
8	25.7859
9	28.1617
10	28.9362
11	28.9362
12	29.4729
13	29.473
14	30.4186

Buttons: Copy, Close