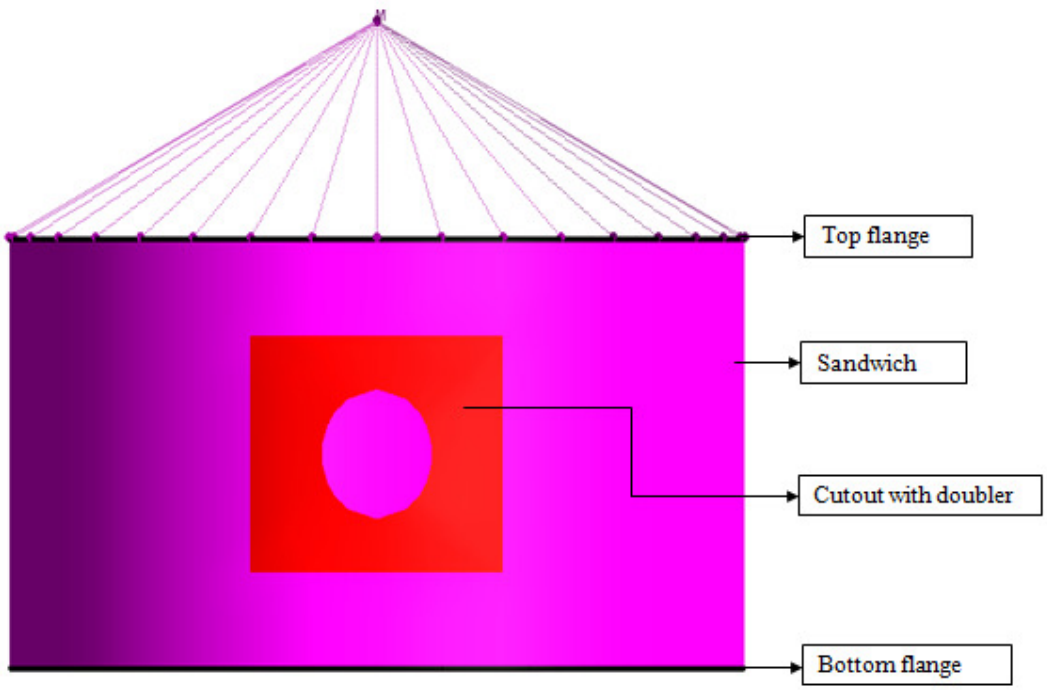
Transient response analysis of honeycomb core cylinder

1. Transient response analysis of honeycomb core cylinder



Cylinder has 2*m* of radius and height, it consist of top and bottom aluminum flanges and sandwich structure. Both flanges are 0.02*m* height and 5mm thickness, sandwich structure comprises of aluminum facet, core and aluminum facet. Aluminum facet has 0.5*mm* thickness and core is 10*mm*. Circular Cutout with doublers is introduced at sandwich structure of radius 0.3m, which comprises of aluminum facet 0.3*mm* thickness and core is 10*mm*. Mass of 235kg is placed at distance of 3m from origin and it is connected with top flange nodes with rigid links. Force 100kN is applied for 0.1sec to determine response for mass element.

PROCEDURE

STEPS

1. **Create a point for following coordinates**

	X	Y	Z
P1	0	0	0
P2	2	0	0
P3	2	0.02	0
P4	2	1.98	0
P5	2	2	0

*For creating Key Point P1

Command : POINT, ADD

Menu : Geometry→ Key Point→Create→By X/Y/Z

Parameters :

Coordinate data	0/0/0
Coordinate Type	0
Entity ID	

*Likewise you have to create point for other given coordinates.

2. **Create the line**

*Create line for above Key Points P2/P3.

Command : CURVE, LINE

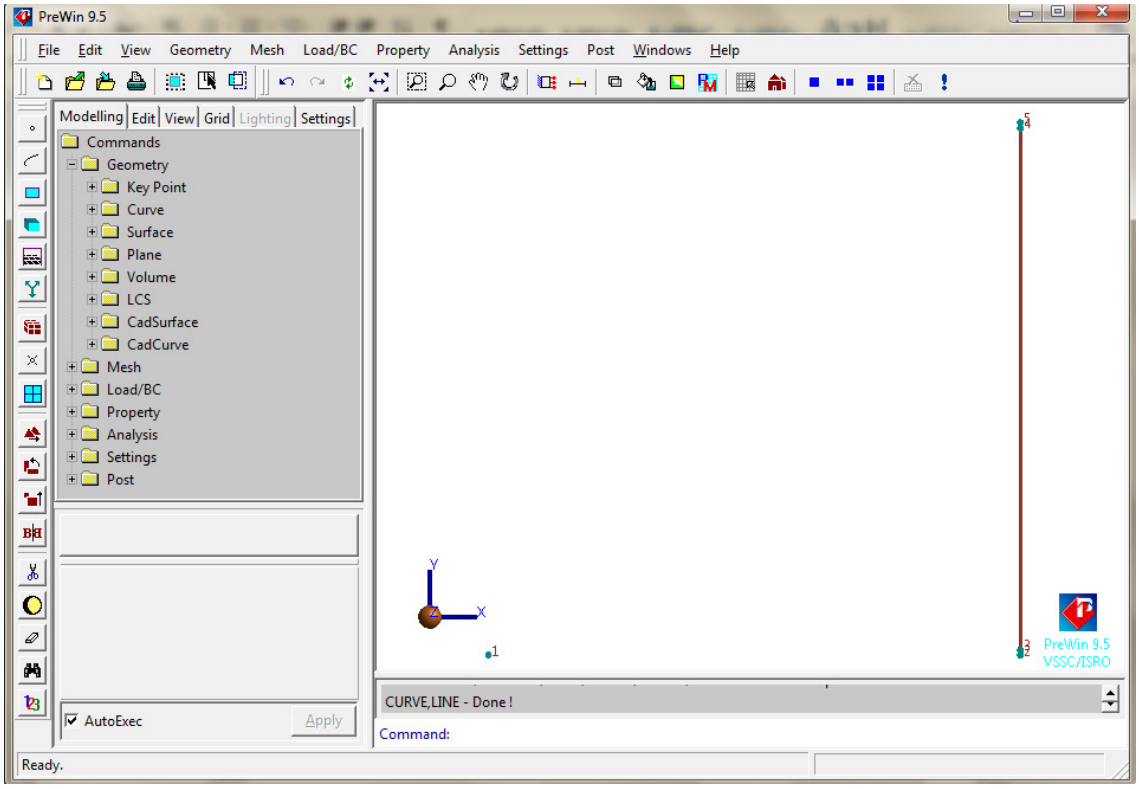
Menu : Geometry → Curve→ Create→ Line

Parameters :

Point Data	P2/P3
Entity id	

*Likewise you have to create line for P3/P4 and P4/P5

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

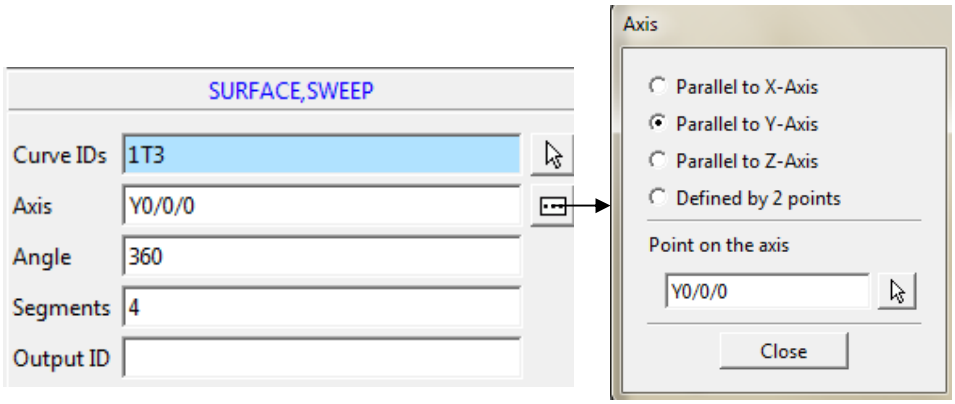


3. Create the rectangle surface

Command : SURFACE, SWEEP

Menu : Geometry → Surface → Create→ Using curves → Sweep

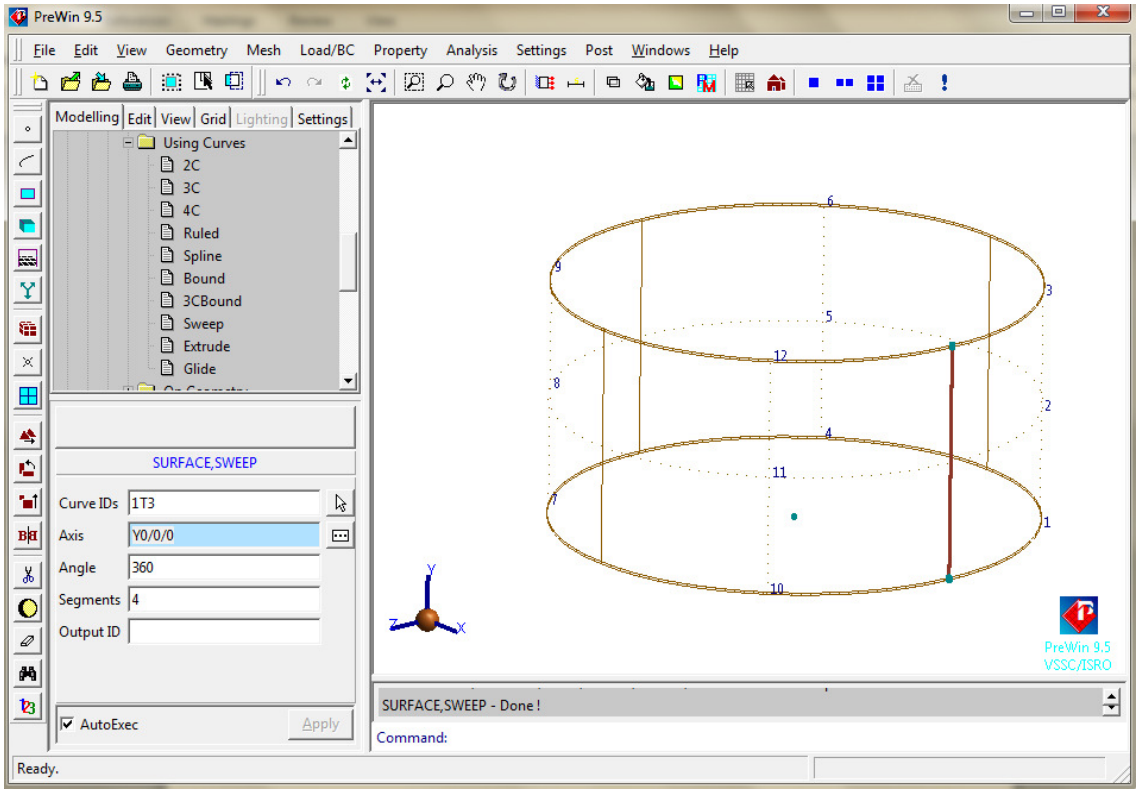
Parameters :



*Pick all the curves and define the axis& click apply

*12 surfaces will be created

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



*You can erase all curves by using the following commands

Command : CURVE,ERASE,ALL

Menu : Geometry → Curve→ Miscellaneous → Erase

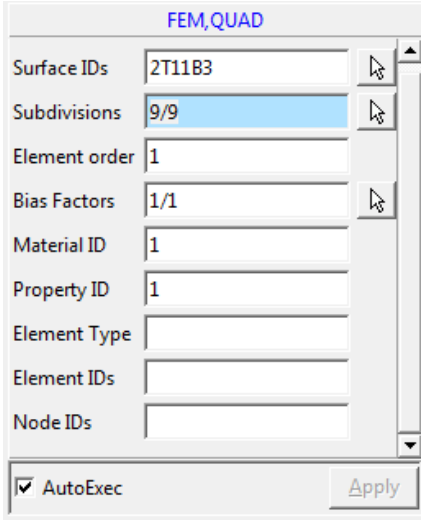
Note: To exit from any command action press 'Esc'.

4. Mesh the surface

Command : FEM,QUAD

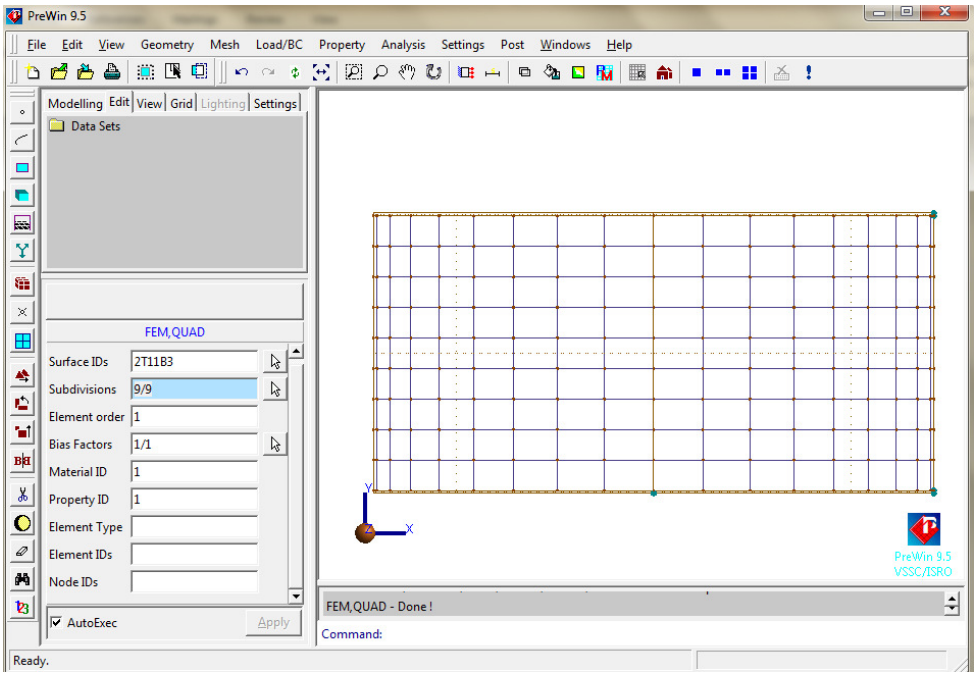
Menu : Mesh → FE Mesh → Generate QUAD

Parameters :



*First pick only mid surfaces in XY plane.

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



***Next step, repeat the same procedure as above pick all bottom and top surfaces in XY plane instead of mid surfaces & change sub divisions to 1/9**

* Click Apply.

*Once the Meshing is over, erase all the surfaces by using following commands

Command : SURFACE, ERASE, ALL

Menu : Geometry → Surface → Miscellaneous → Erase

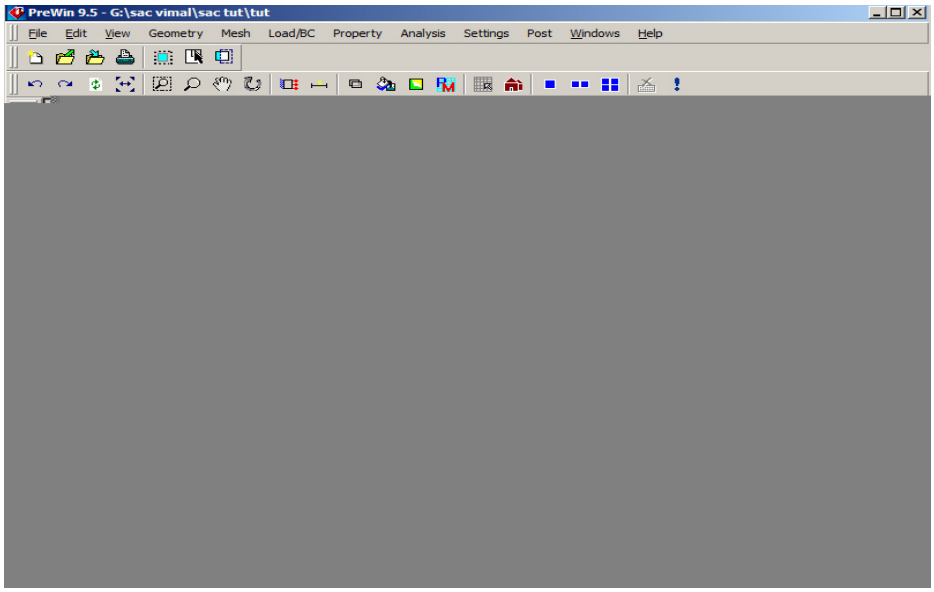
*Verify the total number of elements is **396** by using following commands

Command : ELEM, FIND, ALL.

5. Create Circular Cut-out

* You have to create cutout centre point before creating circular cutout

*For cutout point - Create point at centre on highlighted elements (Cutout with doubler surface) as shown below
* Click on "fill" button located on the menu bar to see the shaded view.

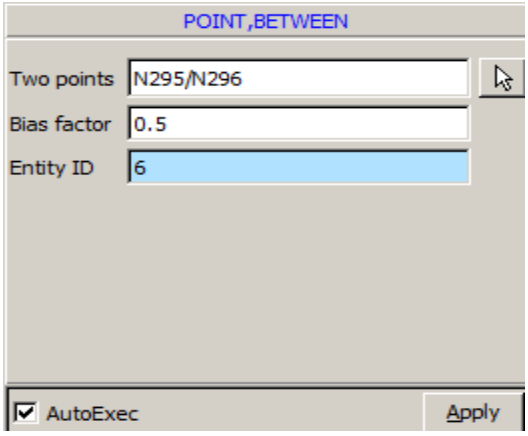


*Use the following command to create a cutout point
* Click to “snap to node” button to pick the node.

Command : POINT, BETWEEN

Menu : Geometry→ Key Point→Create→On Geometry→Between

Parameters:



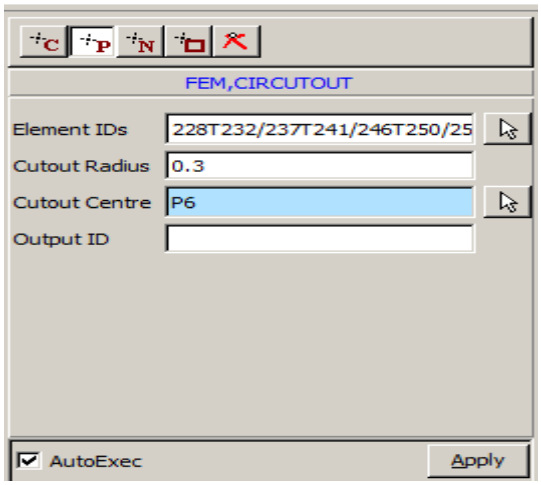
*Pick two nodes adjacent to point and click apply

*After creating cutout point, you can use the following command to create circular cutout

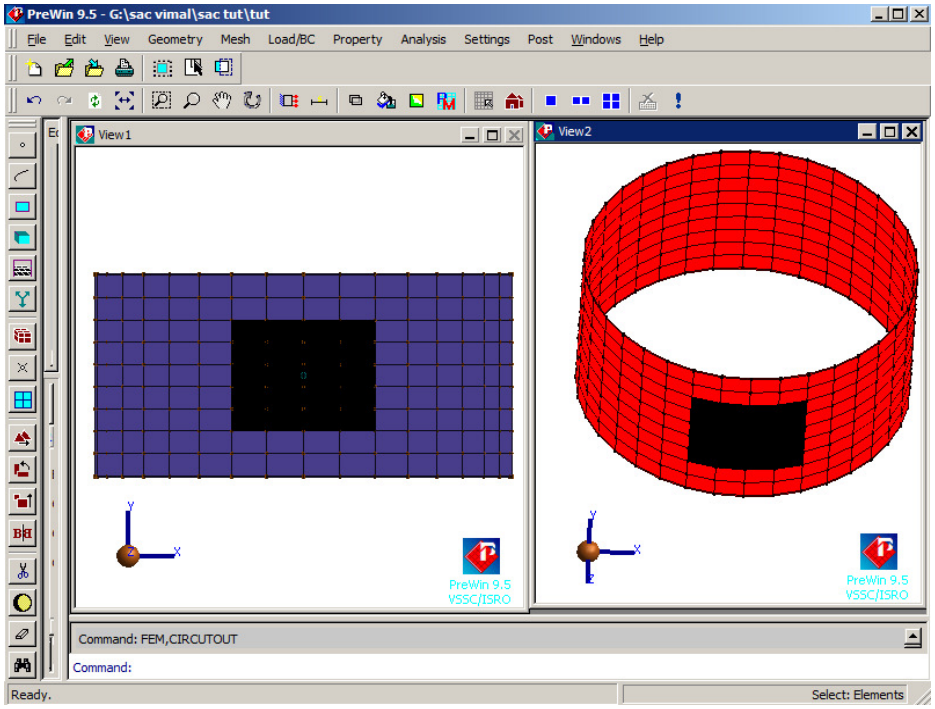
Command : FEM, CIRCUTOUT

Menu : Mesh → FE Mesh → Cutout → CircCutout

Parameters :



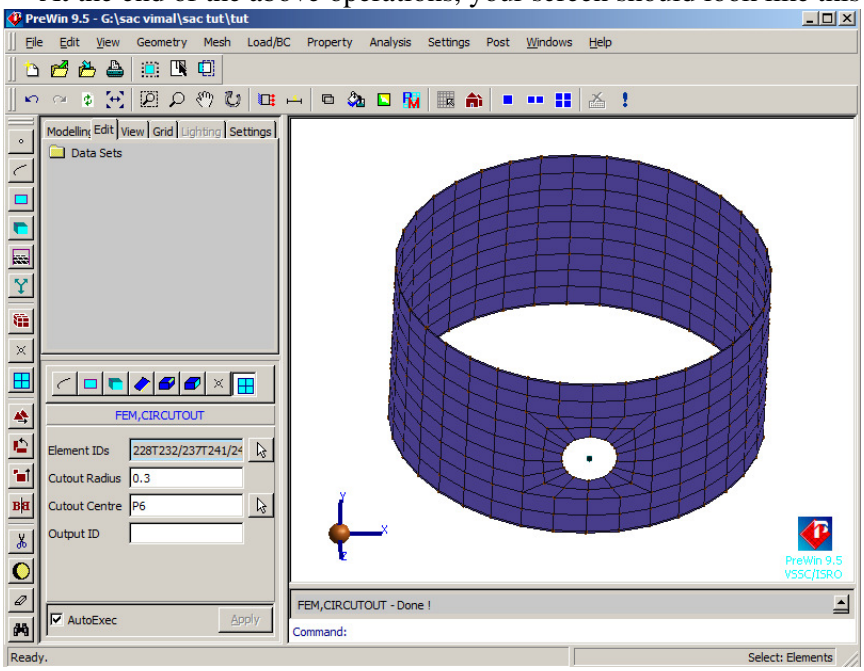
*Select the highlighted elements by individual click as shown below,



*Enter Cutout Radius value as 0.3

*Click Apply.

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



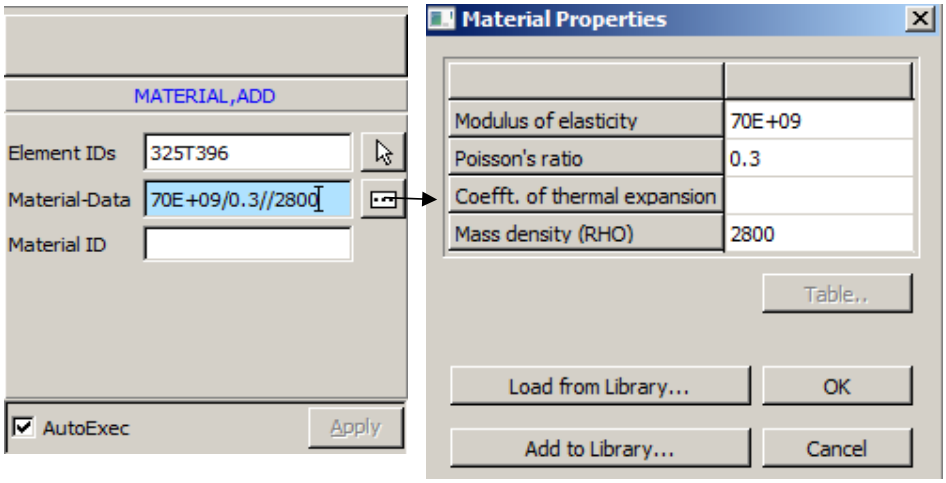
6. Create Material properties

Specify material properties for flange, facet and core, individually

1. Isotropic Material property for flange

Command: MATERIAL, ADD
Menu : Property → Material→ Isotropic → Add

Parameters:

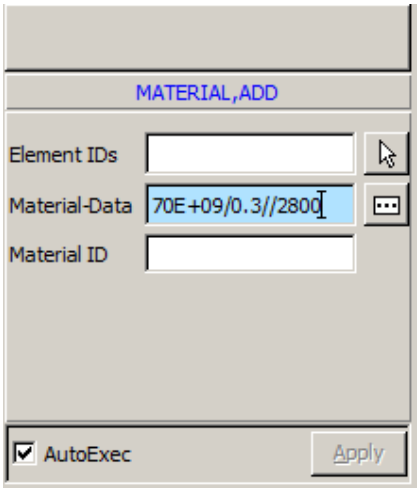


2. Isotropic Material property for aluminum facet

Command: MATERIAL, ADD

Menu : Property → Material → Isotropic → Add

Parameters:



*For sandwich and doubler no need to pick element, leave it as blank. Enter only material data and apply. Later, you will enter this material set id on Layup details.

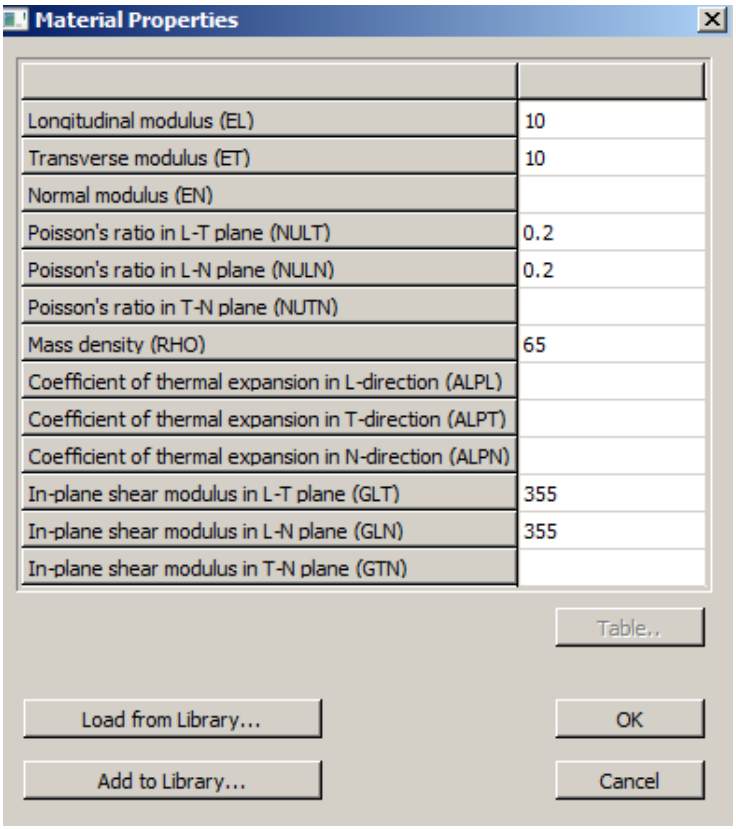
*Likewise have to create material properties for sandwich and doubler core.

*For sandwich and doubler core, pick anisotropic material, instead of isotropic.

3. Anisotropic Material property for core

Command: MATERIAL, ANISO

Menu : Property → Material → Anisotropic → Add



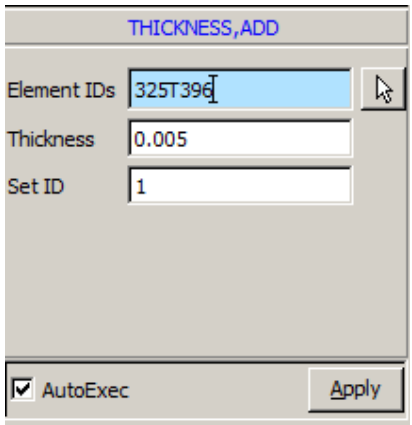
7. Create thickness

Specify shell thickness for flange, sandwich facet, doubler facet and core individually

1. Shell thickness for flange

Command: THICKNESS, ADD
Menu : Property → Physical → Thickness → Add

Parameter:

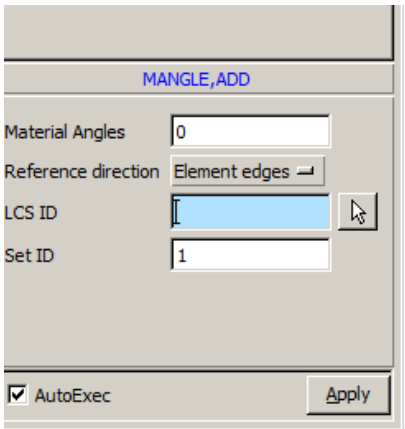


*Likewise will have to create thickness for sandwich facet, doubler facet and core.
*For sandwich facet, doubler facet and core no need to pick element in element id's, leave it as blank. Enter only thickness value and apply. Later, you will enter this thickness set id on Layup details.

8. Create Material angles

Command: MANGLE, ADD
Menu : Property → Physical → Material angle → Add

Parameters:



*Enter material angle as 0 and Click apply.
*Later, you will enter this Material angle set id on Layup details

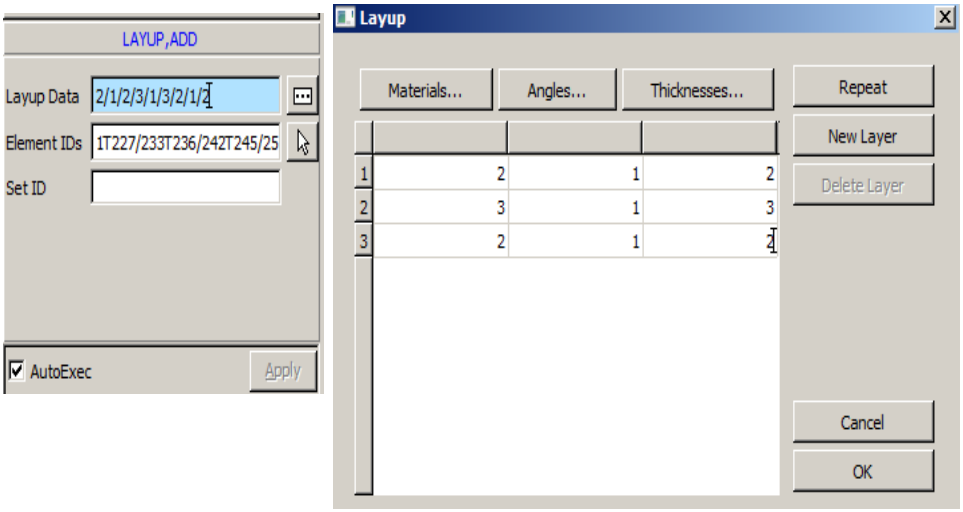
9. Create Layup details

1. Layup details for sandwich

Command: LAYUP, ADD

Menu : Property → Physical → Layup Details → Add

Parameters:

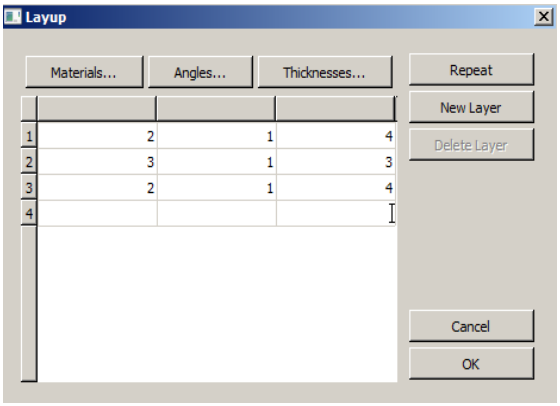


*Select the sandwich structure element IDs
*Enter material, angles and thickness set IDs into layup details in the order of Facet/Core/Facet.

2. *Layup details for cutout*

Command: LAYUP, ADD

Menu : Property → Physical → Layup Details → Add



* Select the cutout with doubler element IDs

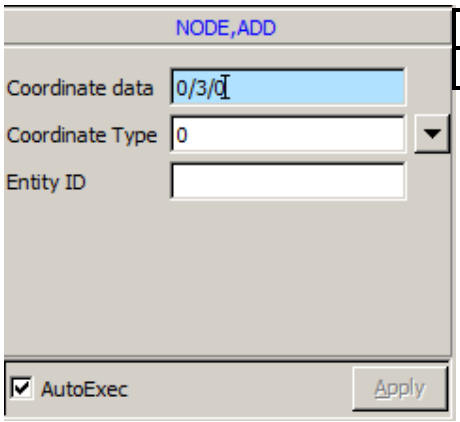
*Enter material, angles and thickness set IDs into layup details in the order of Facet/Core/Facet.

10. Create node for mass element

Command: NODE, ADD

Menu : Mesh→ Node →Create → Specify X/Y/Z

Parameters:



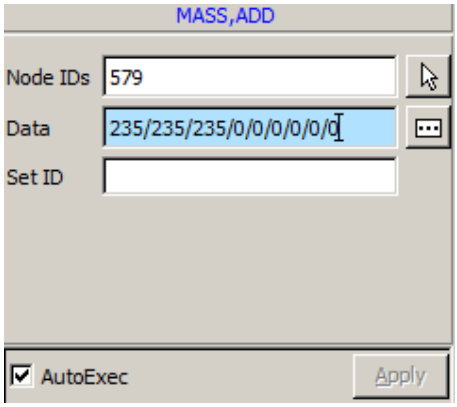
	X	Y	Z
N	0	3	0

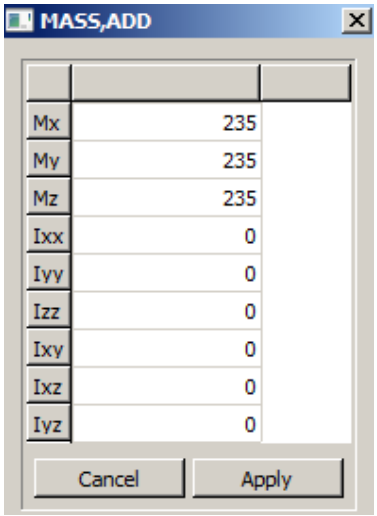
11. Create mass element

Command: MASS, ADD

Menu : Load → Concentrated Mass →Add

Parameters:



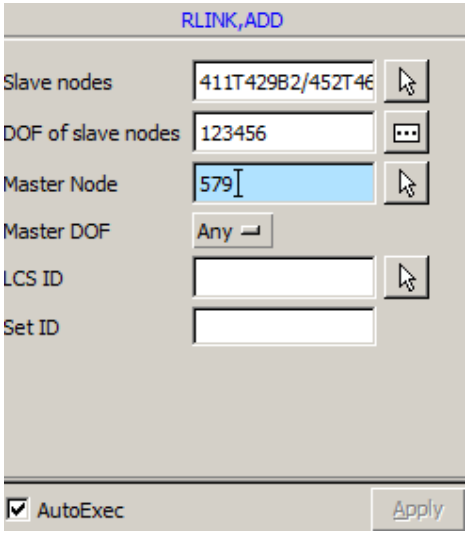


12. Create rigid link

Command: RLINK, ADD

Menu : Load → Rigid Link →Add

Parameters:



*For slave nodes pick only top flange nodes

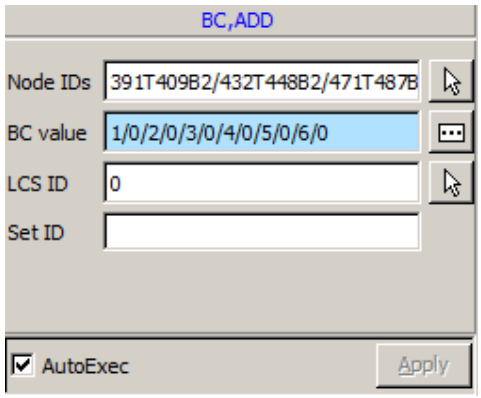
*For master nodes select mass element node and apply.

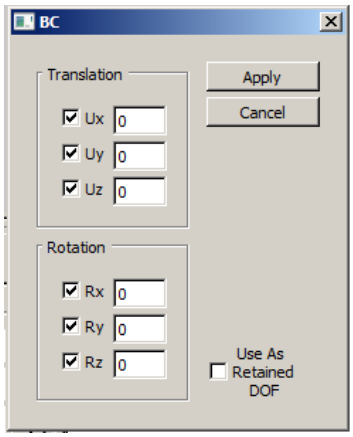
13. Specify displacement boundary conditions

Command : BC, ADD

Menu : Load/BC → Displacement BC → Add

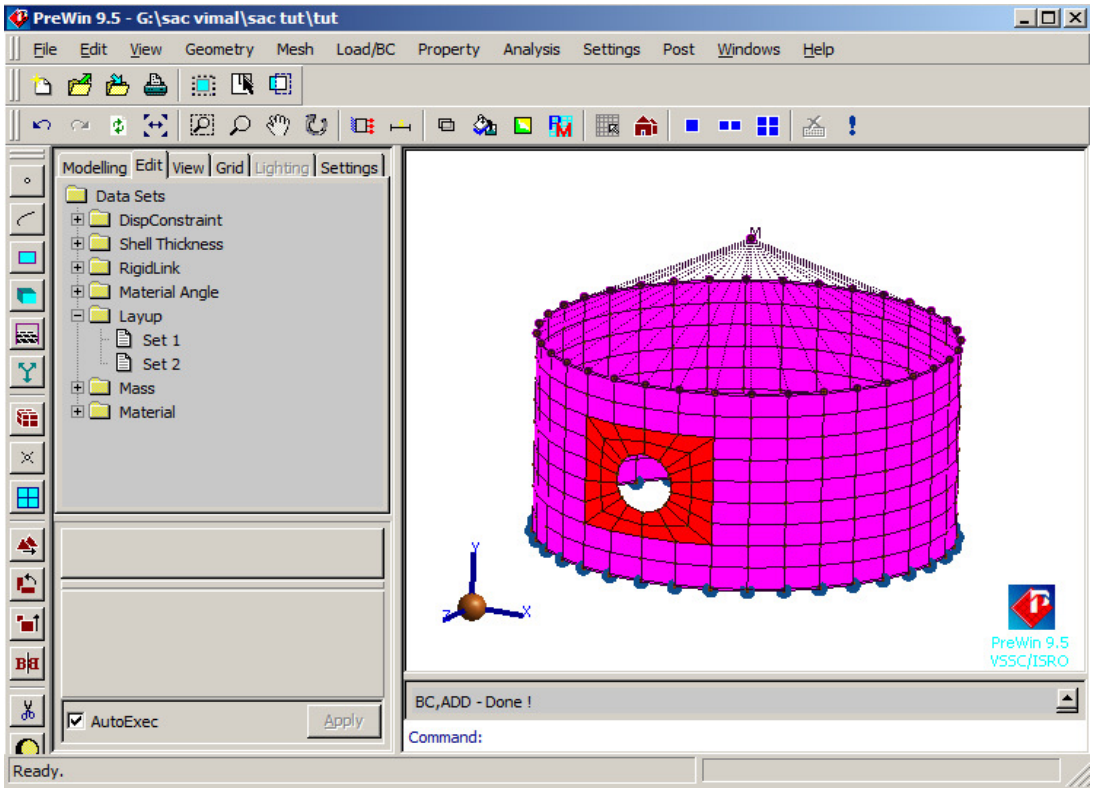
Parameters :





*Select bottom flange nodes only.

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

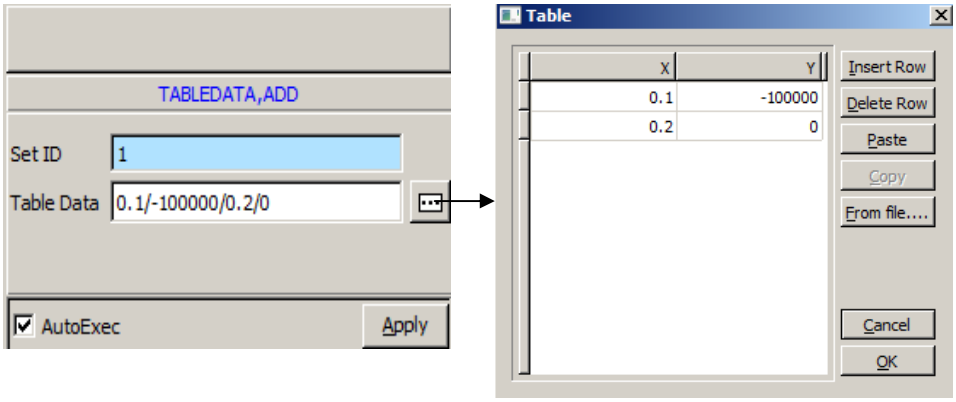


14. Create Table Data

Command : TABLEDATA, ADD

Menu : Property → Table Data→Add

Parameters:



*X value represents the Time (seconds) and Y represents Force (N).

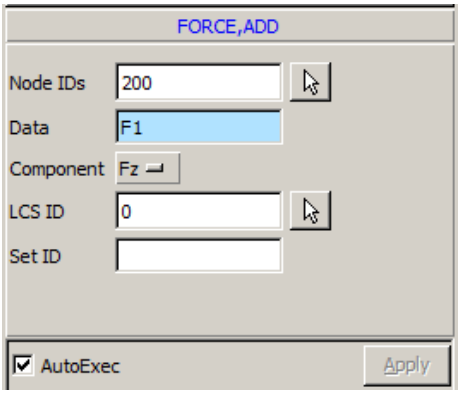
*This table data is the input for force, and it is denoted by **F1**.

15. Create Force as function of table data.

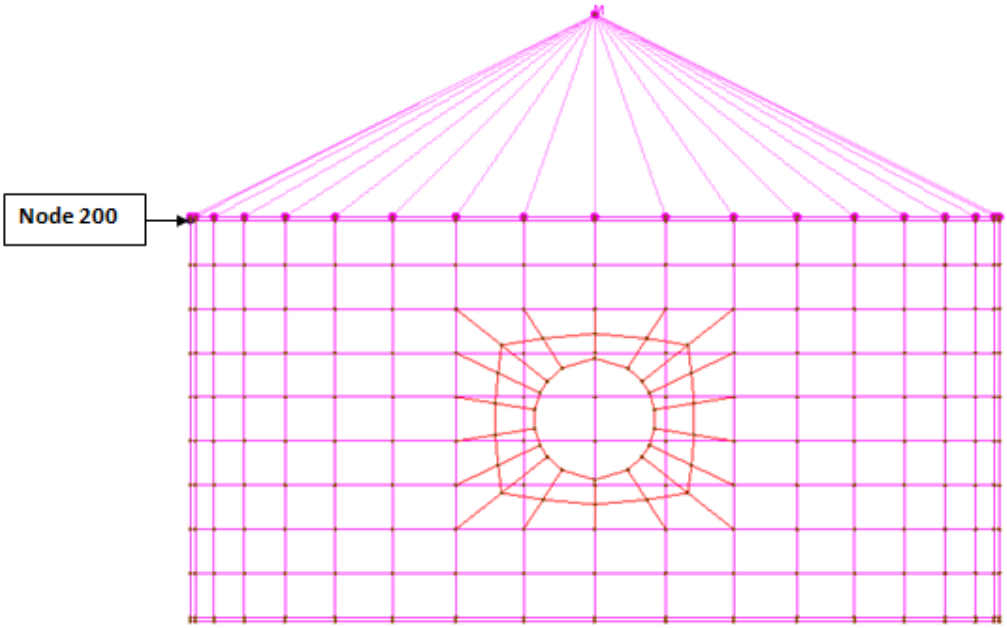
Command : FORCE, ADD

Menu : Load/BC → Point Load→Add

Parameters:



*select the node as mentioned below.



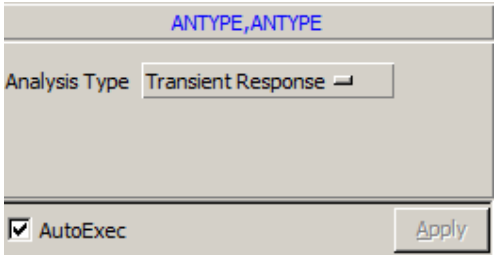
*Enter data as F1 and change component to Fz.

16. Set analysis type

Command : ANTYPE, ANTYPE

Menu : Analysis → Analysis Type→Transient Response

Parameters:



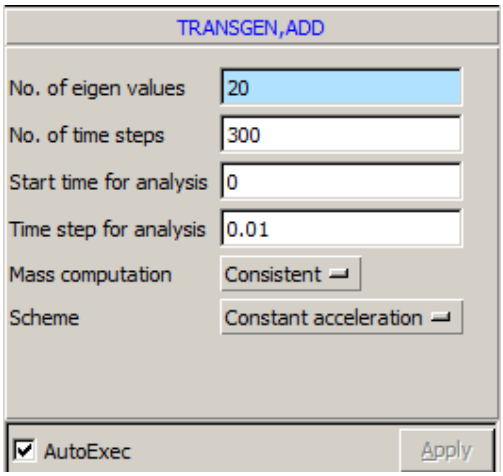
*Click Apply.

17. Set Transient Response General Data

Command : TRANSGEN, ADD

Menu : Analysis → Transient Response → General →Add

Parameters:



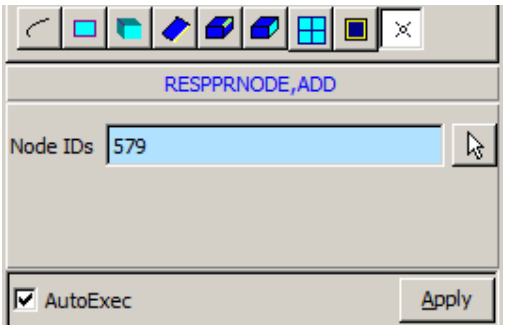
*Enter the mentioned data and apply.

18. Set Transient Response Node Data

Command : RESPPRNODE, ADD

Menu : Analysis → Transient Response → Nodes for output →Add

Parameters:



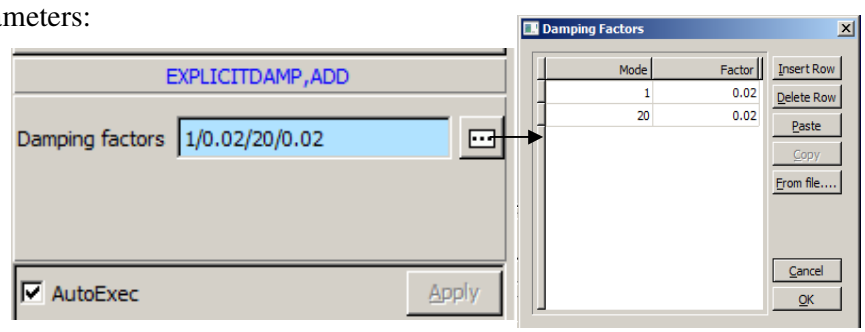
*Select the Node of Mass element and apply.

19. Create Damping Data

Command : EXPLICITDAMP, ADD

Menu : Analysis → Transient Response → Damping→Add

Parameters:



*Enter Damping factor is 0.02 for 1st and 20th Mode.

*Once created the Damping data, save the project file using following method
Menu : File →Save

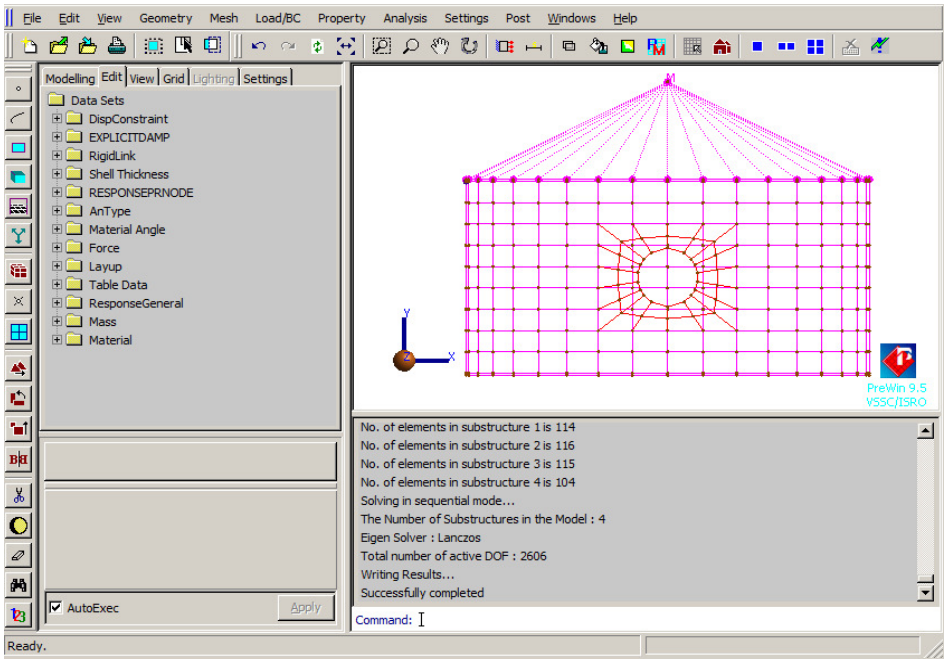
20. Submit the job into FEAST^{SMT}

*After saving the *.prj file, will have to submit the model into solver.

Menu: Analysis → Run Solver

*once click the Run solver, it will ask confirm message to create data file and output file.

*Click yes



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

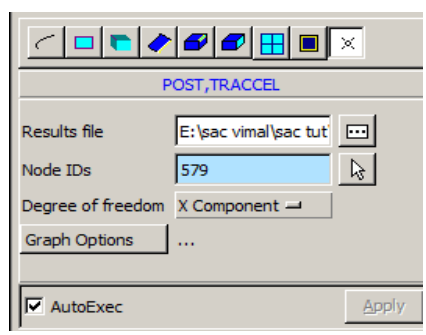
21. Perform post processing

a. Frequency vs. Acceleration plot

Command: POST, TRACCEL

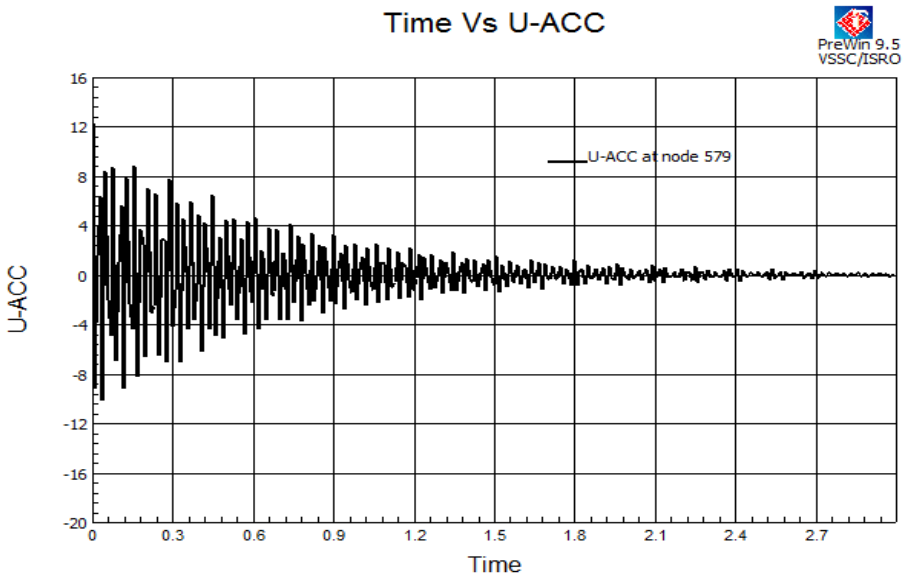
Menu: Post→ Transient Response→X-Y plot → Acceleration

Parameters:



*Select the Node of Mass element
*Change DOF to X component and apply.

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



*Likewise you can plot graph for Frequency vs. displacement and Frequency vs velocity.

b. Output can be seen in *.OUT file.