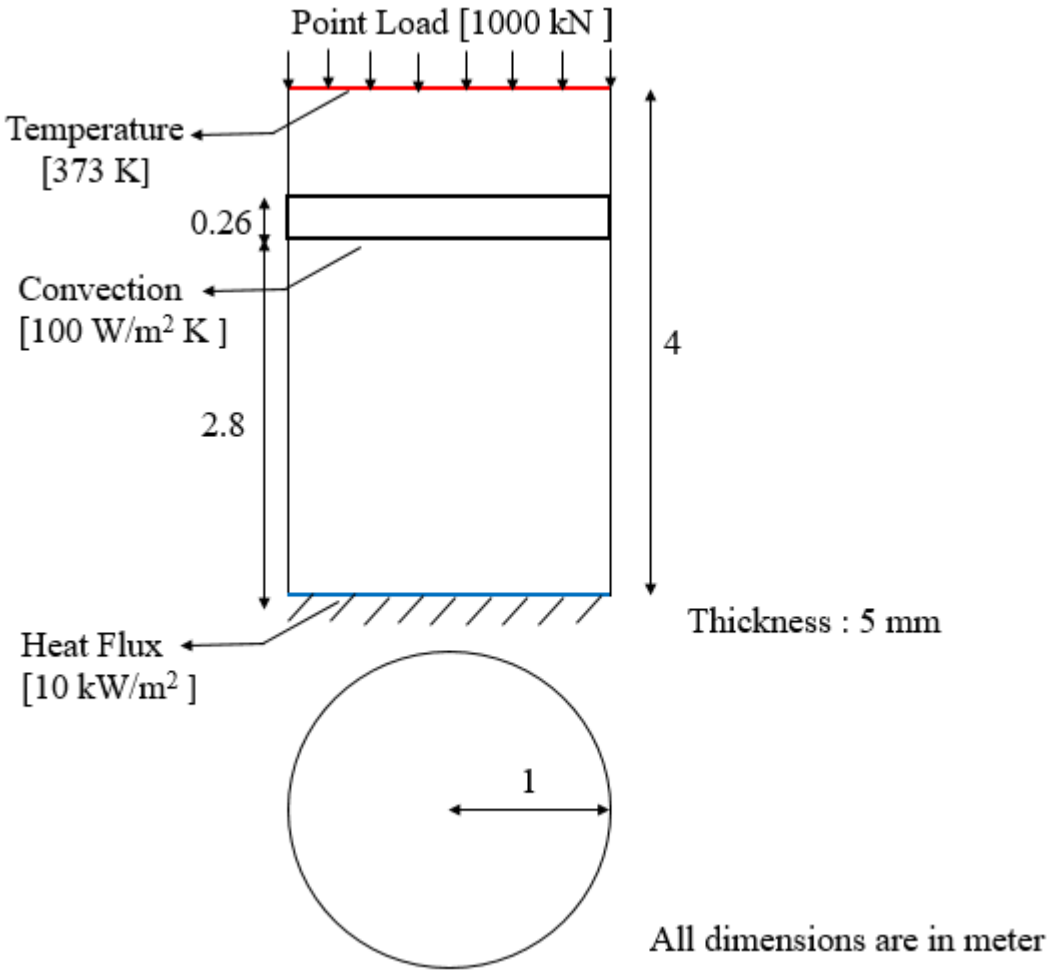
HEAT TRANSFER AND THERMOSTRUCTURAL ANALYSIS OF A CYLINDER



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 sorrounding which has a convective coefficient of 100 W/m^2 K at 300 K. A constant heat flux 10 kW/m^2 is acting at the bottom, whereas the top edge of the cylinder is maintained at 373 K.

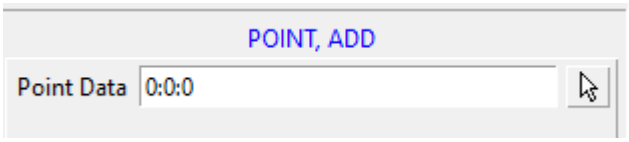
A structural load of 1000 kN is acting at the top edge of the cylinder and the bottom of the cylinder is assumed to be fixed. The objective of this analysis is to find the maximum stress due to the thermal environment along with the mechanical loads.

PROCEDURE

1. **Create Keypoints**

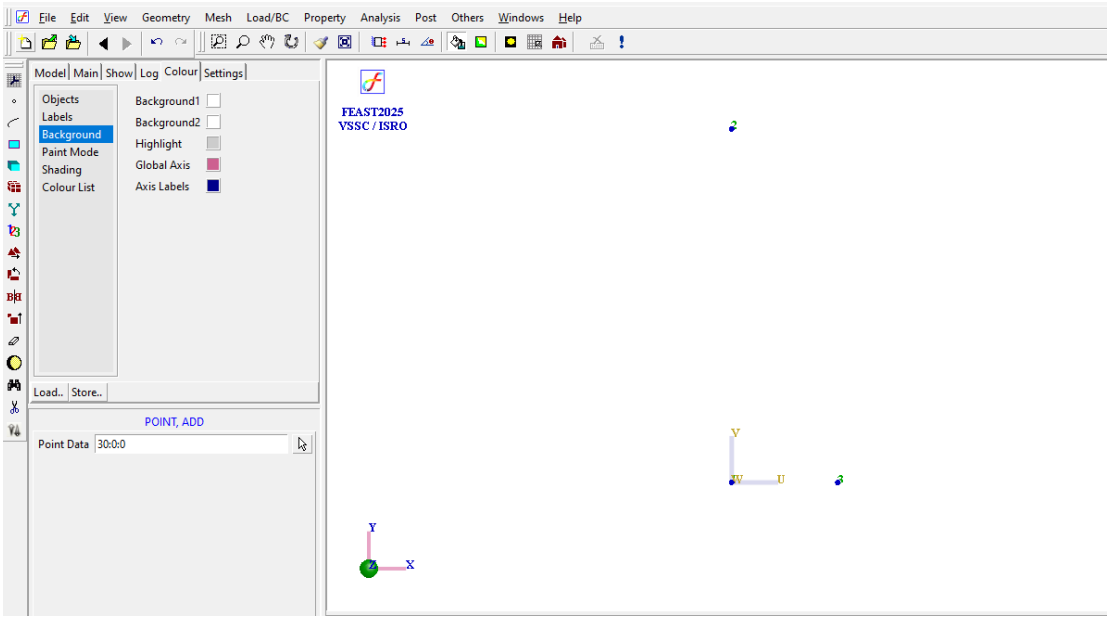
   Command: POINT, ADD

   Menu : Geometry →Keypoint→ Create → ADD

   Parameters: 

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

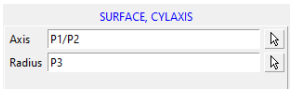
At the end of the operation your screen should look like this.



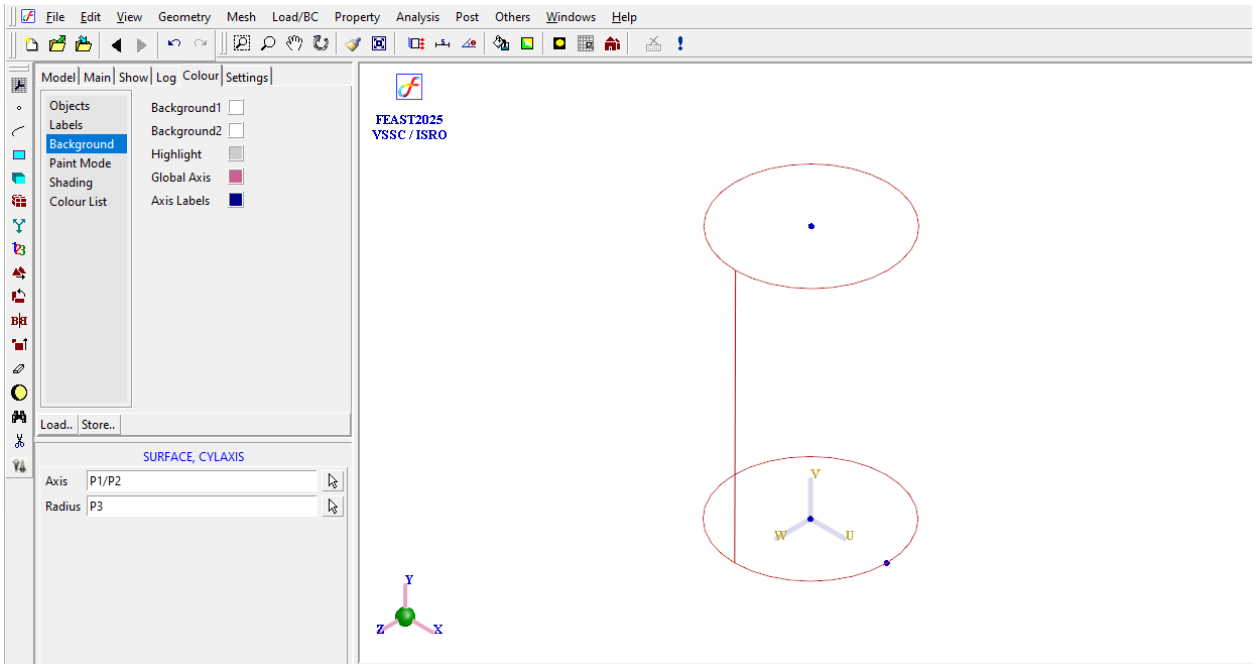
2. **Create Cylindrical Surface**

   Command: SURFACE, CYLAXIS

   Menu : Geometry →Surface→ Create →Cylaxis

Parameters: 

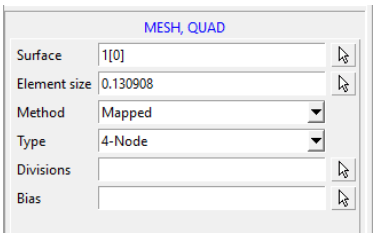
At the end of the operation your screen should look like this.



3. Generate mesh

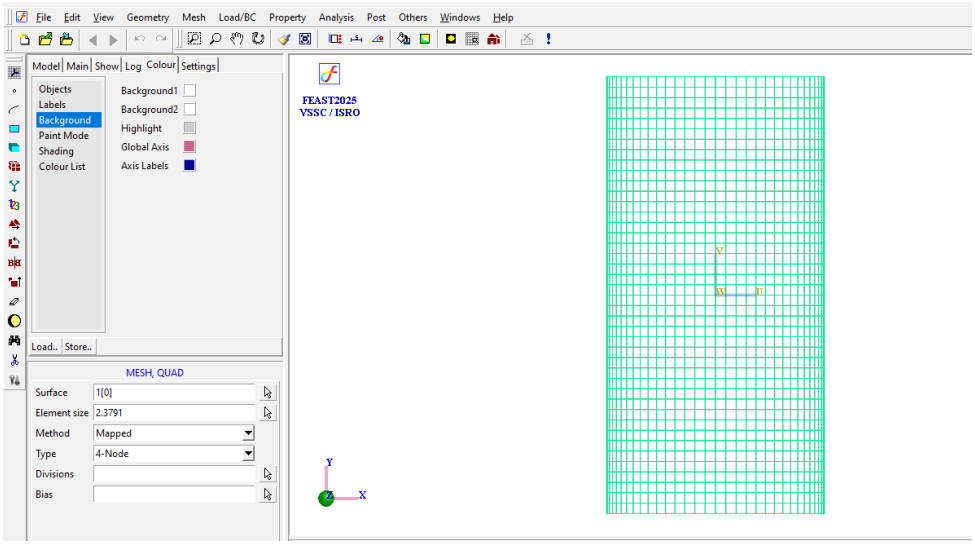
Command: MESH, QUAD

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.

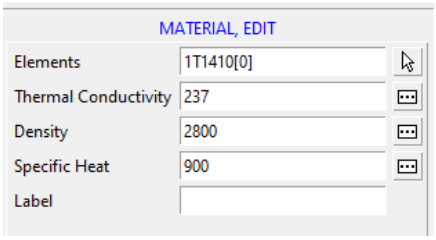


4. Apply Material Property

Command: MATERIAL, HTISOTROPIC

Menu : Property→Material →Thermal →Isotropic

Parameters:

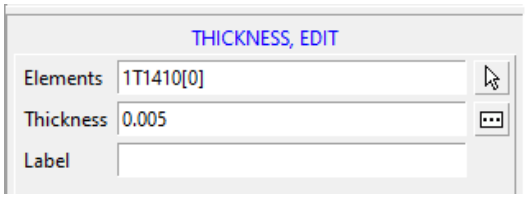


5. Apply Thickness

Command: THICKNESS, ADD

Menu : Property→Physical →Thickness

Parameters:

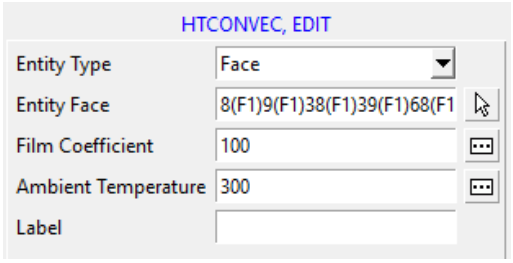


6. Apply Convective Boundary Condition

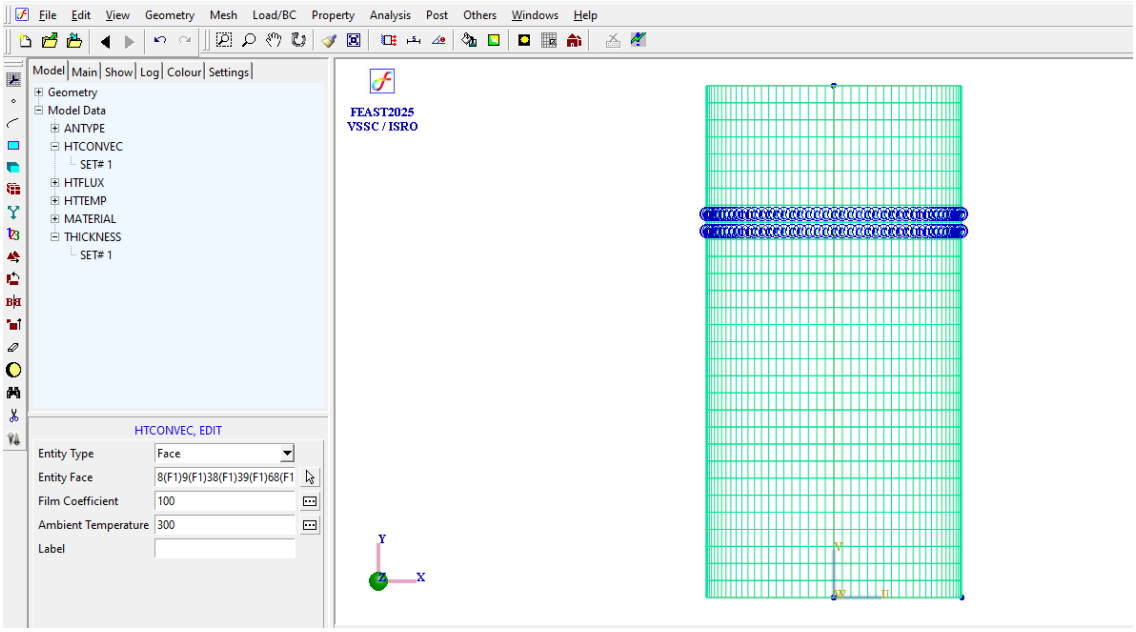
Command: HTCONVEC, ADD

Menu : Load/BC→Thermal→Convection

Parameters:



The selected face is look like below

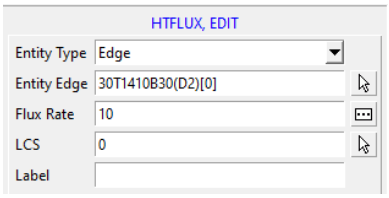


7. Apply Heat Flux Boundary Condition

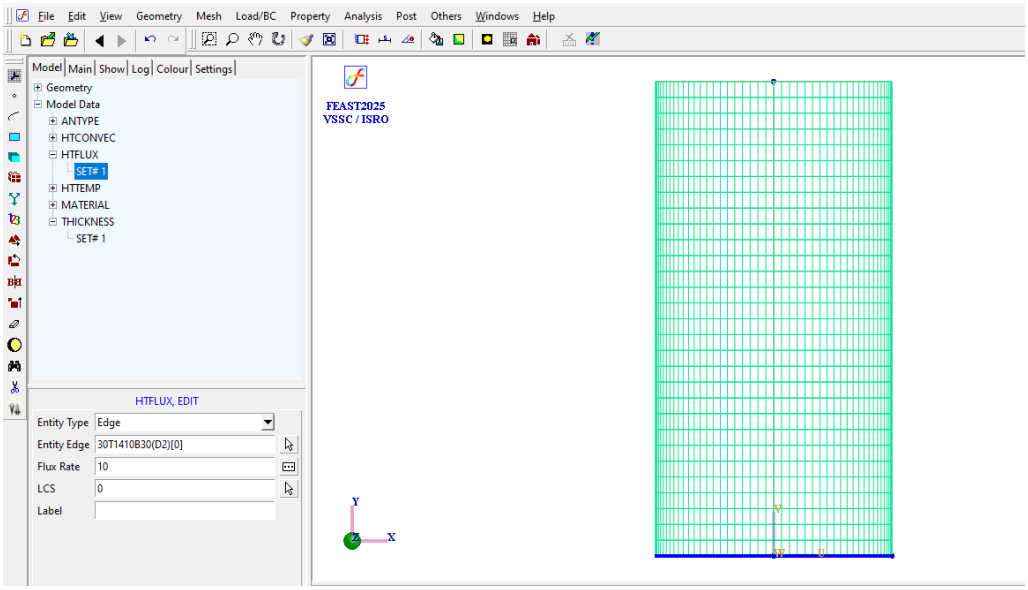
Command: HTFLUIX, ADD

Menu : Load/BC→Thermal→Heat flux

Parameters:



The defined edge is look like this,

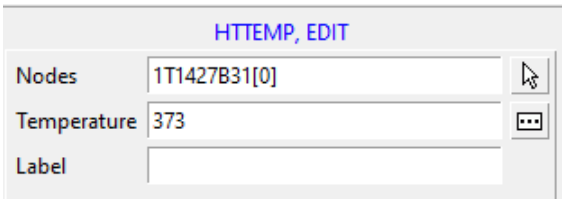


8. Apply Temperature Boundary Condition

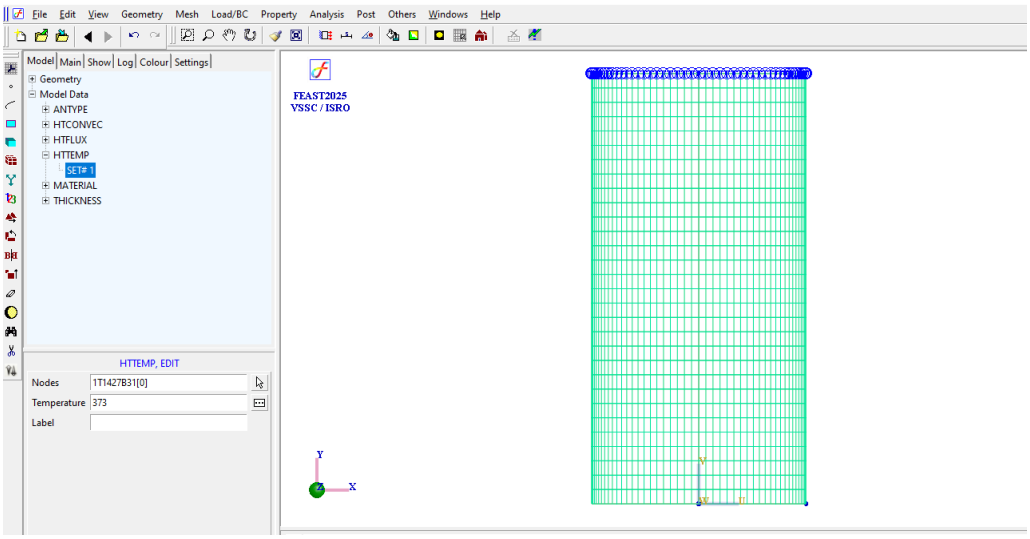
Command: HTTEMP, ADD

Menu : Load/BC→Thermal→Temperature

Parameters:



The defined nodes is look like this,

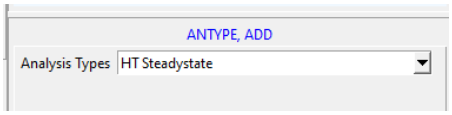


9. Set Analysis Type

Command: ANTYPE, SET

Menu : Analysis→Analysis Type

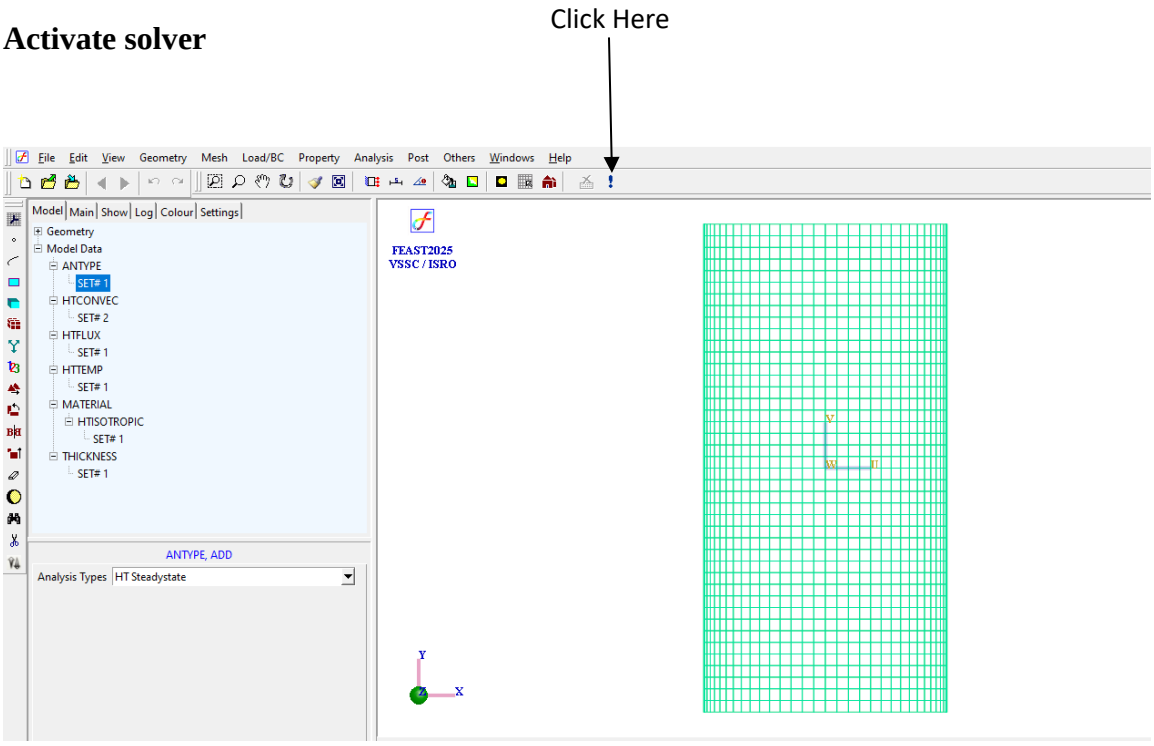
Parameters:



10. Save the project

Menu: File →Save

11. Activate solver



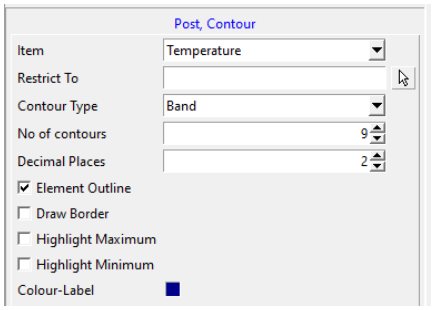
12. Perform Post Processing

i) Graph plots for displacement/ velocity/ acceleration

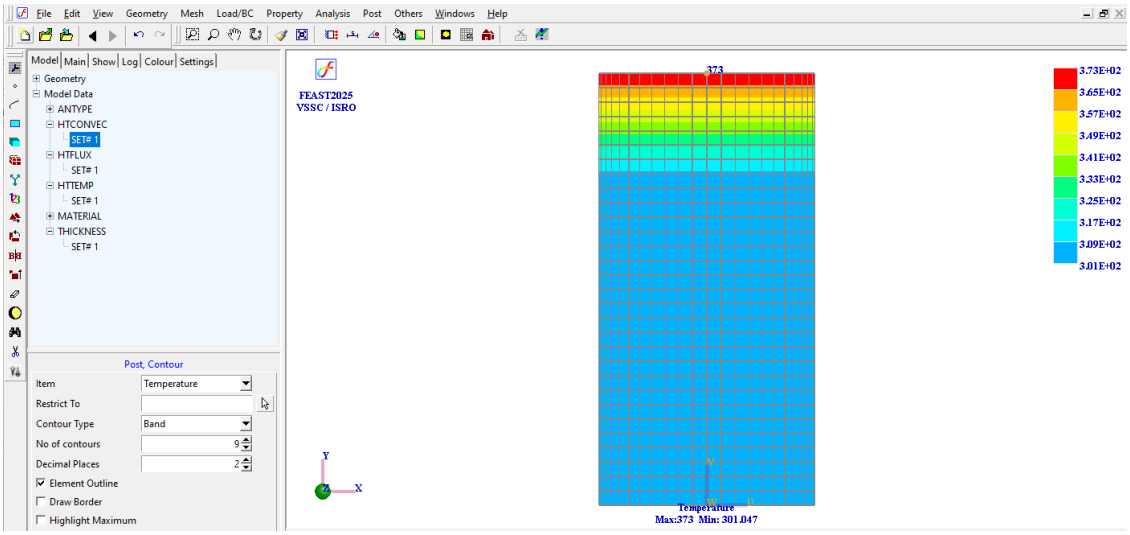
Command: POST, CONTOUR

Menu : Post→Contour

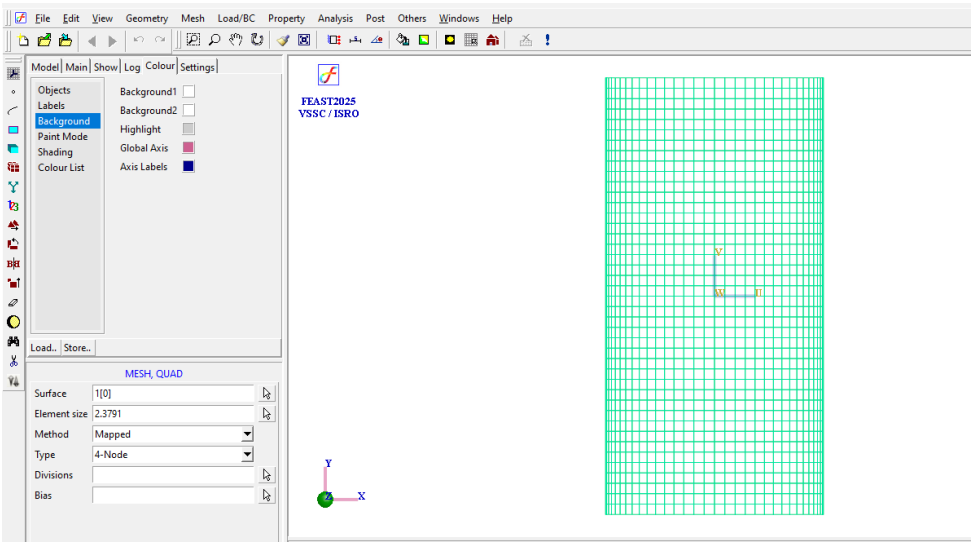
Parameters:



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.



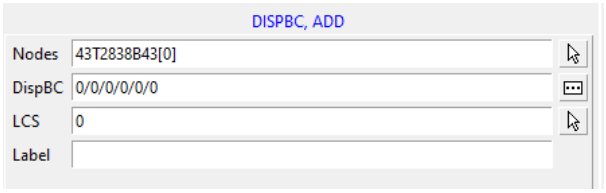
After meshing follow below steps,

1. **Apply Boundary Condition**

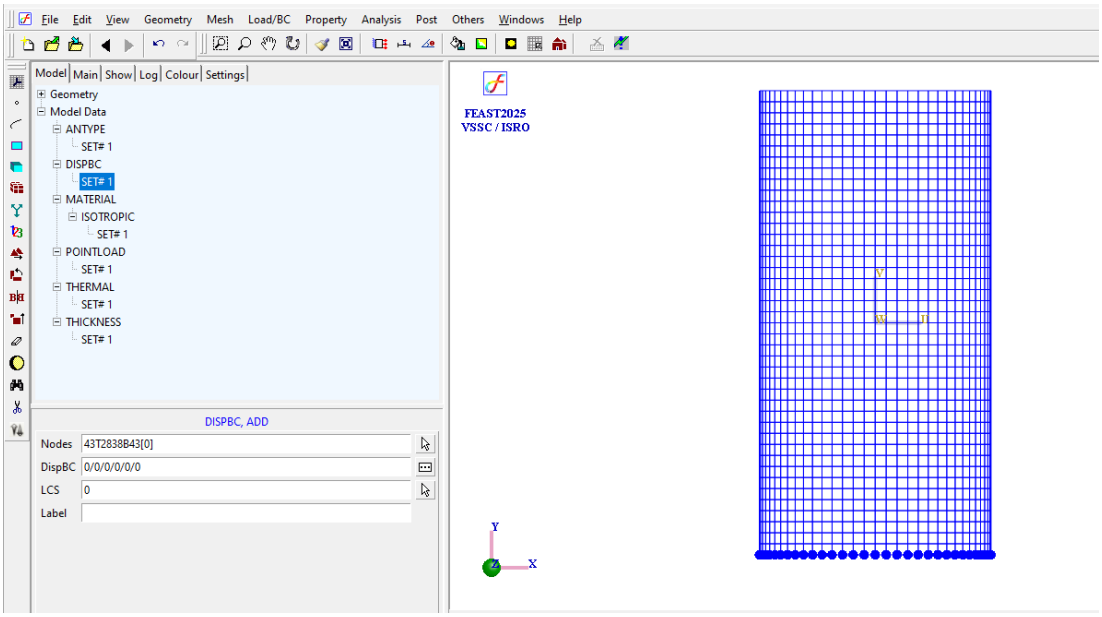
   Command: DISPBC, ADD

   Menu : Load/BC→Structural→Displacement

   Parameter:



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

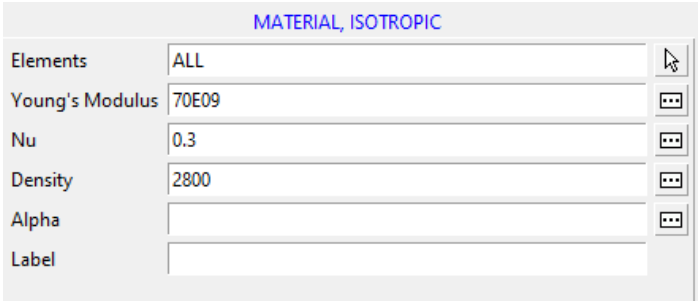


2. **Apply Material Property**

   Command: MATERIAL, ISOTROPIC

   Menu : Property→Material →Structural →Isotropic

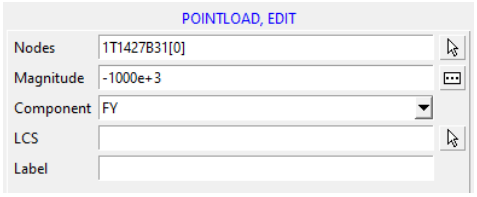
Parameter:



Thickness is same as applied in heat transfer analysis

3. **Apply Point load**
   Command: POINTLOAD, ADD
   Menu : LoadBC →Structural →Point load

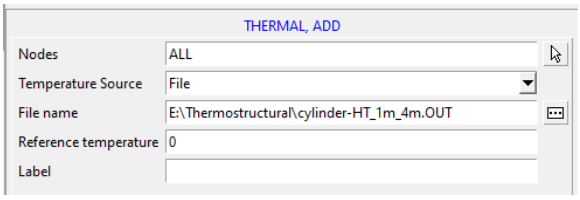
Parameter:



The node for point load is at the top end

4. **Apply Thermal load**
   Command: THERMAL, ADD
   Menu : LoadBC →Structural →Thermal load

Parameter:

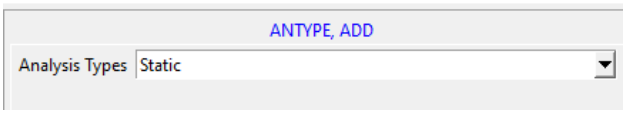


Select the *.out file from heat transfer analysis file location.

5. **Set Analysis type**
   Command: ANTYPE, SET

   Menu : Analysis→Analysis Type

Parameter :



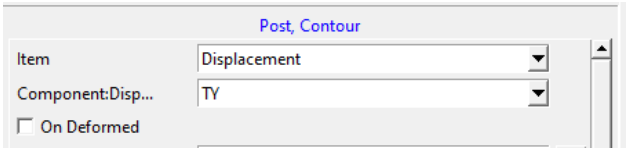
After that follow steps from 10 and 11 from heat transfer analysis

1. **Perform Post processing**

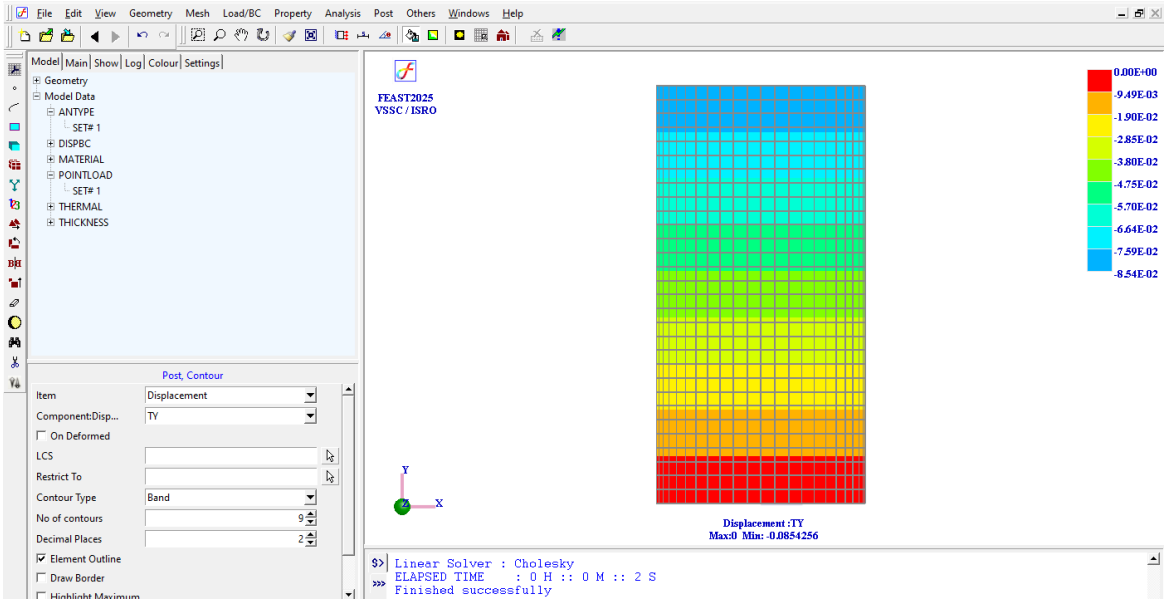
   Command: POST, CONTOUR

   Menu : Post→Contour

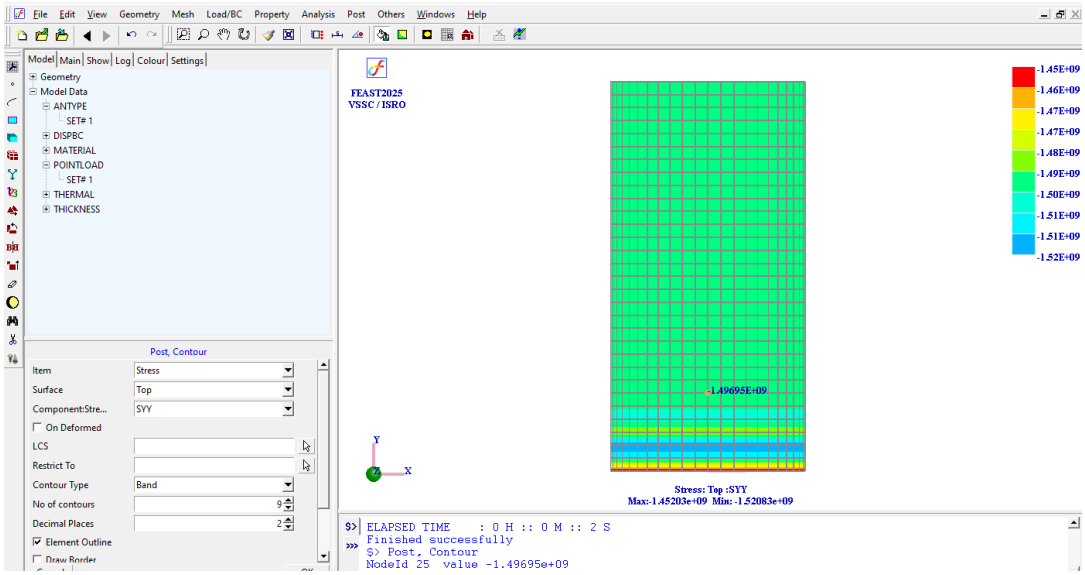
Parameter:



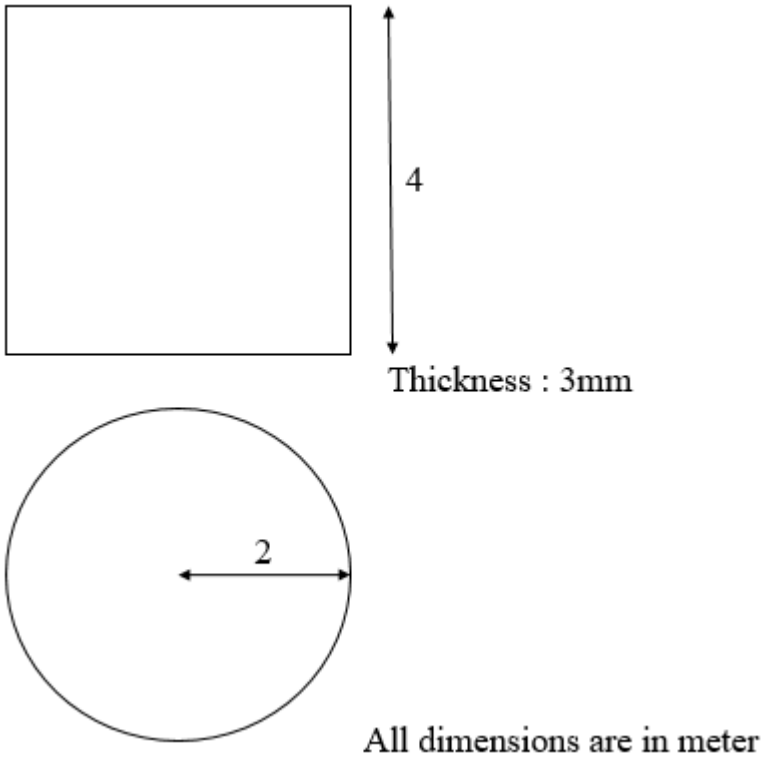
At the end of the operation your screen should look like this.



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.

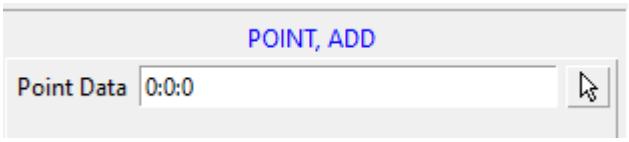
PROCEDURE

2. **Create Key points**

   Command: POINT, ADD

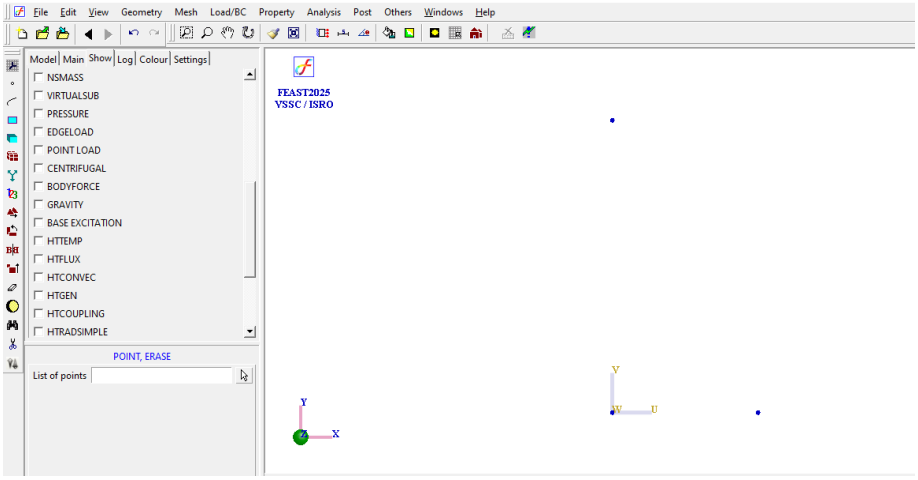
   Menu : Geometry →Keypoint→ Create → ADD

   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.

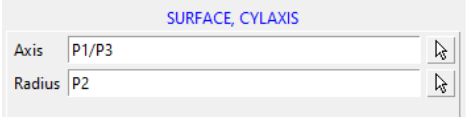


3. **Create Cylindrical Surface**

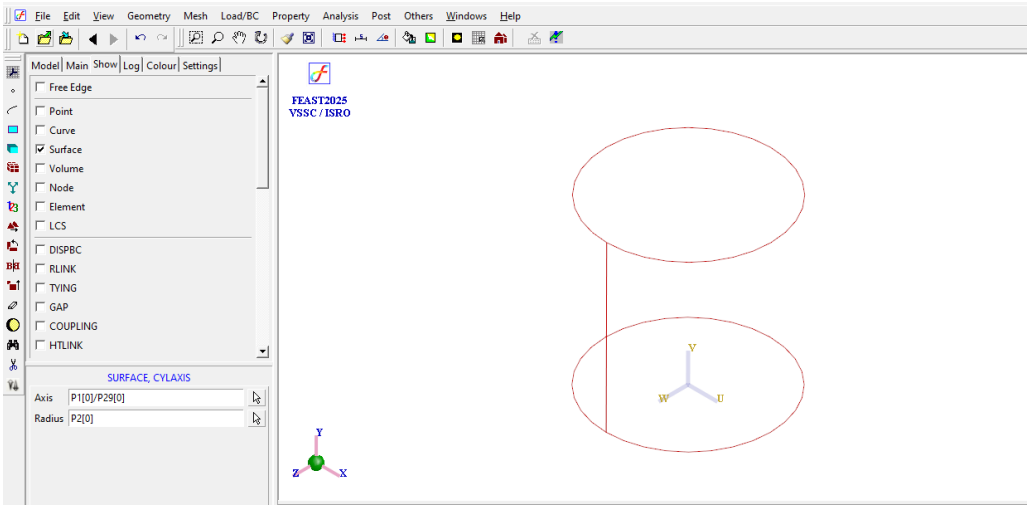
   Command :SURFACE, CYLAXIS

   Menu : Geometry →Surface→ Create →Cylaxis

   Parameters :



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

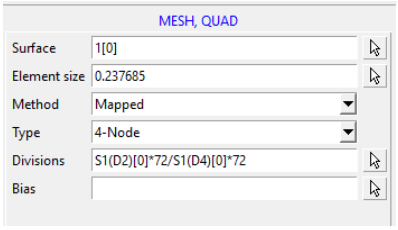


4. Generate mesh

Command :MESH, QUAD

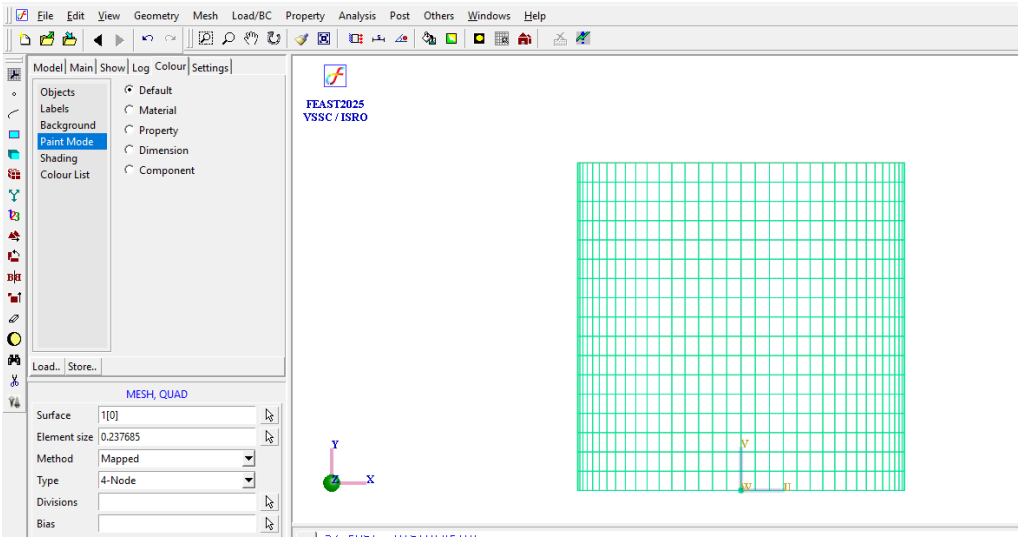
Menu : Mesh→MeshGen→Quad

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.

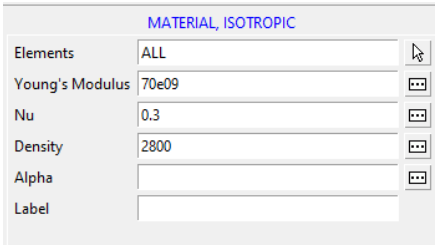


5. Apply Material Property

Command: MATERIAL, ISOTROPIC

Menu : Property→Material →Structural →Isotropic

Parameters:

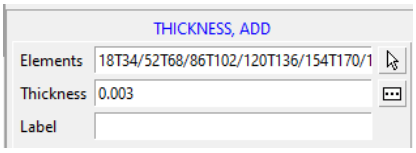


6. Apply Thickness

Command: THICKNESS, ADD

Menu : Property→Physical →Thickness

Parameters:

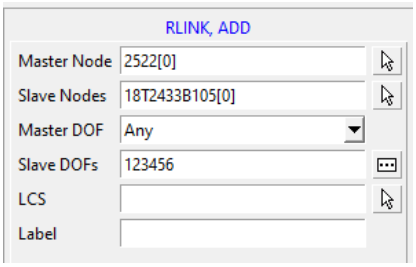


7. Create Rigid link

Command: RLINK, ADD

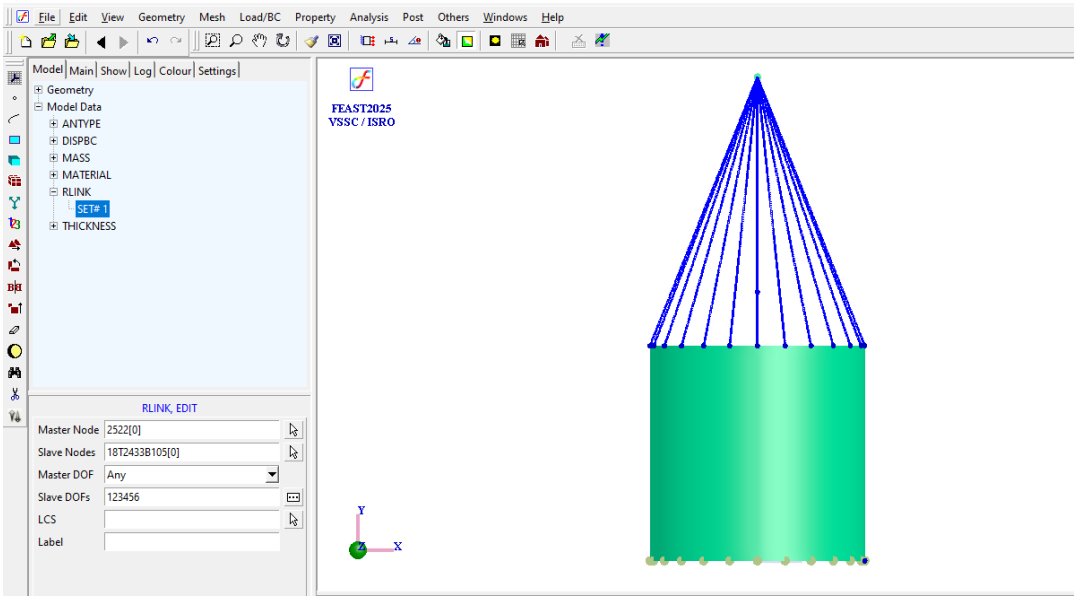
Menu : Load/BC→Structural→Rigid link

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:

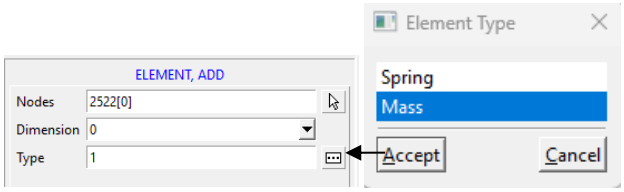
Apply DISPBC at bottom edge nodes corresponding to 24 rigid links

9. Set Mass Data

i. Command: ELEMENT, ADD

Menu : Mesh→ Element → Create → Add

Parameters:

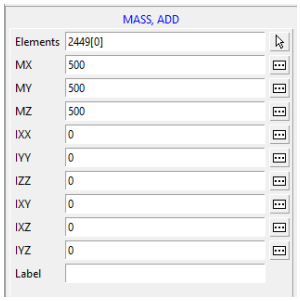


Node id at location 0:9:0.

ii. Command: MASS, ADD

Menu : Property→ Physical→ Mass

Parameters:



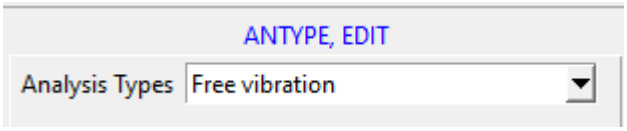
Mass of 500 is defined at top node id 0:9:0

10. Set Analysis Type

Command :ANTYPE,SET

Menu : Analysis→Analysis Type

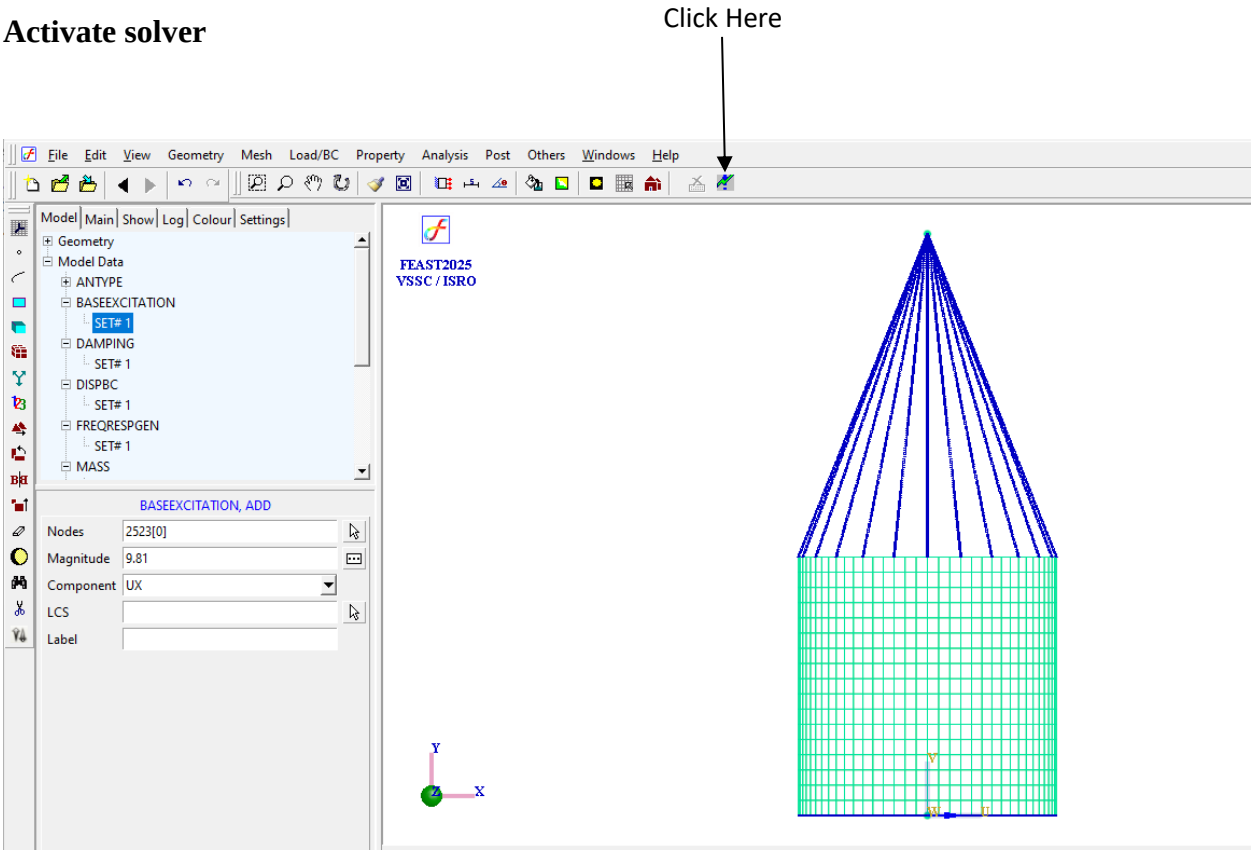
Parameters :



11. Save the project

Menu: File →Save

12. Activate solver



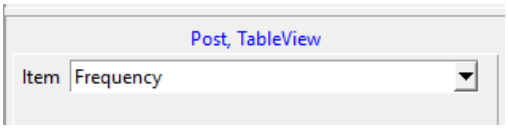
13. Perform Post Processing

I. To check natural frequencies

Command: POST, VIEWRESULT

Menu : Post →Table View

Parameters:

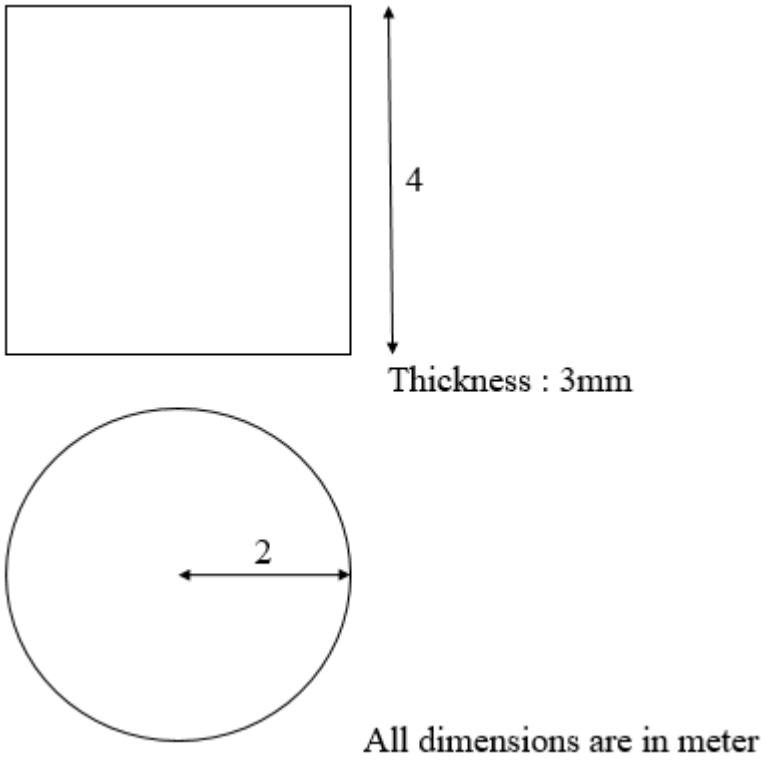


Frequency

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

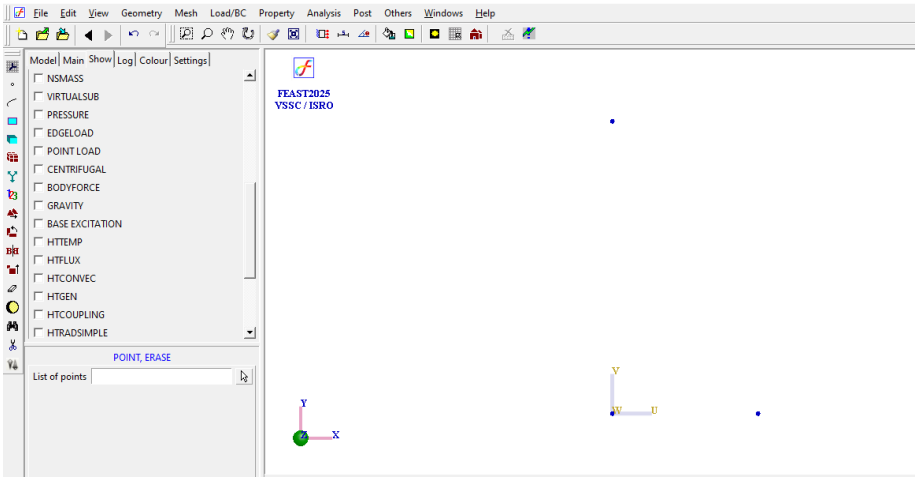
Command: POINT, ADD

Menu : Geometry →Keypoint→ Create → ADD

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.

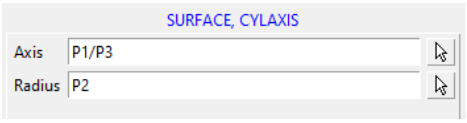


15. Create Cylindrical Surface

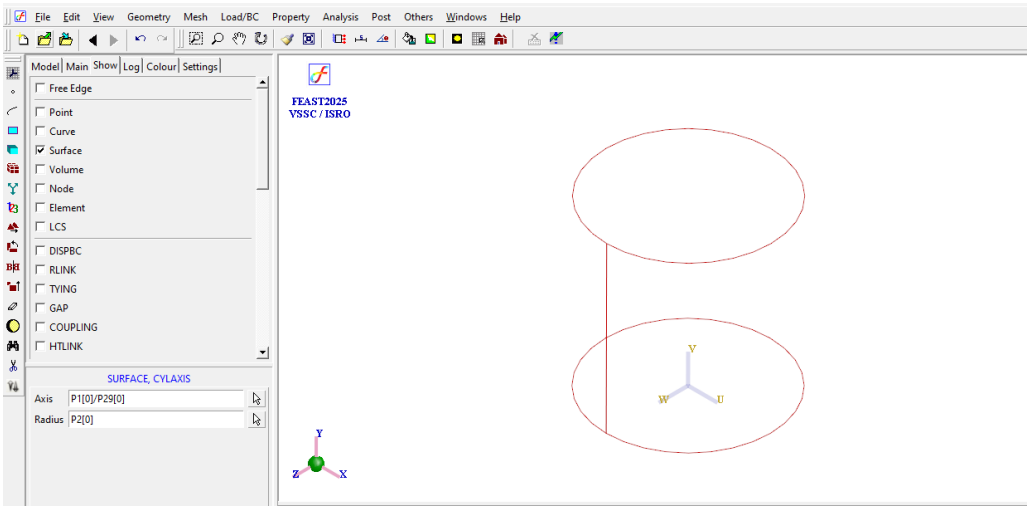
Command :SURFACE, CYLAXIS

Menu : Geometry →Surface→ Create →Cylaxis

Parameters :



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

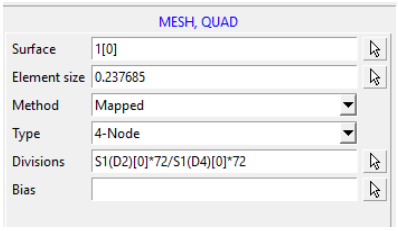


16. Generate mesh

Command :MESH, QUAD

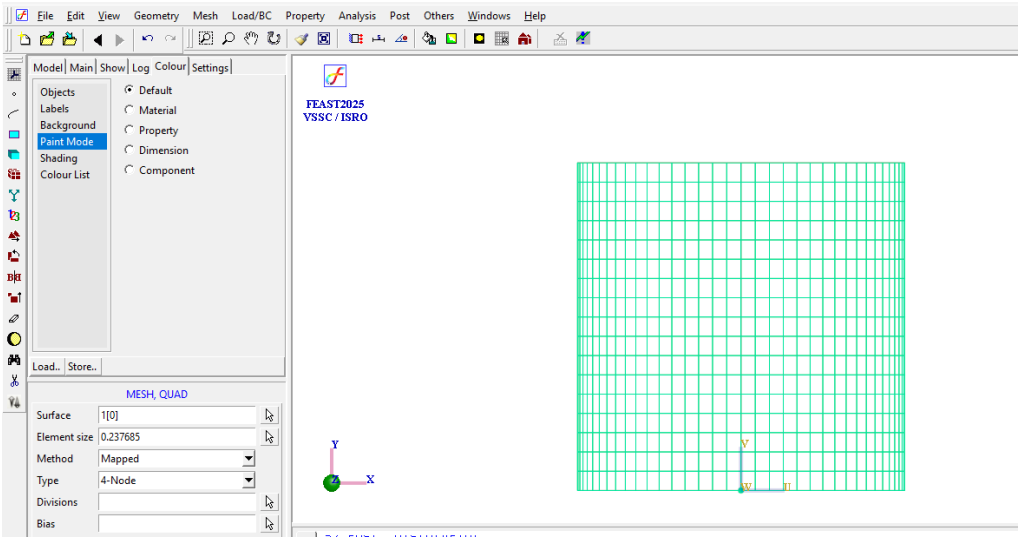
Menu : Mesh→MeshGen→Quad

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

Parameters:

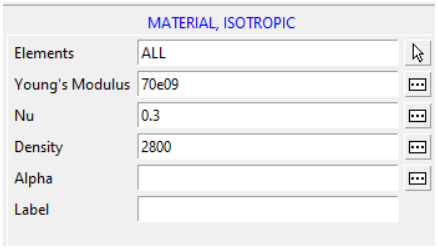
Apply DISPBC at location 0:0:0

18. Apply Material Property

Command: MATERIAL, ISOTROPIC

Menu : Property→Material →Structural →Isotropic

Parameters:

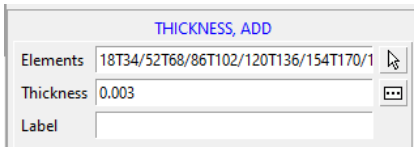


19. Apply Thickness

Command: THICKNESS, ADD

Menu : Property→Physical →Thickness

Parameters:

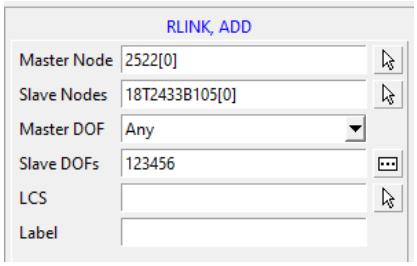


20. Create Rigidlink

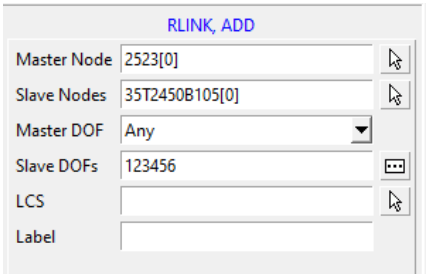
Command :RLINK,ADD

Menu : Load/BC→Structural→Rigidlink

ii. Parameters :

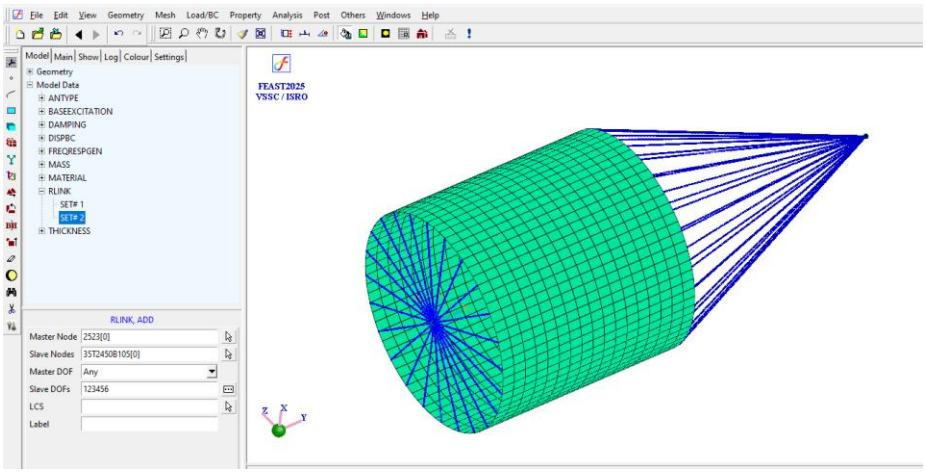


iii. Parameters :



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.

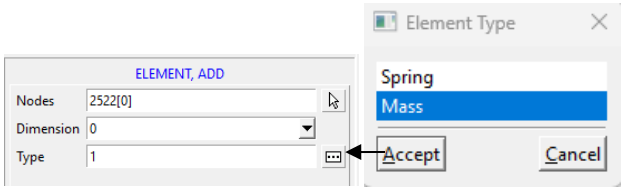


21. Set Mass Data

iii. Command: ELEMENT, ADD

Menu : Mesh→ Element → Create → Add

Parameters:

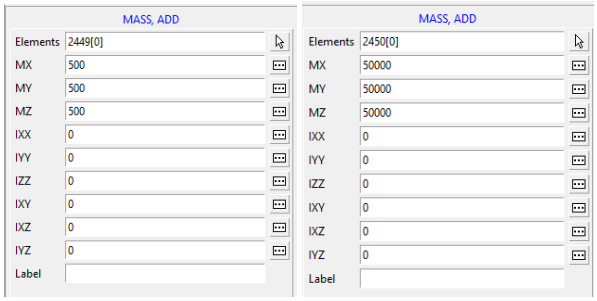


Node id at location 0:9:0. Similarly create mass element at 0:0:0

iv. Command: MASS, ADD

Menu : Property→ Physical→ Mass

Parameters:



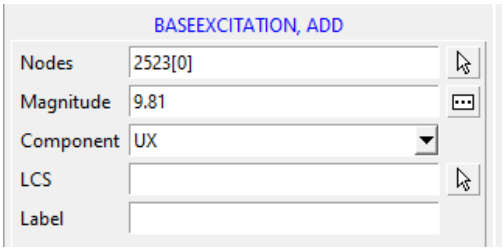
Mass of 500 is defined at top node and mass of 50000 is defined at bottom mass location.

22. Set Base Excitation Data

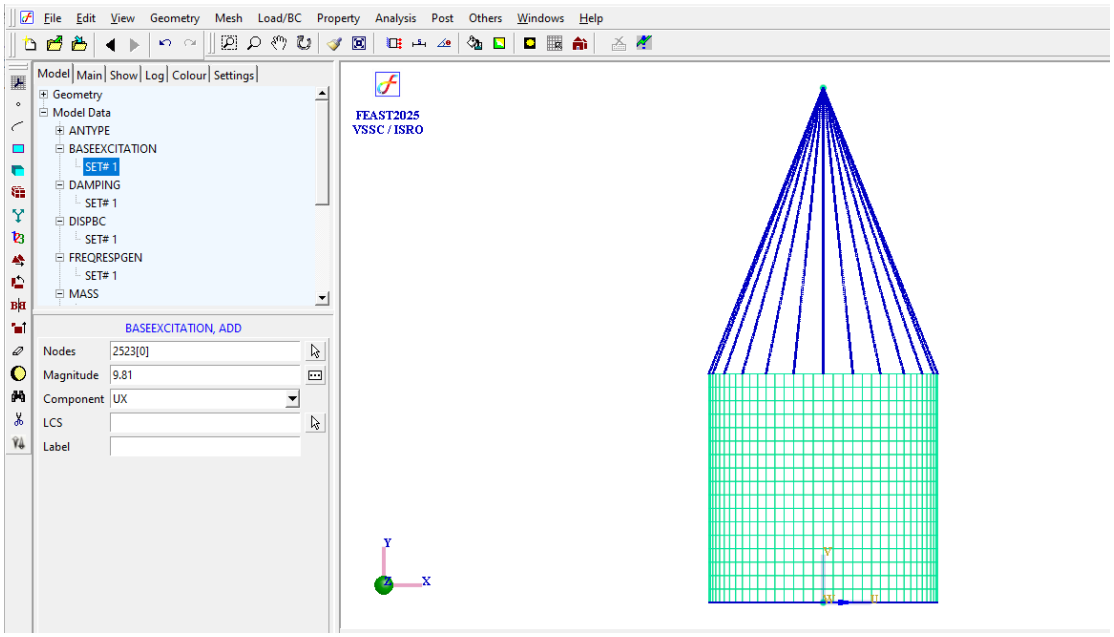
Command: BASEEXCITATION, ADD

Menu : Load/BC→Structural→Base Excitation

Parameters:



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

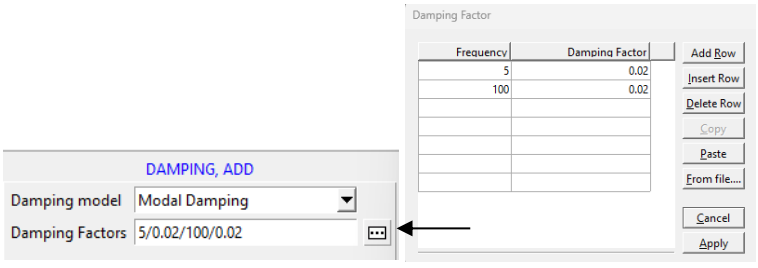


23. Create Damping Data

Command: DAMPING, ADD

Menu : Analysis →Damping

Parameters:

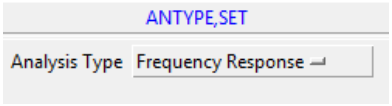


24. Set Analysis Type

Command :ANTYPE,SET

Menu : Analysis→Analysis Type

Parameters :

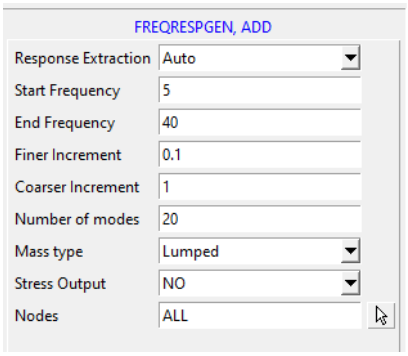


25. Set Frequency Response General Data

Command: FREQRESGEN, ADD

Menu : Analysis →Frequency Response →General

Parameters:



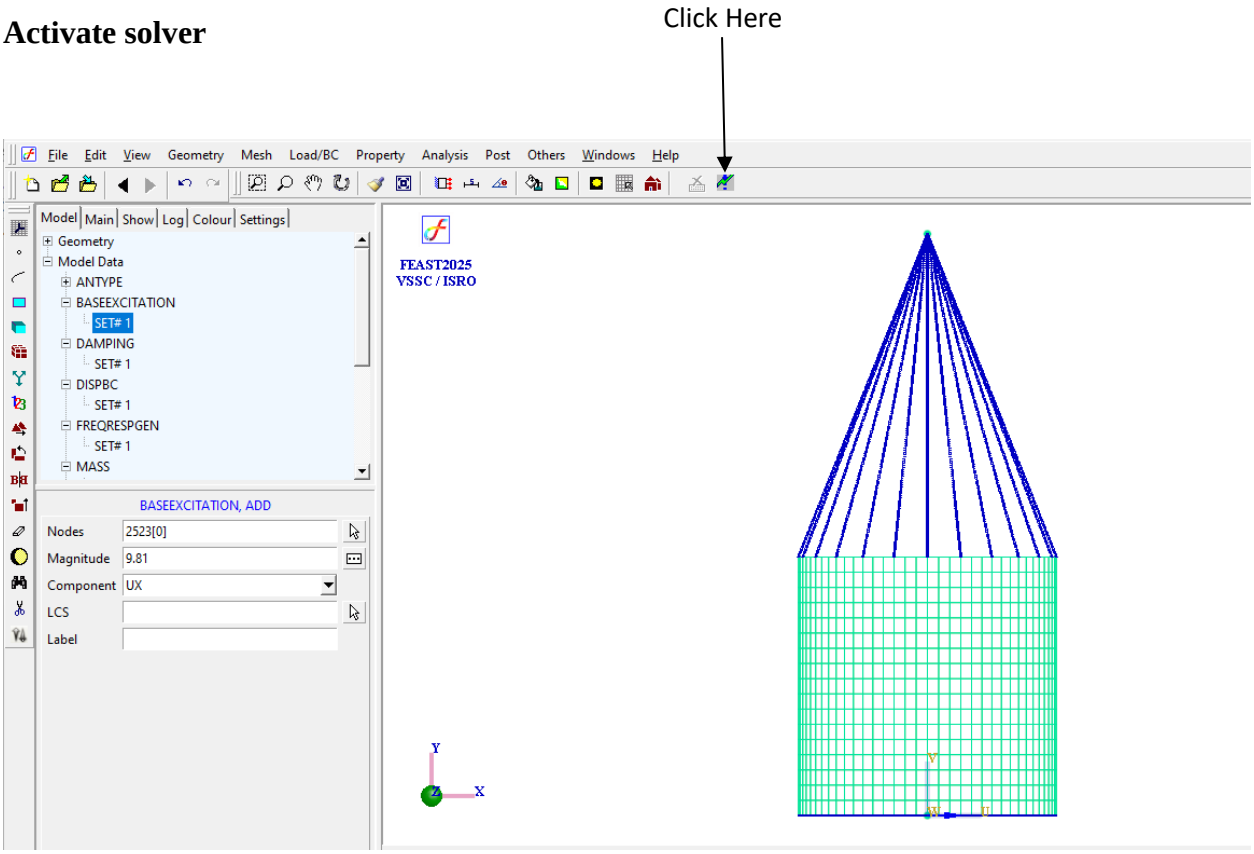
Note :

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



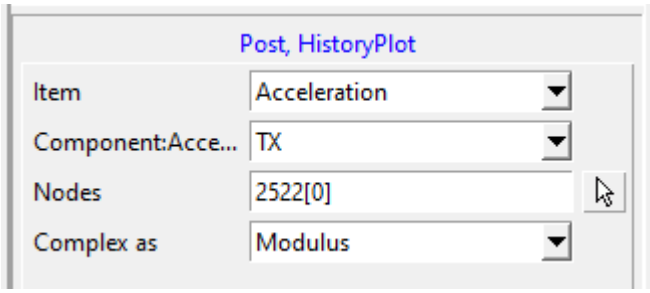
28. Perform Post Processing

I. Graph plots for displacement/ velocity/ acceleration

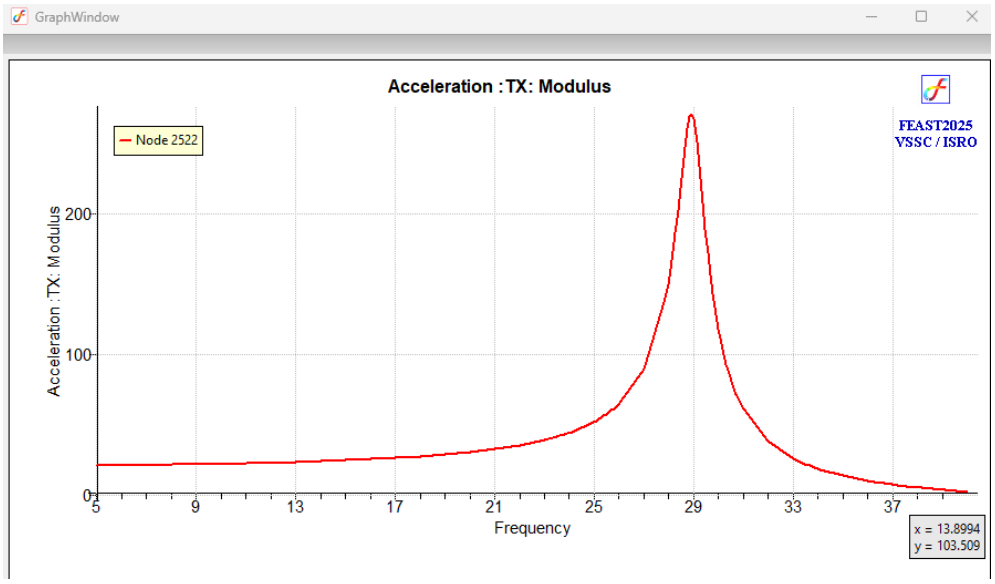
Command: POST, HISTORYPLOT

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: