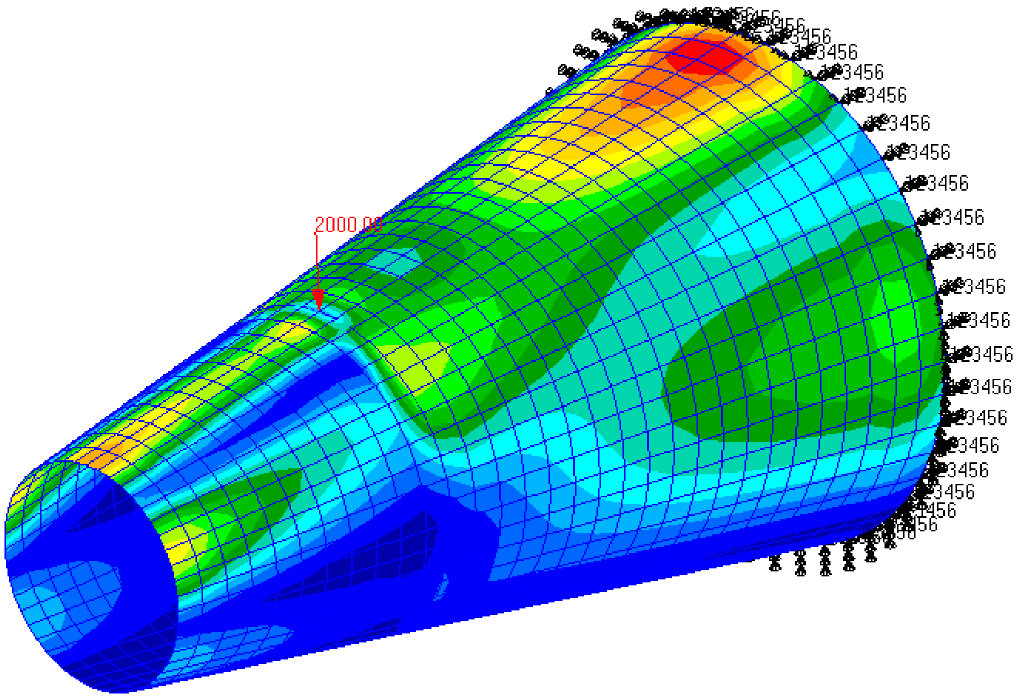
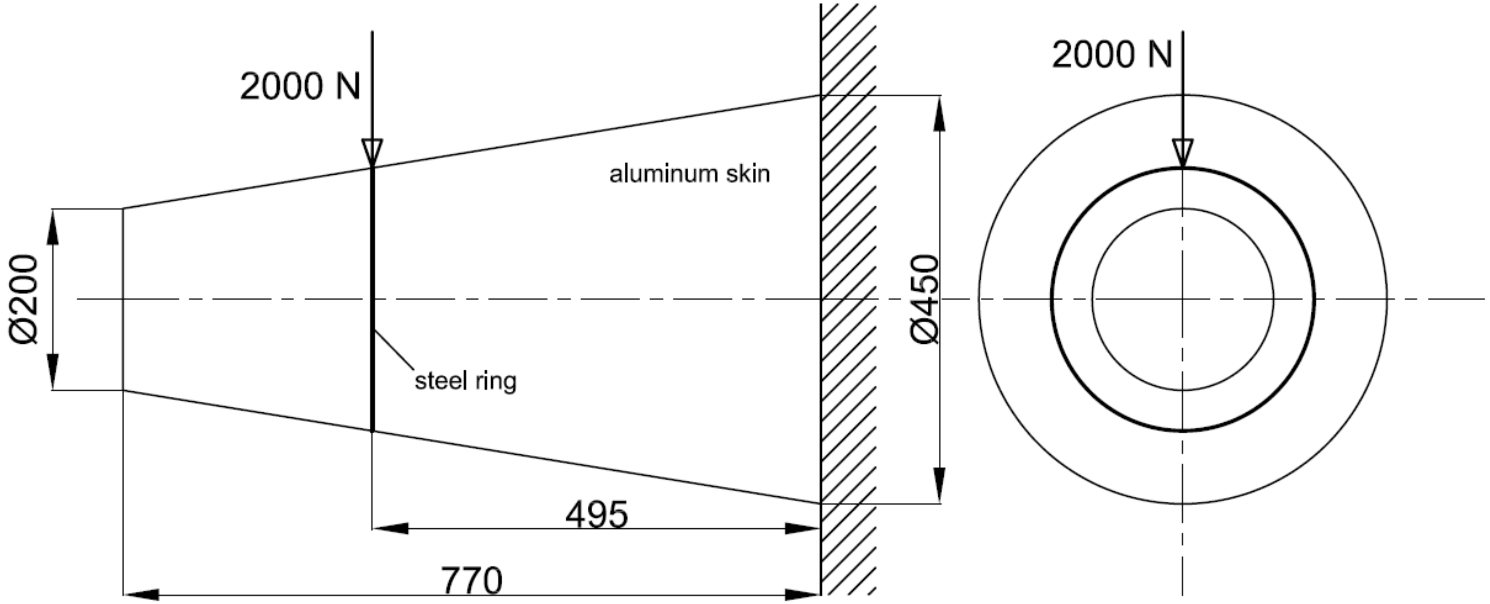
Mechanics of Thin-walled Structures

CONICAL



PROBLEM DESCRIPTION



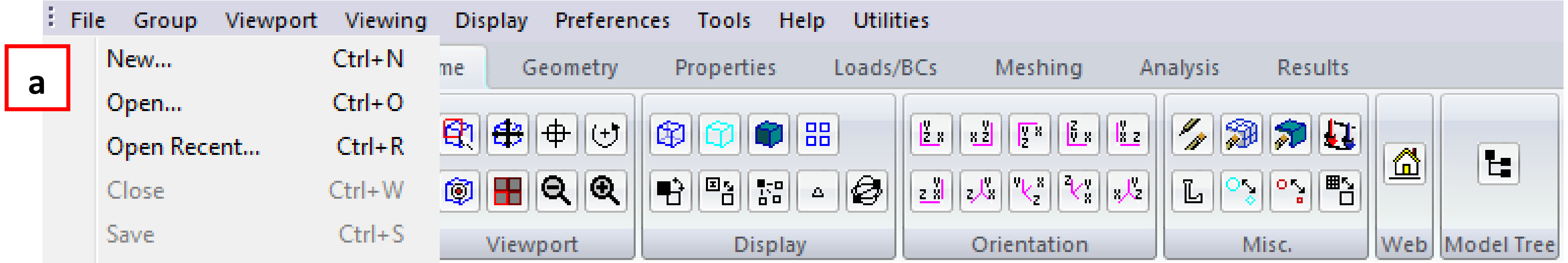
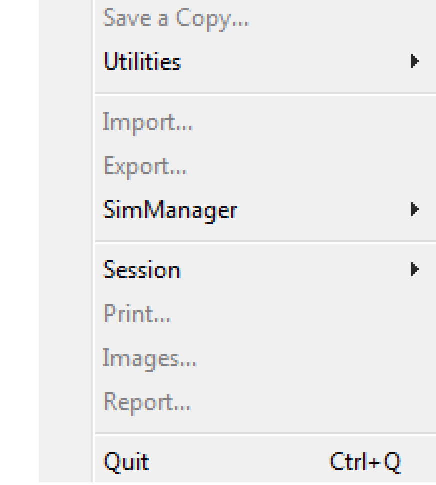
Static, linear analysis of thin-walled aluminum skin, reinforced by a stiffening ring, will be performed.

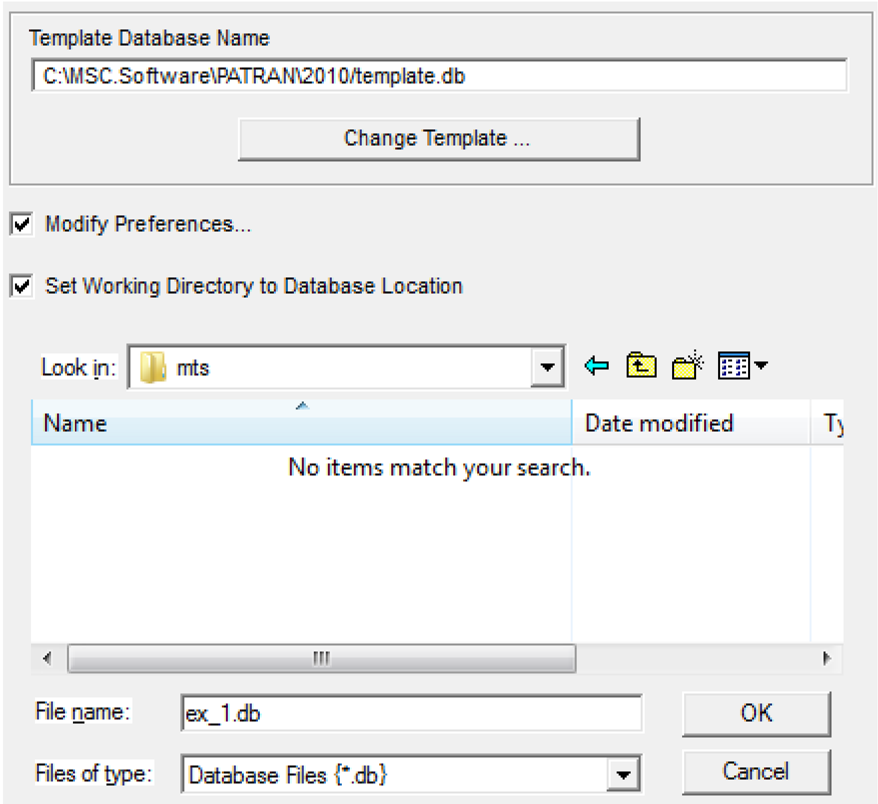
In the course of this exercise, students will gain basic knowledge about: **creating geometry and mesh; applying loads and boundary conditions**; **setting up static, linear analysis** and **results post-processing.**

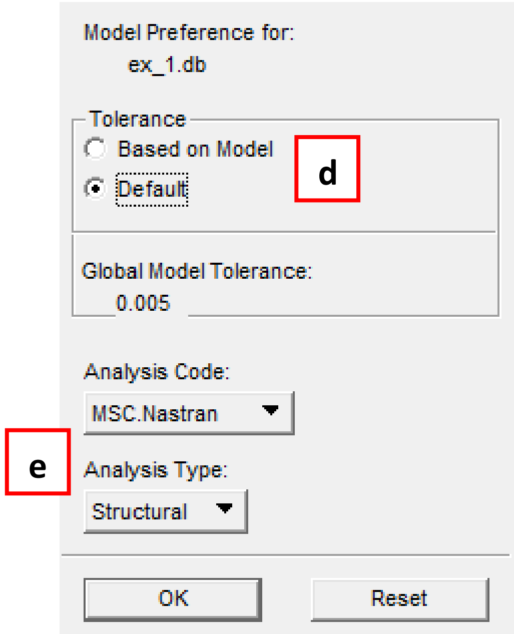
Two cases will be considered:

1. When stiffening ring is (relatively) **rigid**
2. When stiffening ring is made of **steel** (and its deformations influence the deformation of the aluminum skin)

Units: mm, N, MPa

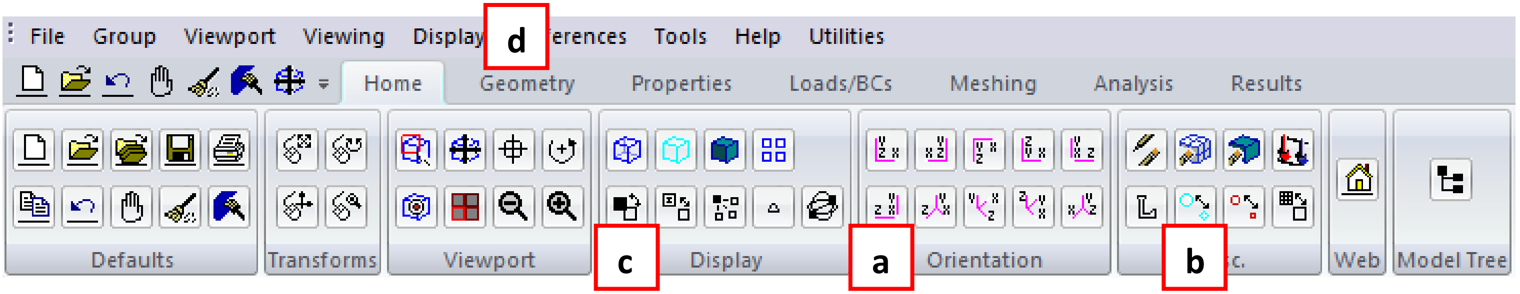



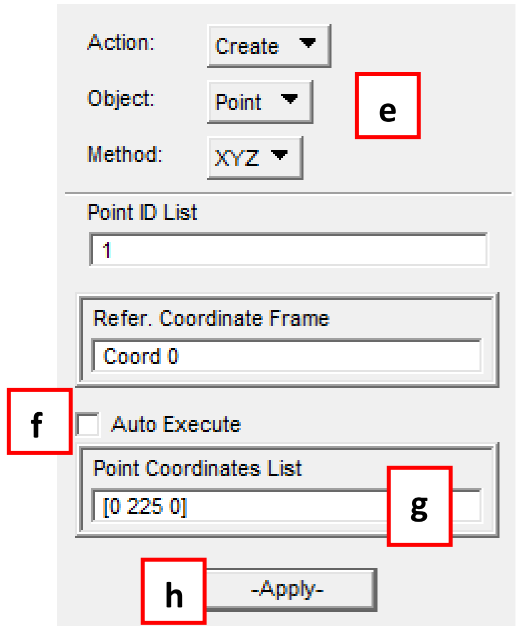


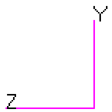


Create a new database:

a. **File / New...**
b. Enter **ex_1.db** as the File name
c. Click **OK**
d. Select **Default**
e. Click **OK**





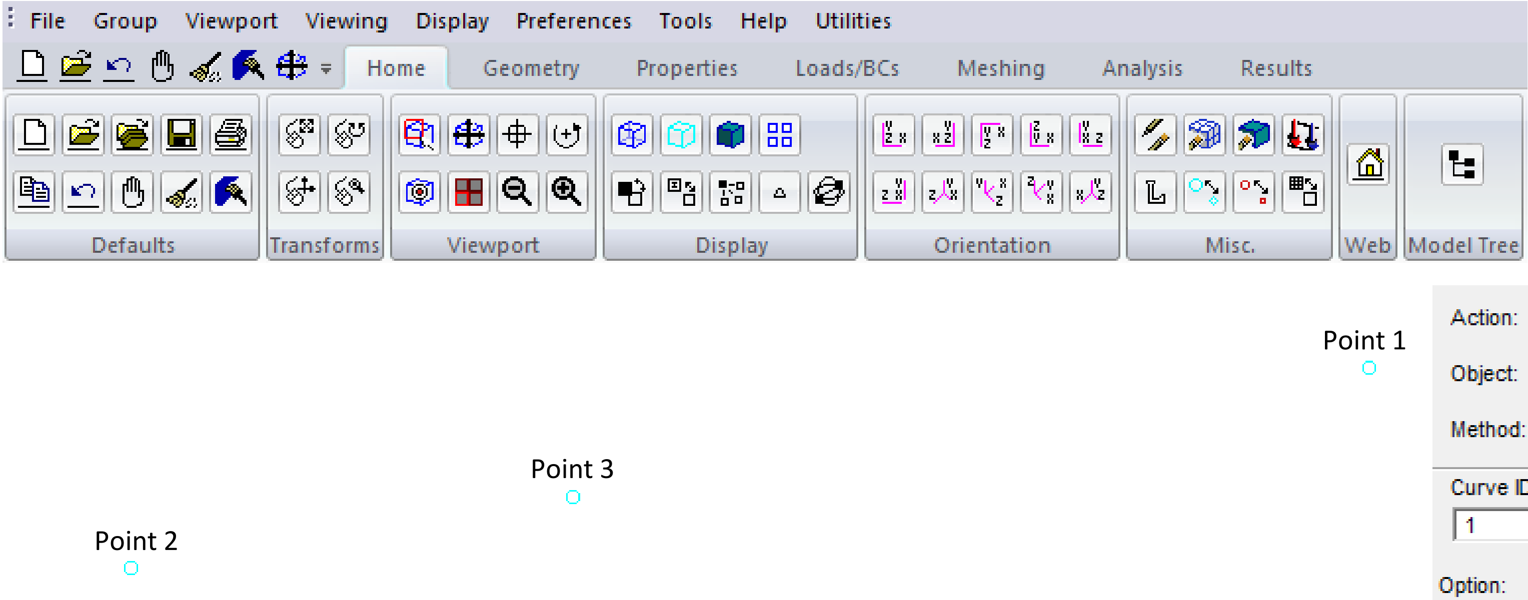


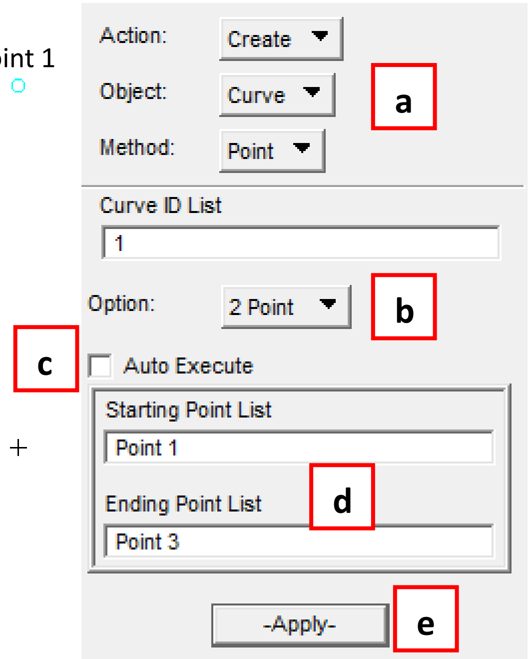
Create geometry points:

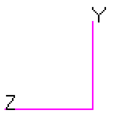
a. Click on the **Right side view** icon
b. Click on the **Point size** icon
c. Click on the **Cycle Background** icon
d. Click on the **Geometry** icon
e. Geometry: **Create/Point/XYZ**
f. Uncheck **Auto Execute**
g. Enter **[0 225 0]** as the Point Coordinates List
h. Click **Apply**
i. Create two more points using coordinates: **[0 100 770]**, **[0 144 495]**

Auto-Execute Option

The Auto-Execute option will automatically apply the operation when the last field is filled in.



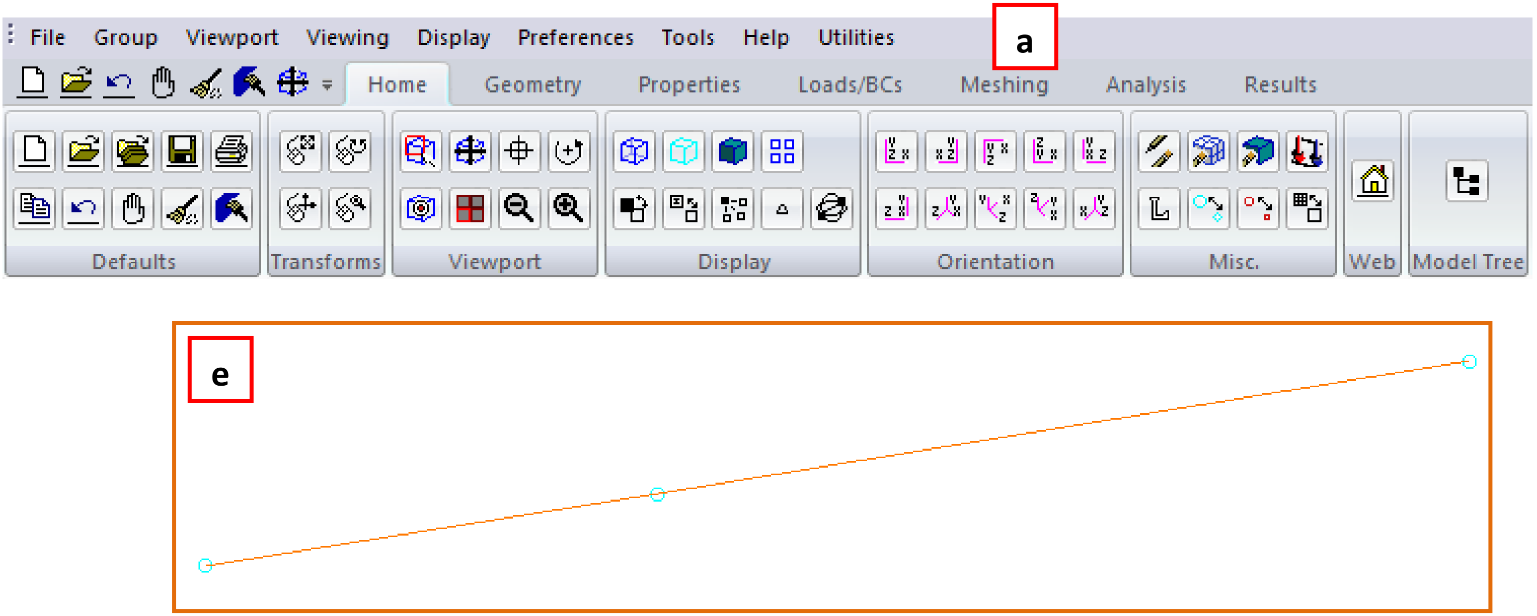




Create curves:

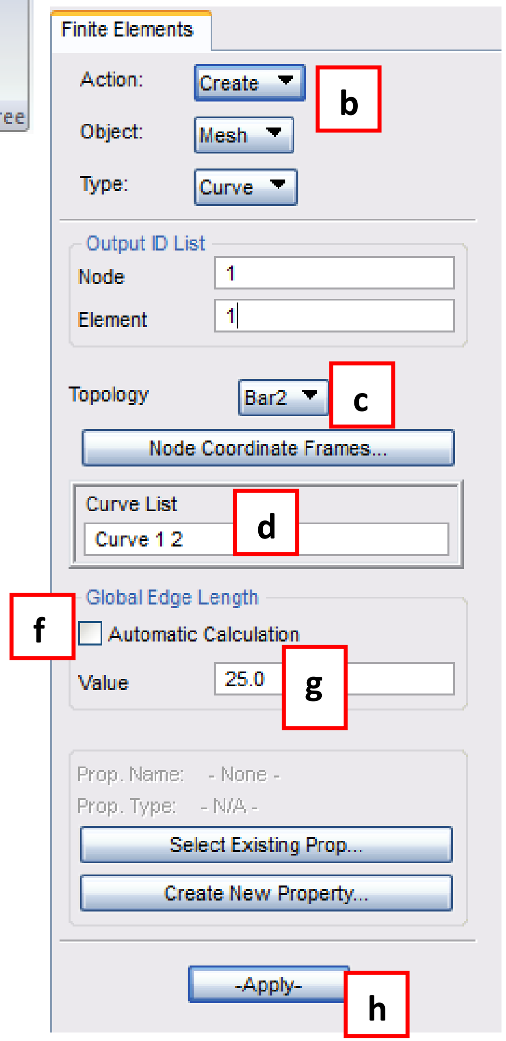
a. Geometry: **Create/Curve/Point**
b. Option: **2 Point**
c. Uncheck **Auto Execute**
d. Select **Point 1** as the starting point and **Point 3** as the ending point
e. Click **Apply**
f. Create one more curve using **Point 3** and **Point 2**

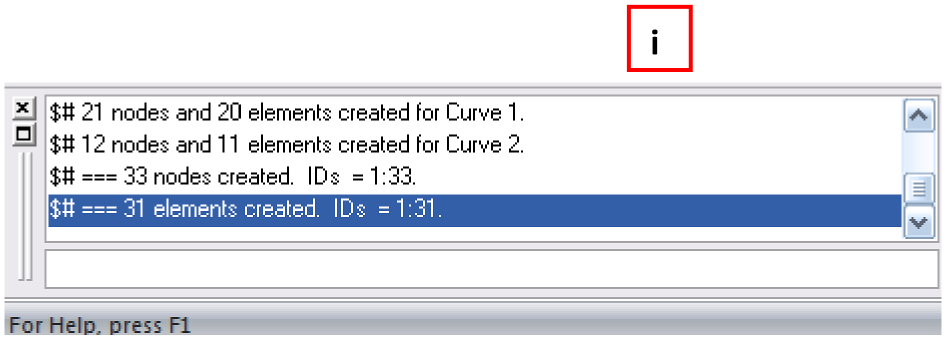
Remark: Points numbers may be different.

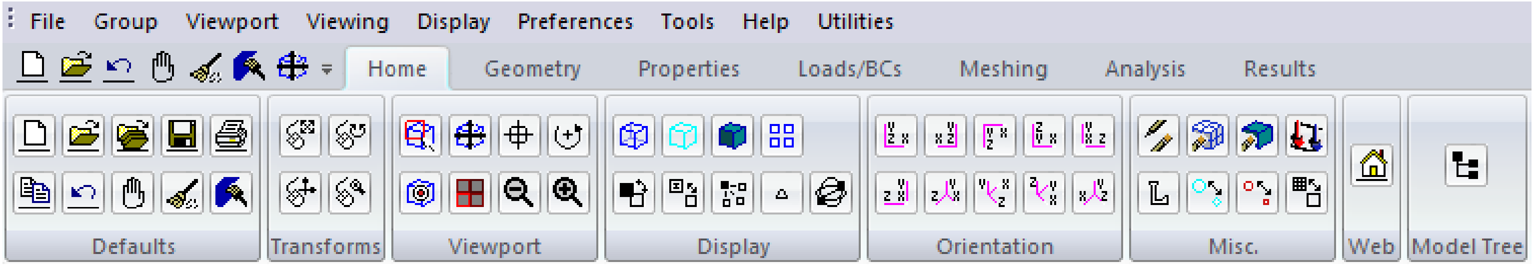


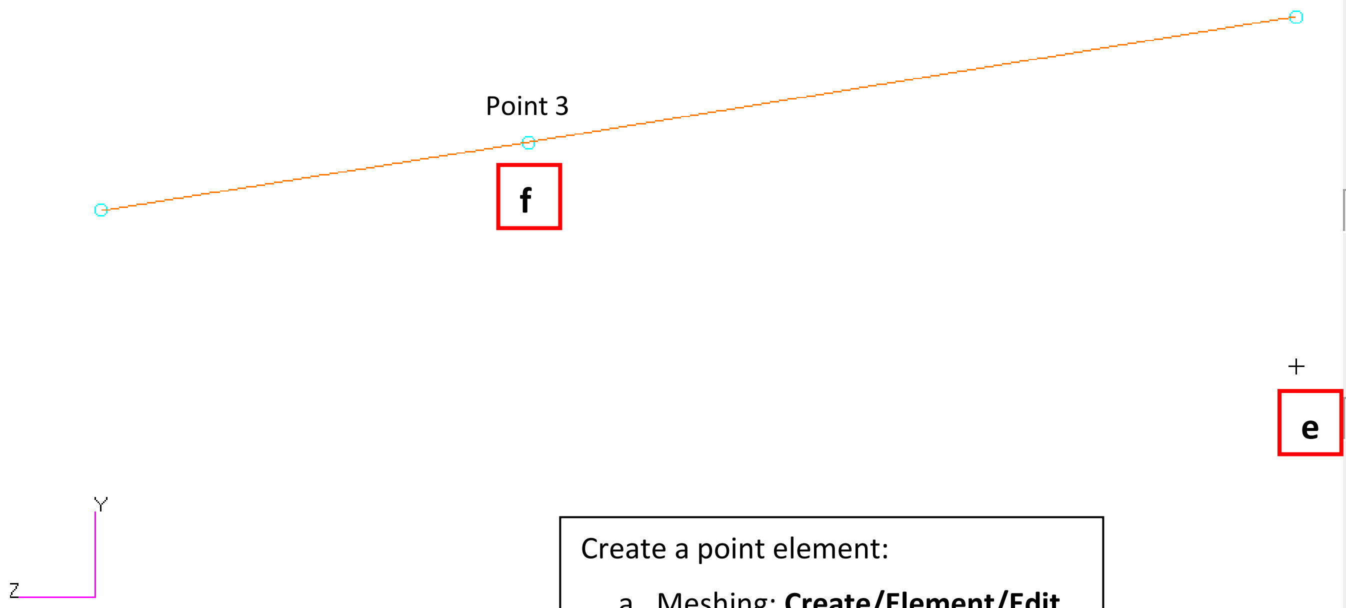
Mesh the curves:

a. Click on the **Meshing** tab
b. Meshing: **Create/Mesh/Curve**
c. Topology: **Bar2**
d. Click on the **Curve List** panel
e. Select all curves by clicking and dragging the mouse
f. Uncheck **Automatic Calculation**
g. Enter **25** as the Value of the Global Edge Length
h. Click **Apply**
i. Check message window whether elements were created



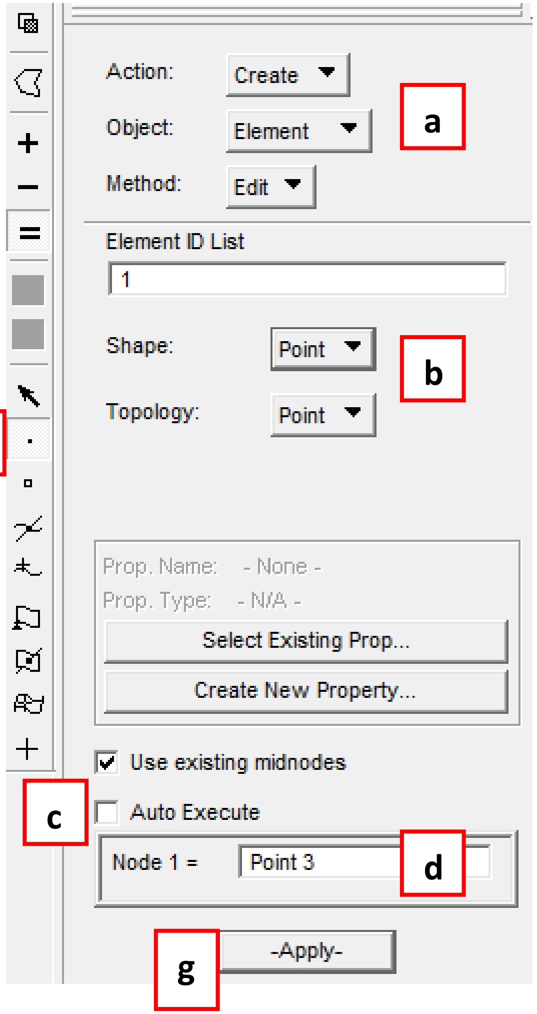


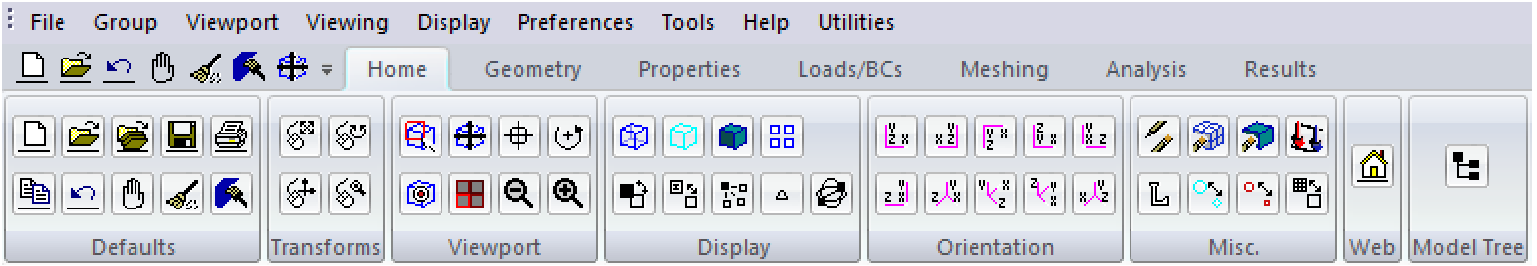




Create a point element:

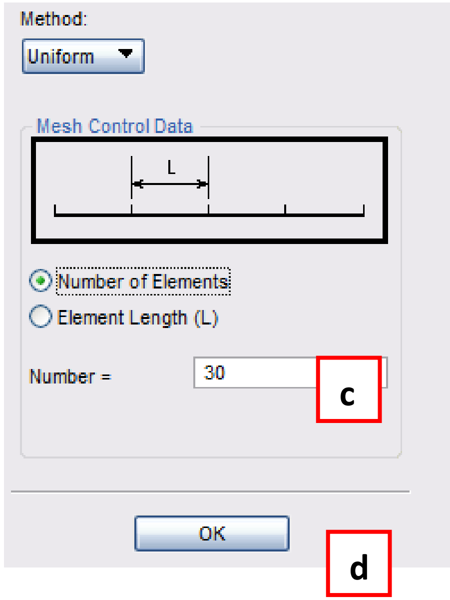
a. Meshing: **Create/Element/Edit**
b. Shape and Topology: **Point**
c. Uncheck **Auto Execute**
d. Click on the **Node 1** panel
e. Select **Point** icon
f. Select **Point 3**
g. Click **Apply**

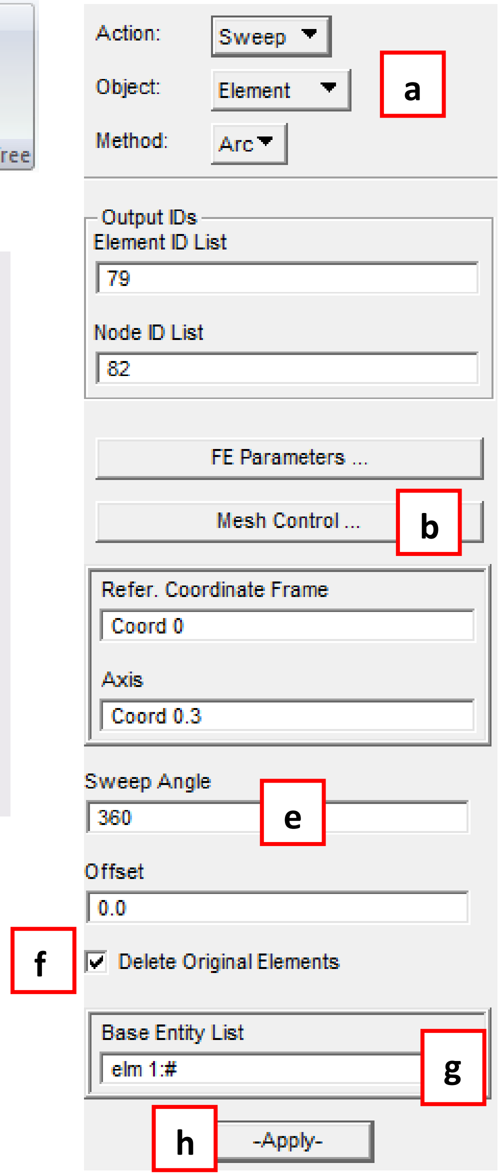


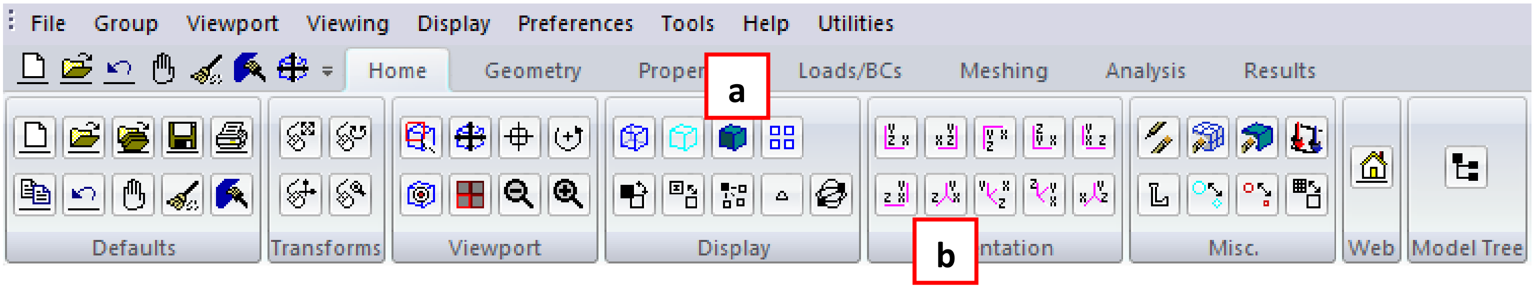


Sweep the elements to create the final mesh:

a. Meshing: **Sweep/Element/Arc**
b. Click **Mesh Control…**
c. Enter **30** as the Number of Elements
d. Click **OK**
e. Enter **360** as the Sweep Angle
f. Check **Delete Original Elements**
g. In the **Base Entity List** enter **elm 1:#** to select all previously created elements
h. Click **Apply**

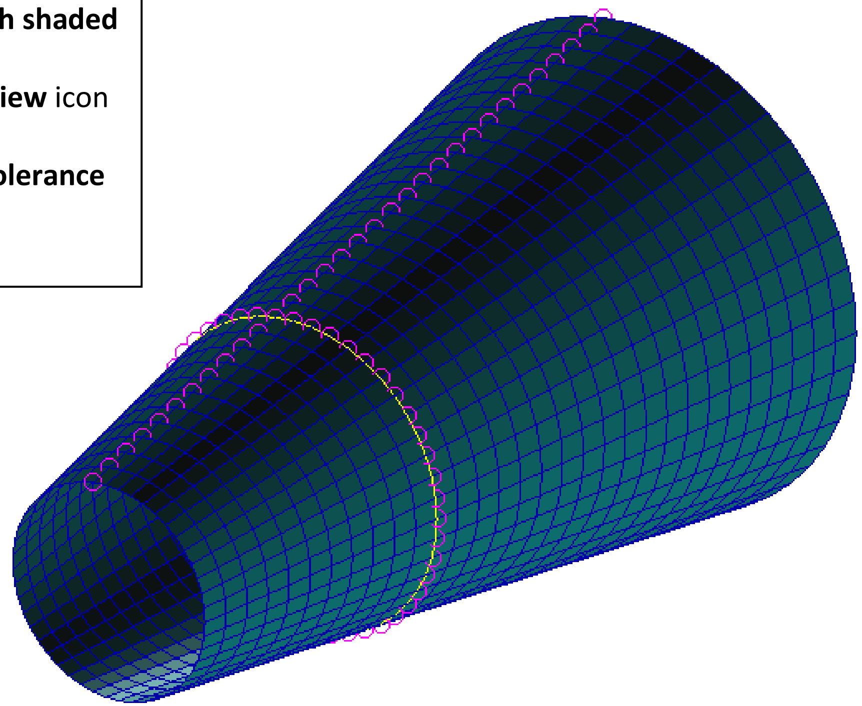


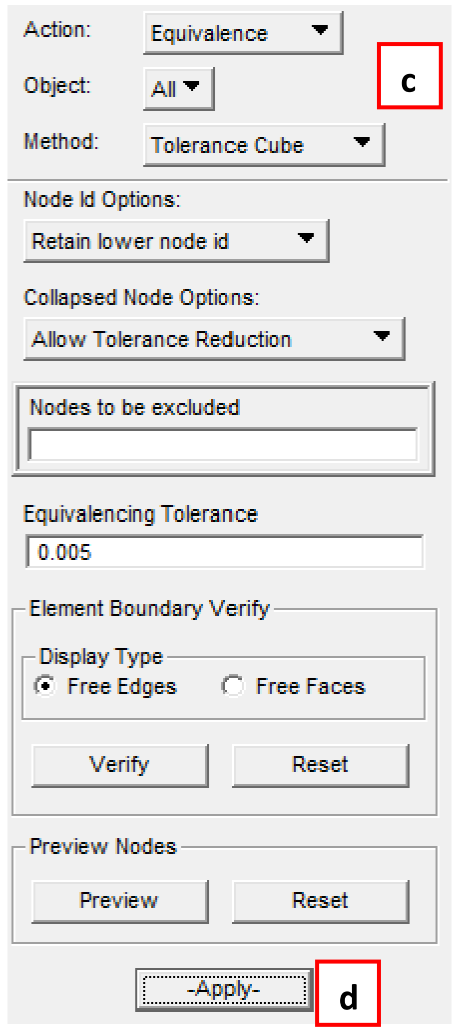


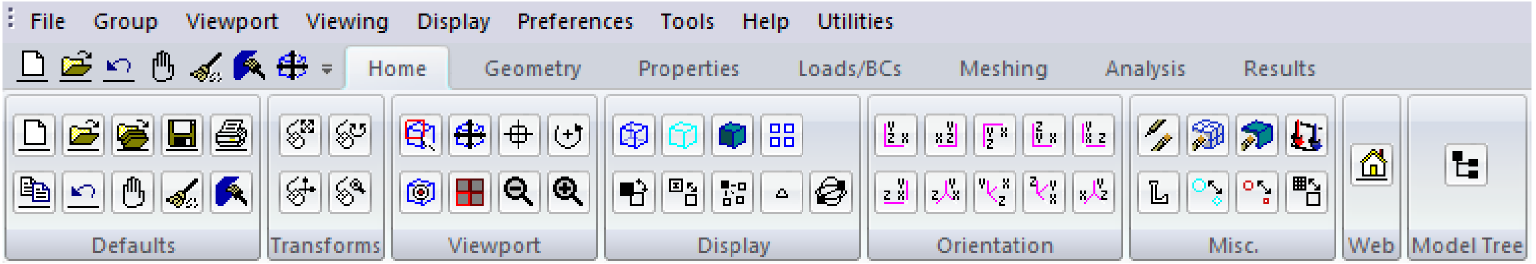


Delete the duplicate nodes:

a. Click on the **Smooth shaded** icon
b. Click on the **Iso 1 View** icon
c. Elements: **Equivalence/All/Tolerance Cube**
d. Click **Apply**



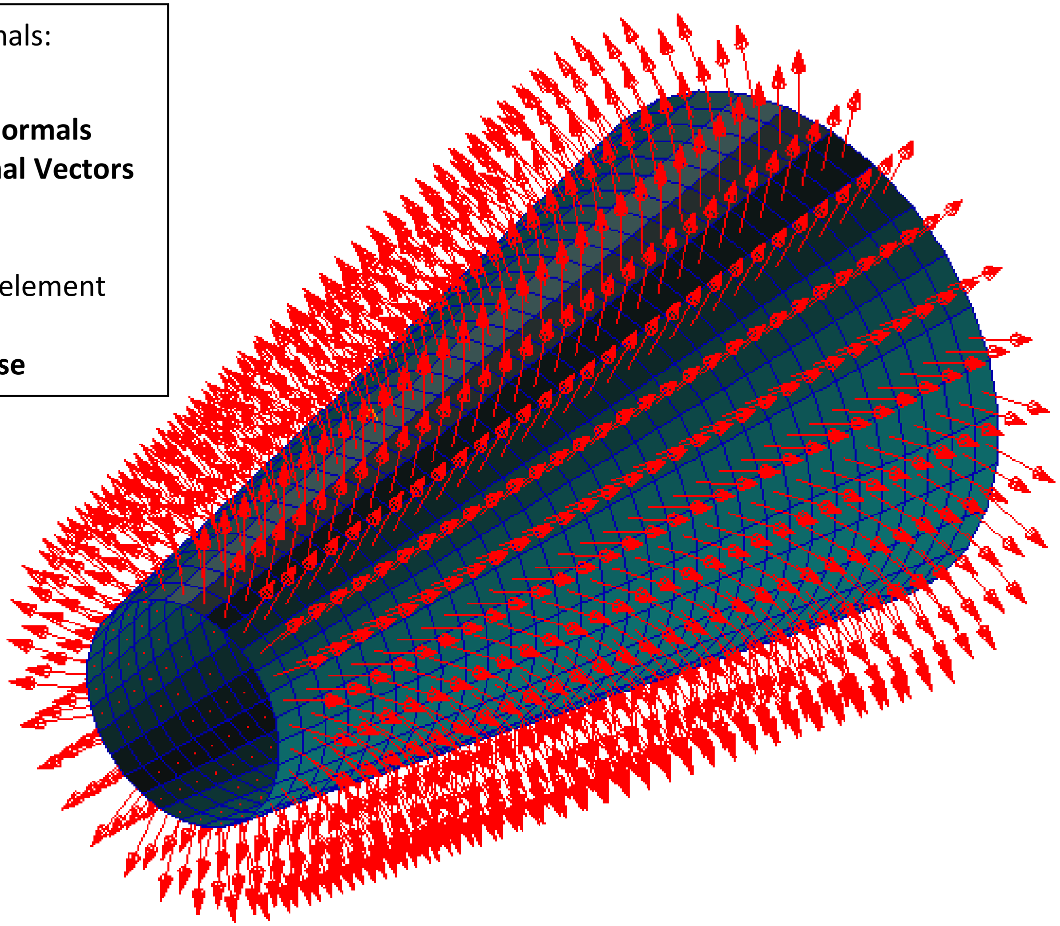


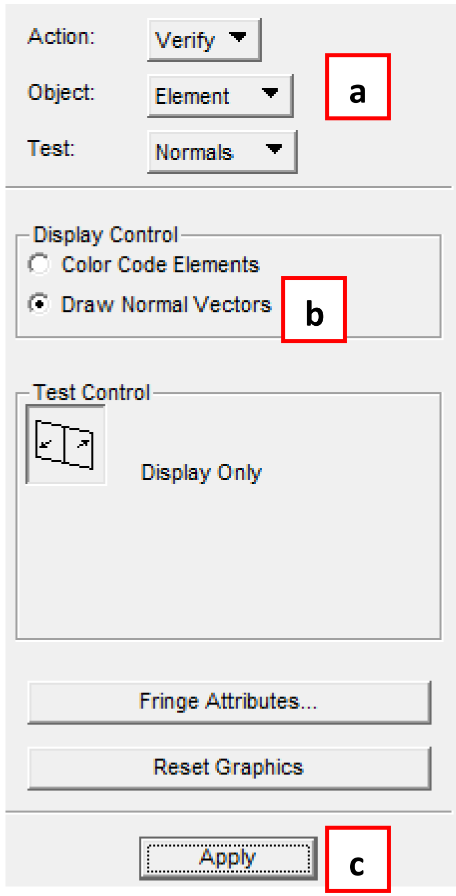


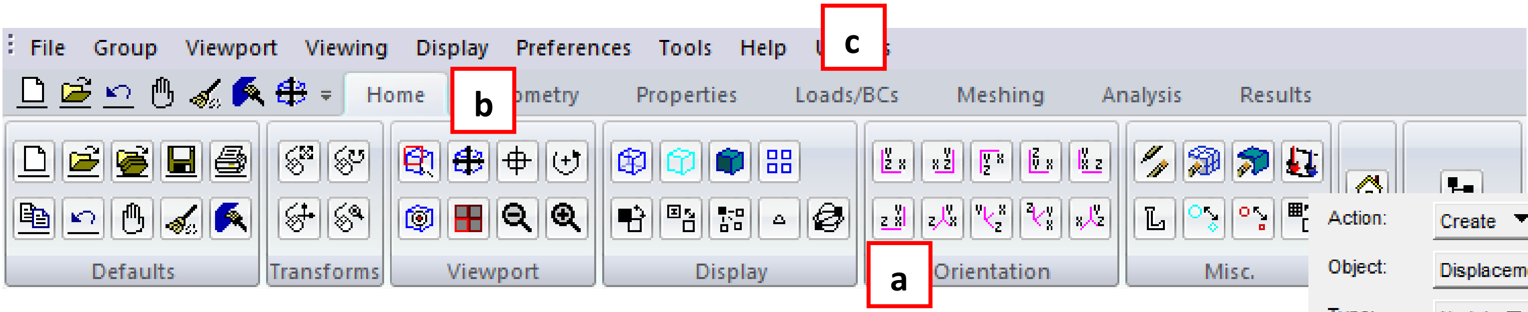
Verify the element normals:

a. Meshing: **Verify/Element/Normals**
b. Select **Draw Normal Vectors**
c. Click **Apply**

<u>Remark</u>: To reverse the element normals use **Elements: Modify/Element/Reverse**

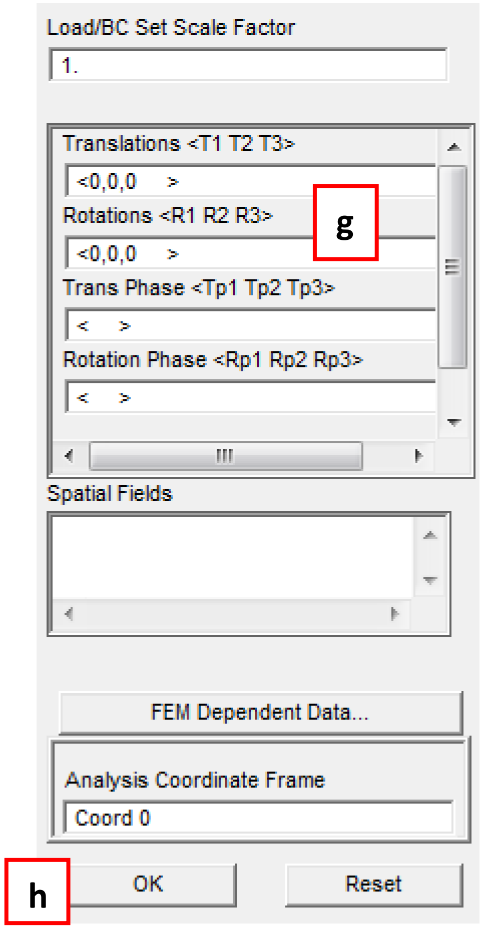


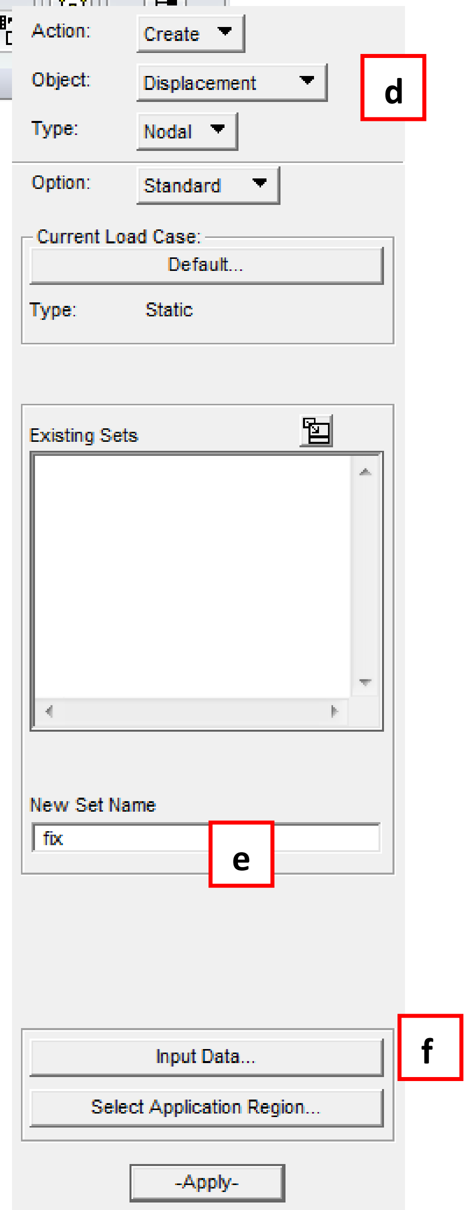




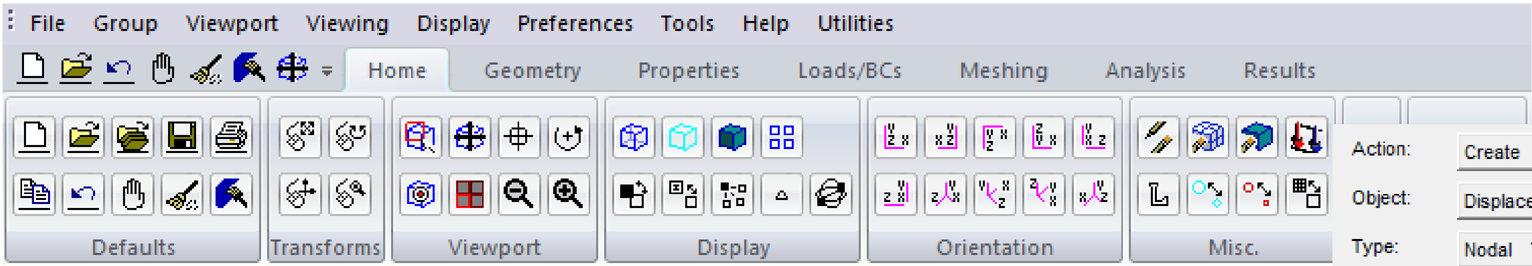
Apply the boundary conditions:

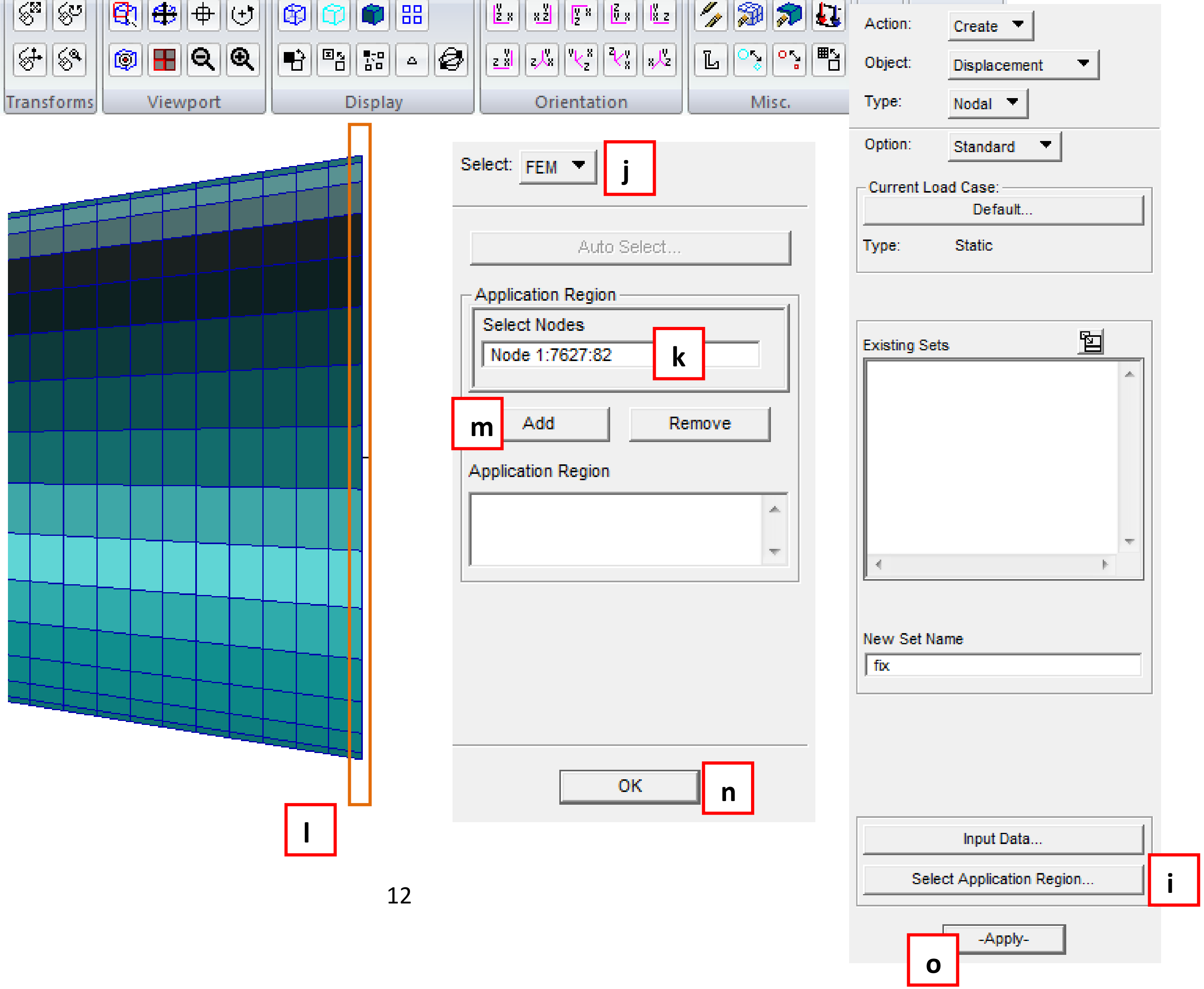
a. Click on the **Right side view** icon
b. Click on the **Fit view** icon
c. Click on the **Loads/BCs** icon
d. Loads/BCs: **Create/Displacement/Nodal**
e. Enter **fix** as the New Set Name
f. Click **Input Data…**
g. Enter **<0,0,0>** for the Translations and for the Rotations
h. Click **OK**

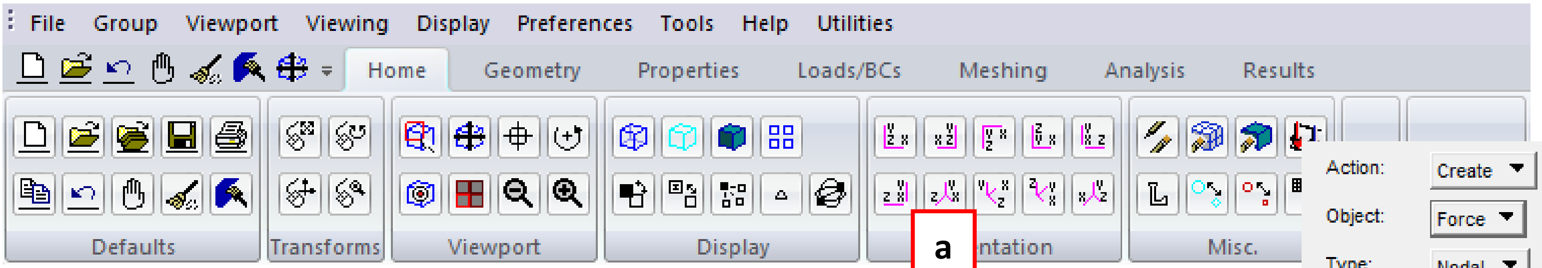




i. Click **Select Application Region…**
j. Select **FEM**
k. Click on the **Select Nodes** panel
l. Select nodes by clicking and dragging the mouse
m. Click **Add**
n. Click **OK**
o. Click **Apply**

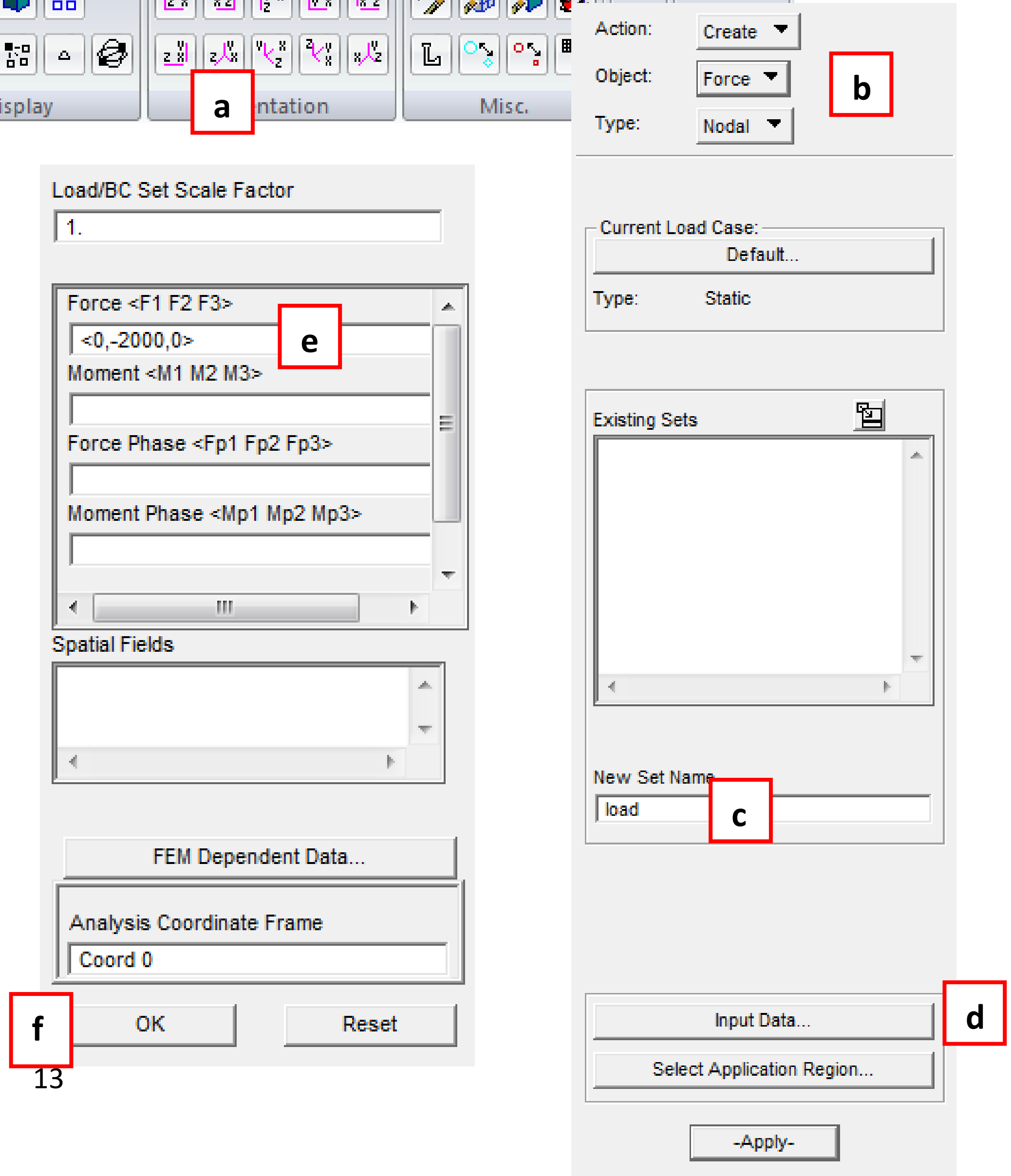


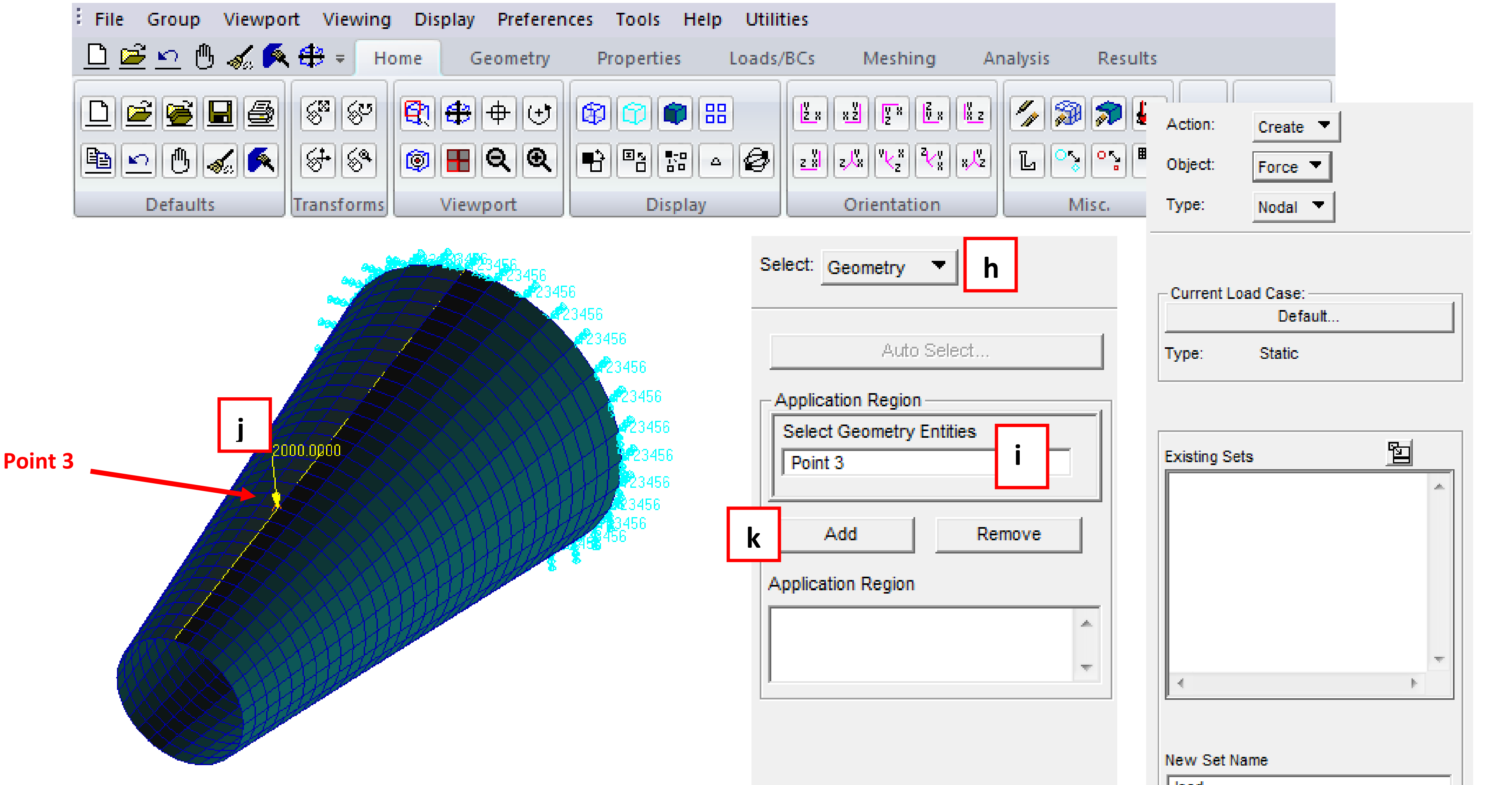


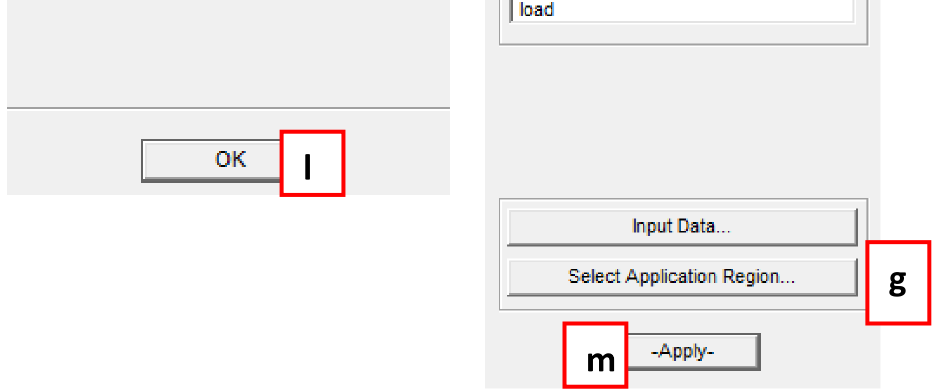


Apply the force:

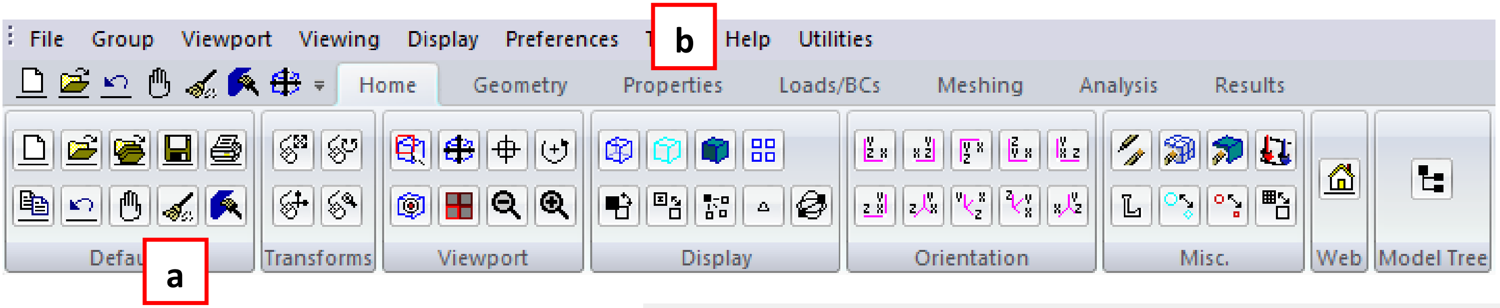
a. Click on the **Iso 1 View** icon
b. Loads/BCs: **Create/Force/Nodal**
c. Enter **load** as the New Set Name
d. Click **Input Data…**
e. Enter **<0,-2000,0>** for the Force
f. Click **OK**







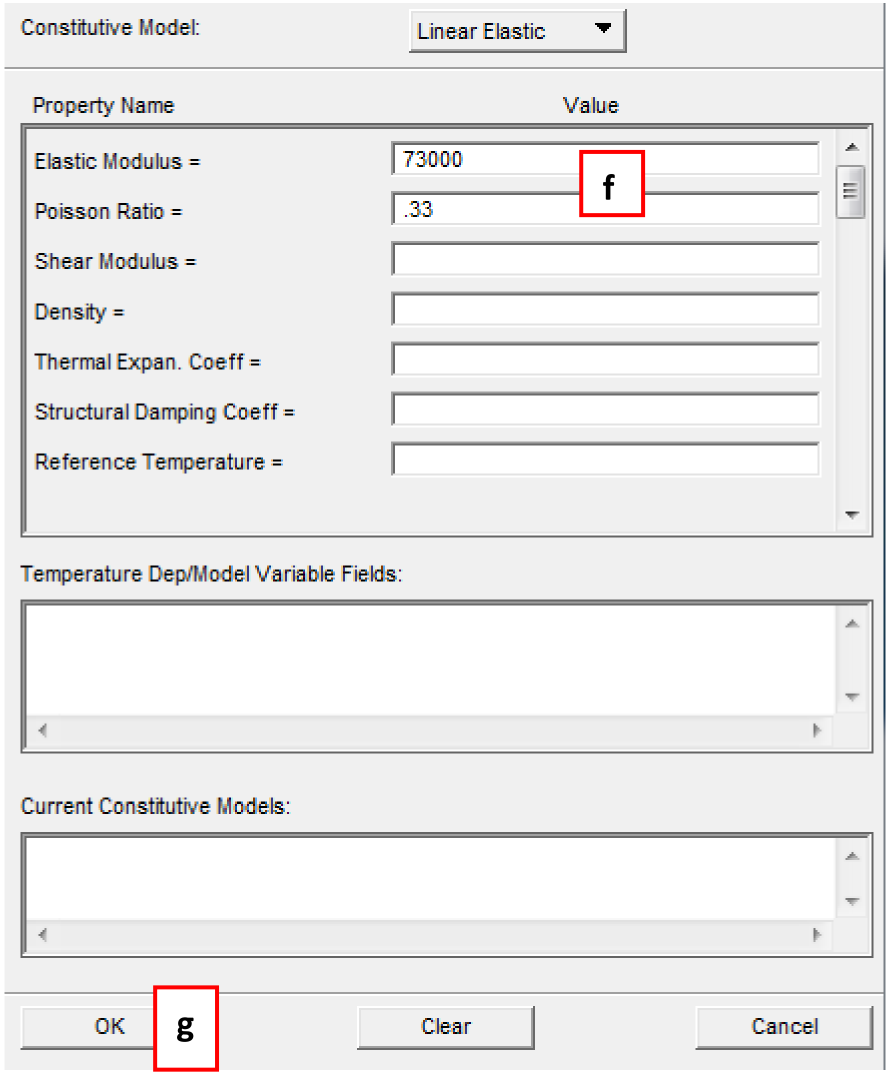
g. Click **Select Application Region…**
h. Select **Geometry**
i. Click on the **Select Geometry Entities** panel
j. Select **Point 3** (where Curve 1 connects with Curve 2)
k. Click **Add**
l. Click **OK**
m. Click **Apply**

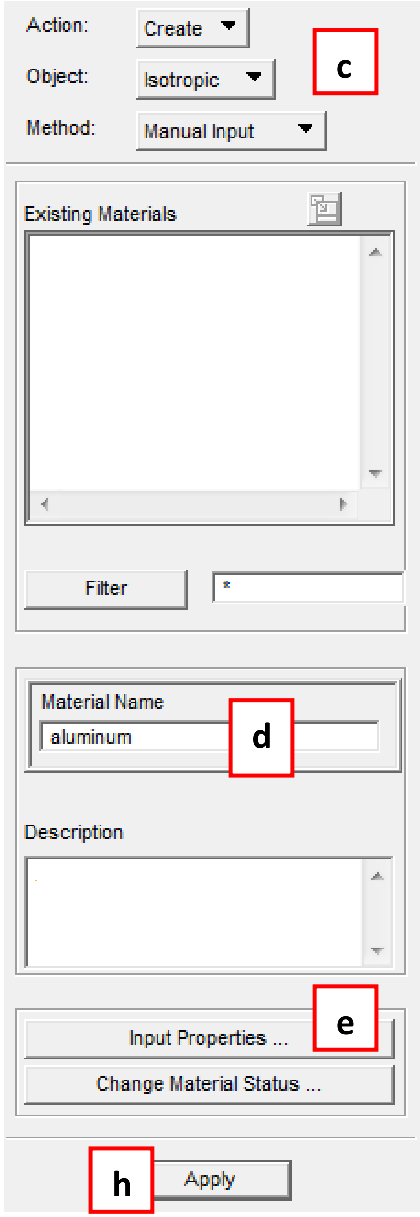


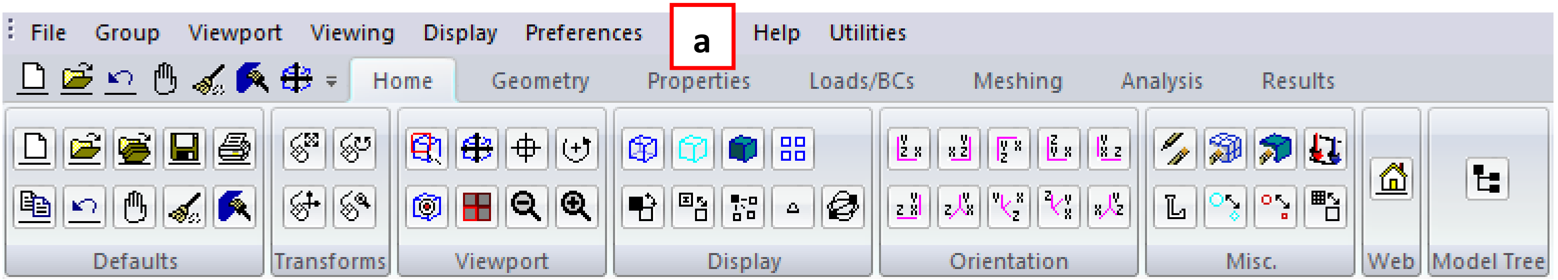
a. Click on the **Reset graphics** icon

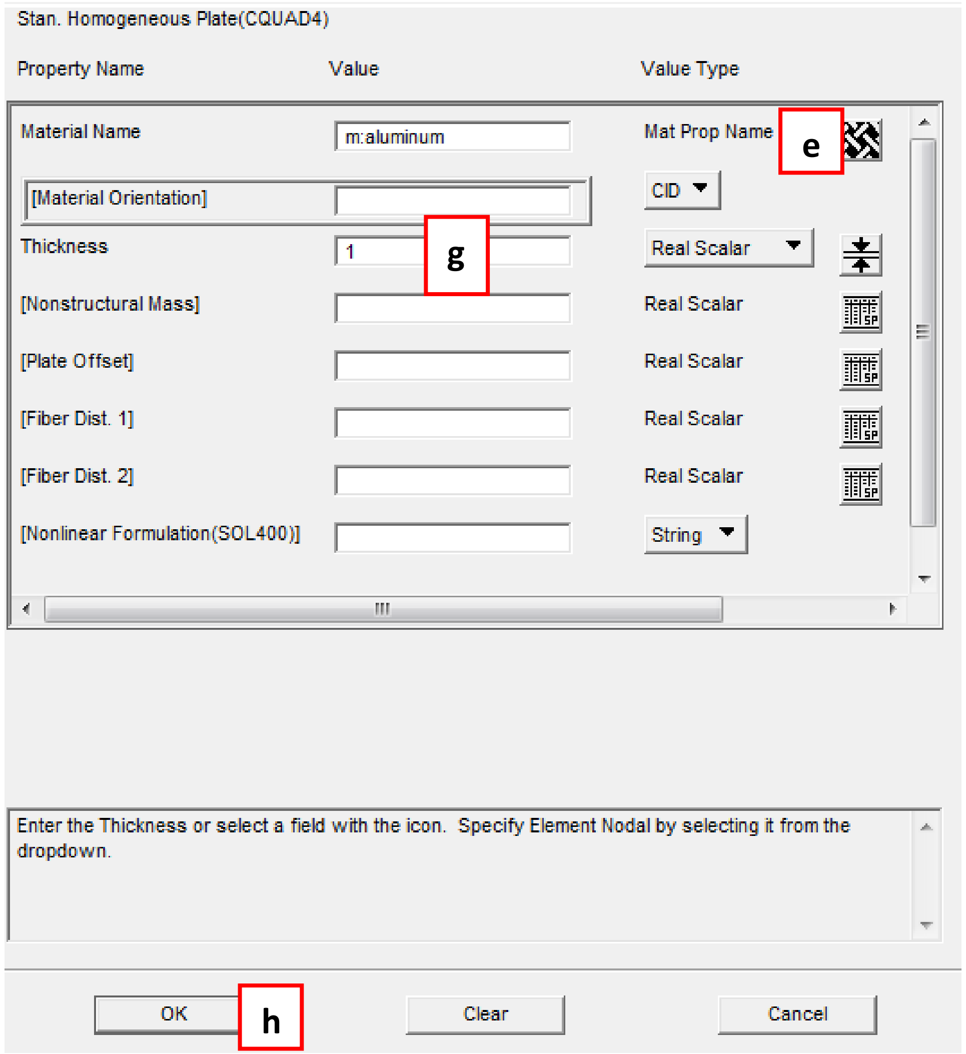
Define materials:

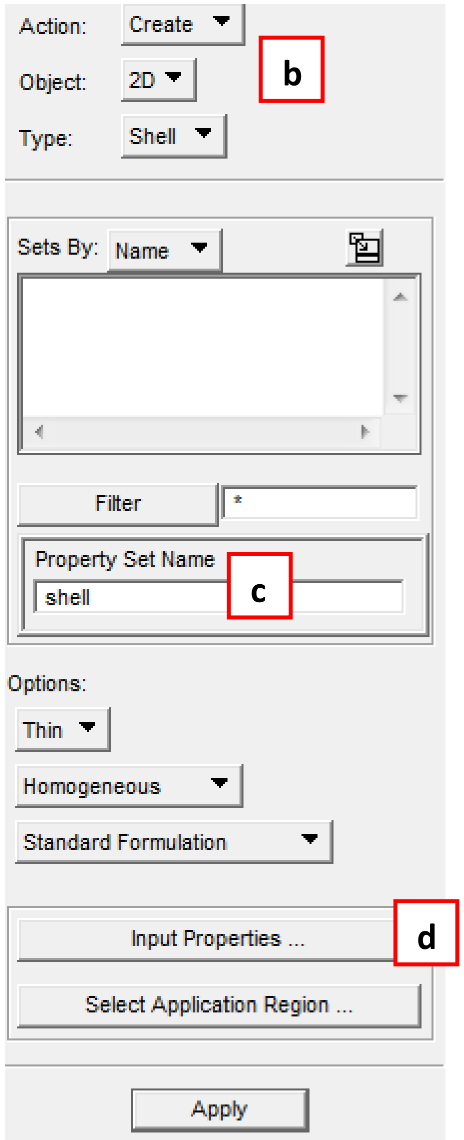
b. Click on the **Properties** tab
c. Click on **Isotropic** icon, then: **Create/Isotropic/Manual Input**
d. Enter **aluminum** as the Material Name
e. Click **Input Properties…**
f. Enter **73000** as Elastic Modulus and **0.33** as Poisson Ratio
g. Click **OK**
h. Click **Apply**
i. Repeat steps **f÷k** to define the second, relatively stiff material e.g.: **ring_mat**, **E = 2e7**, **ν = 0.29**





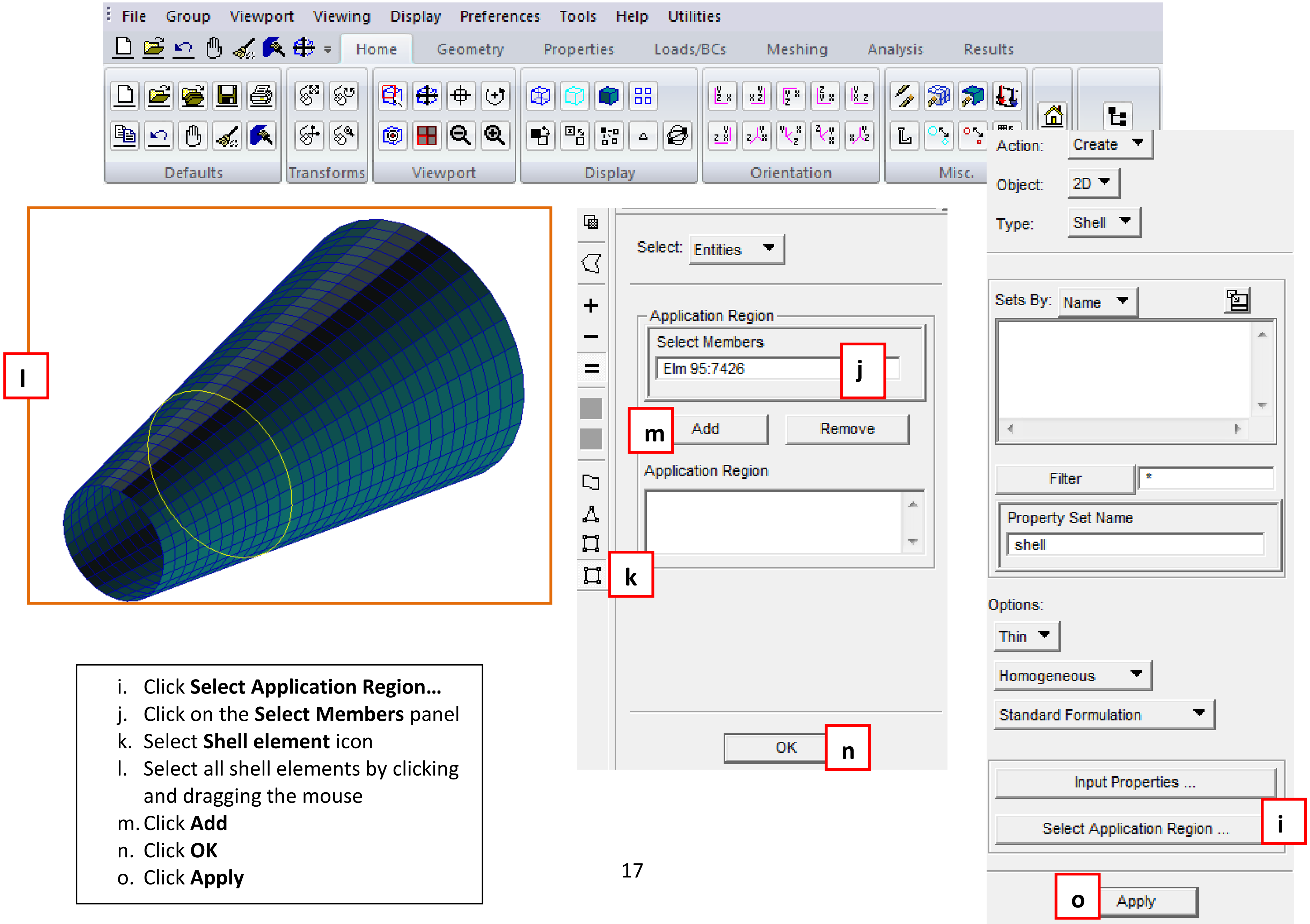






Assign the properties (shell):

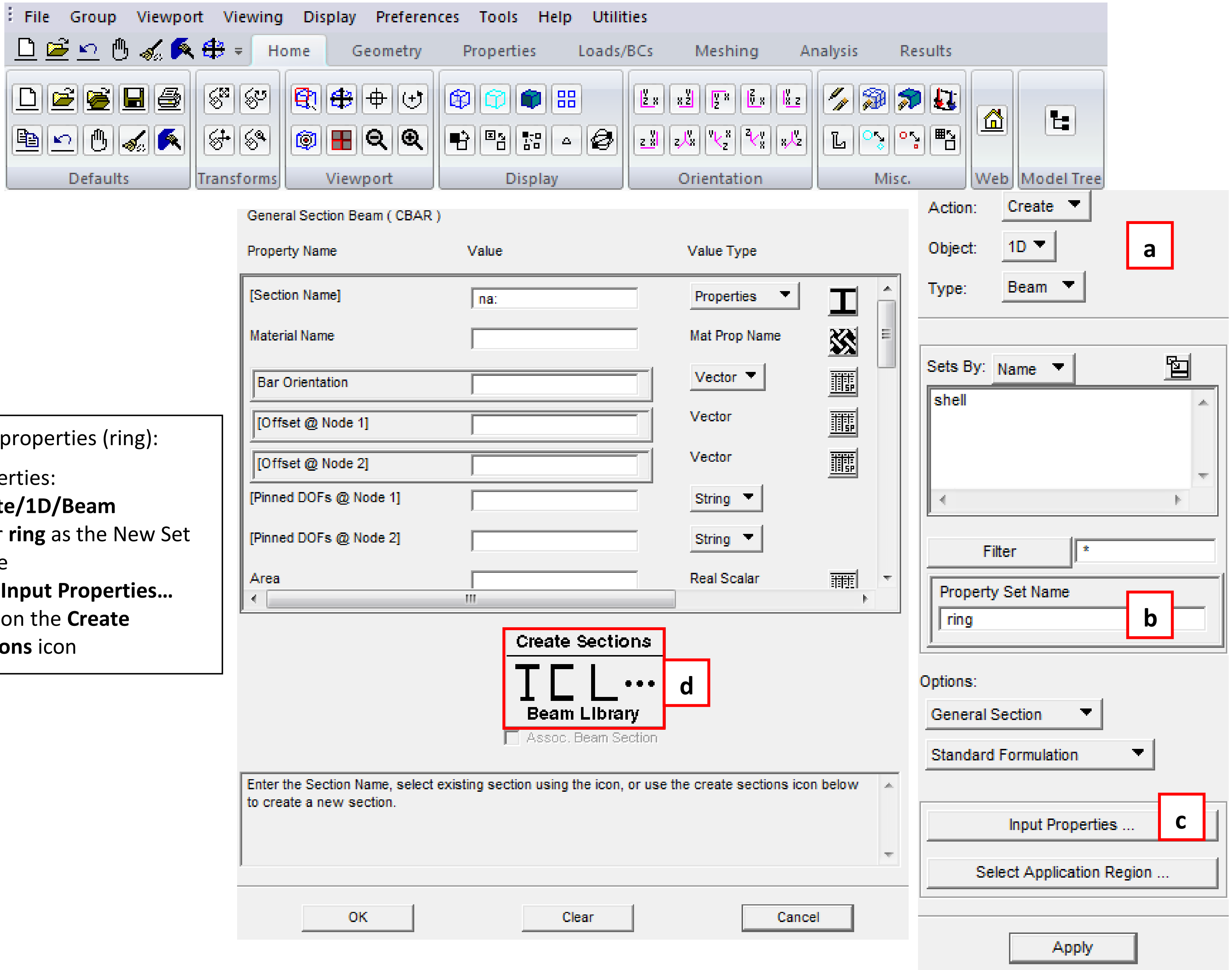
a. Double-click **Properties** tab
b. Properties: **Create/2D/Shell**
c. Enter **shell** as the New Set Name
d. Click **Input Properties…**
e. Click on the **Mat Prop Name** icon
f. Select **aluminum**
g. Enter **1** as the Thickness
h. Click **OK**

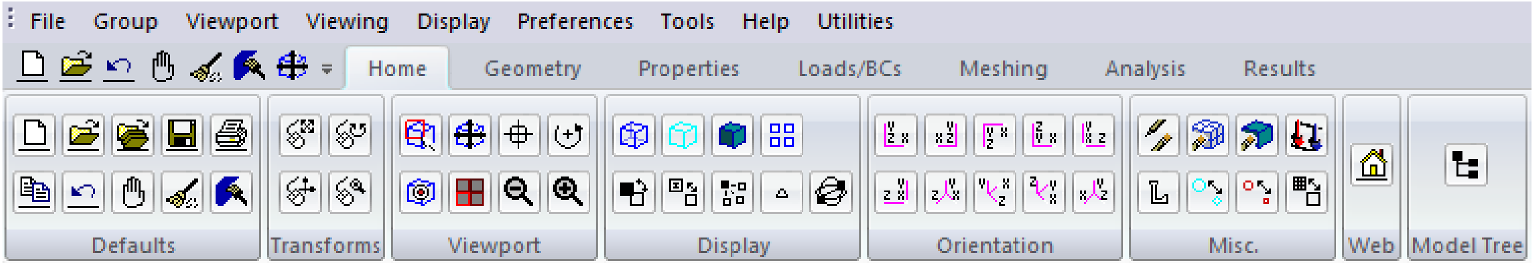


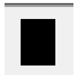
i. Click **Select Application Region…**
j. Click on the **Select Members** panel
k. Select **Shell element** icon
l. Select all shell elements by clicking and dragging the mouse
m. Click **Add**
n. Click **OK**
o. Click **Apply**

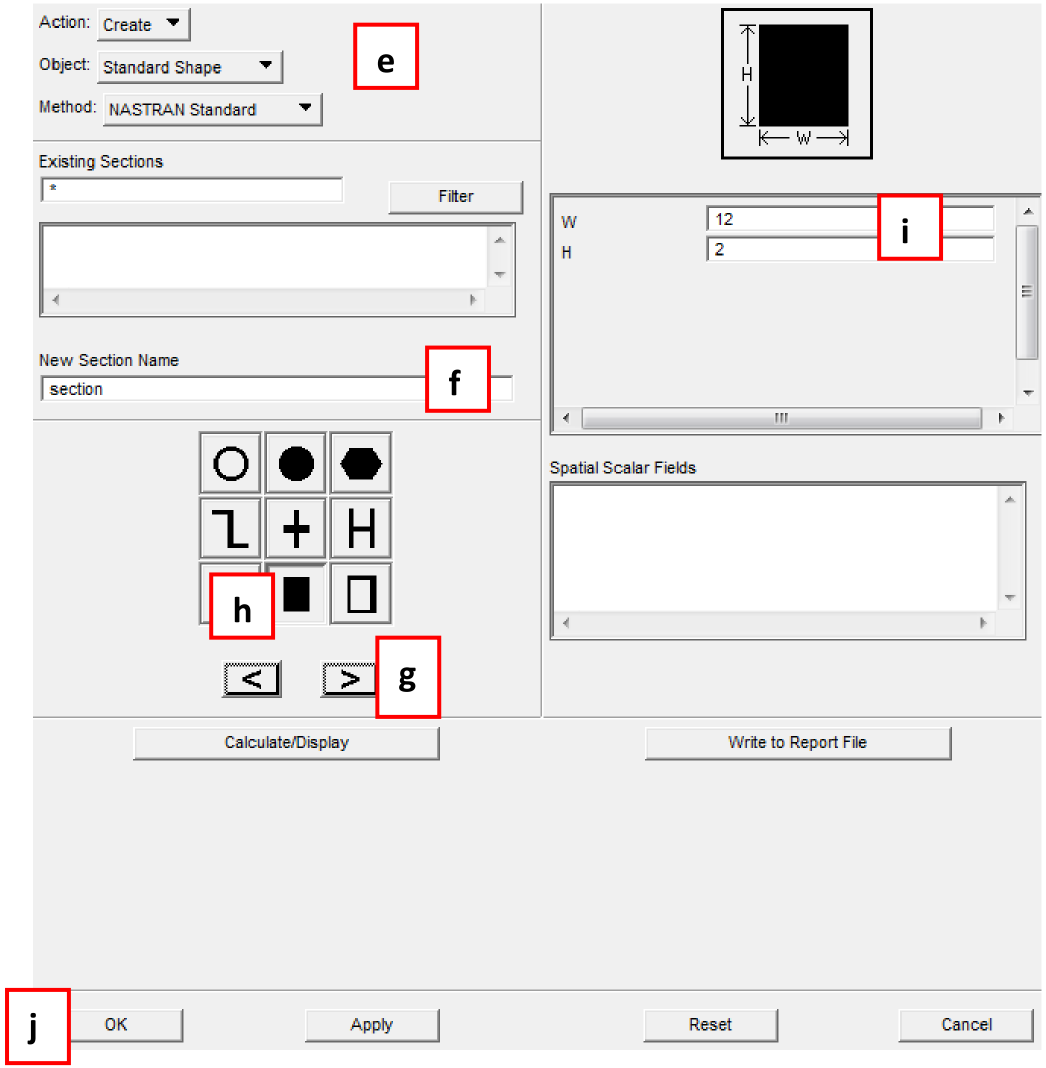
Assign the properties (ring):

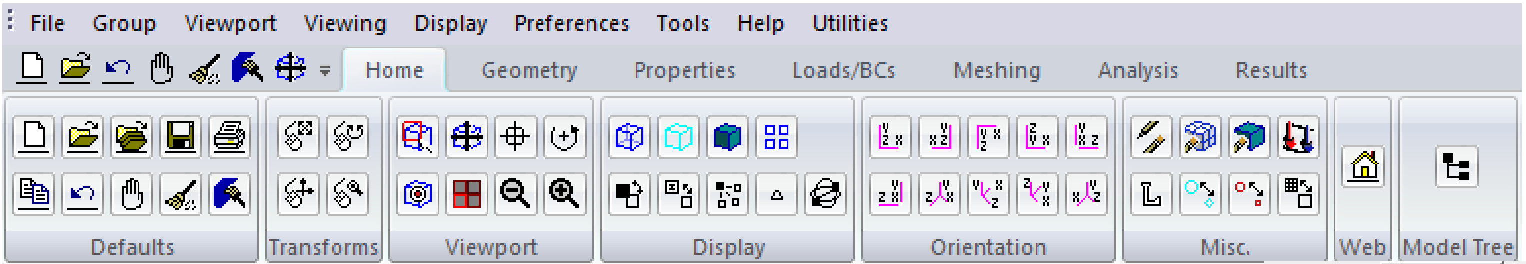
a. Properties: **Create/1D/Beam**
b. Enter **ring** as the New Set Name
c. Click **Input Properties…**
d. Click on the **Create Sections** icon

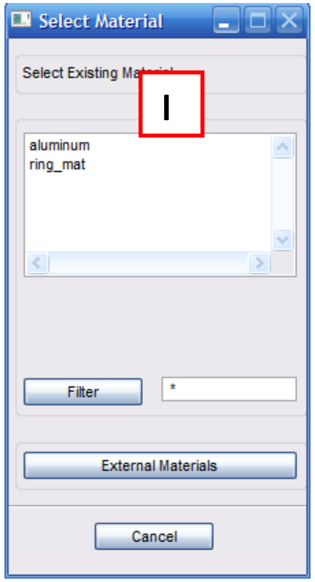


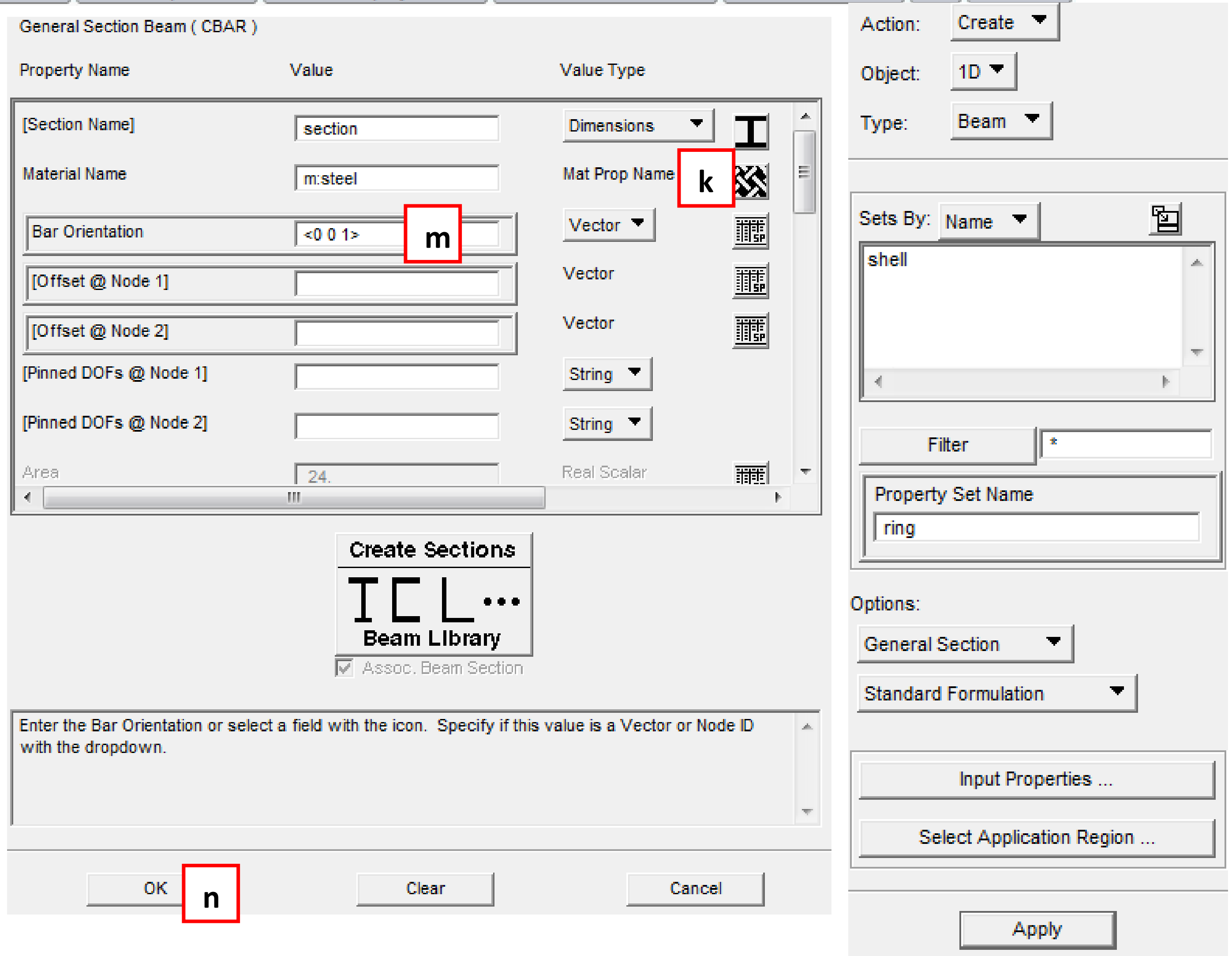


1. e. Create Sections: **Create/Standard Shape/NASTRAN Standard**
2. f. Enter **section** as the New Section Name
3. g. Click >
4. h. Click 
5. i. Enter **12** as the value of W and **2** as the value of H
6. j. Click **OK**

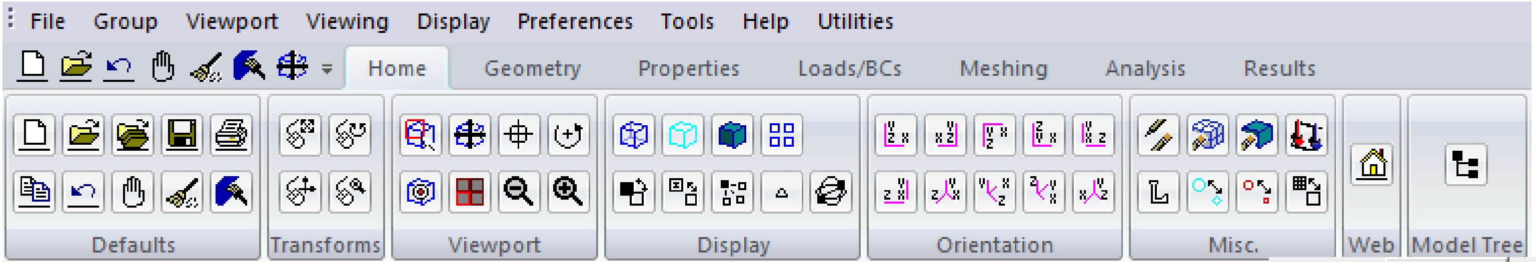


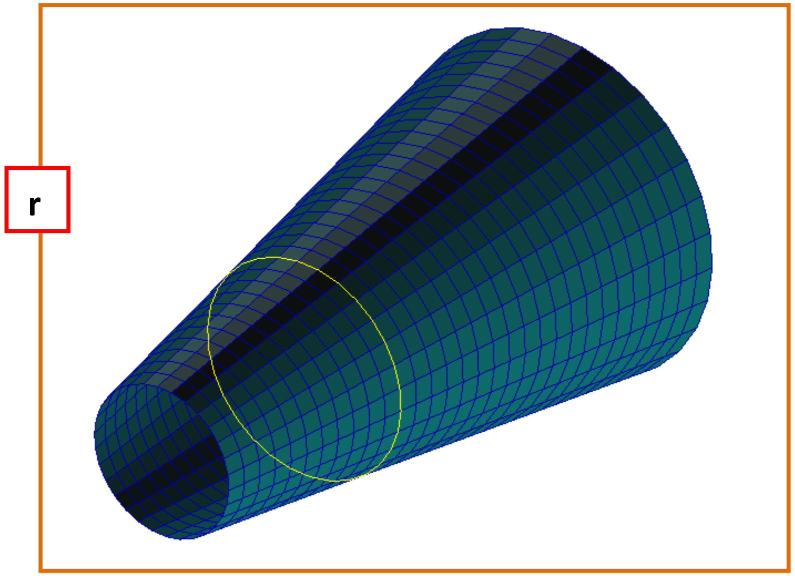


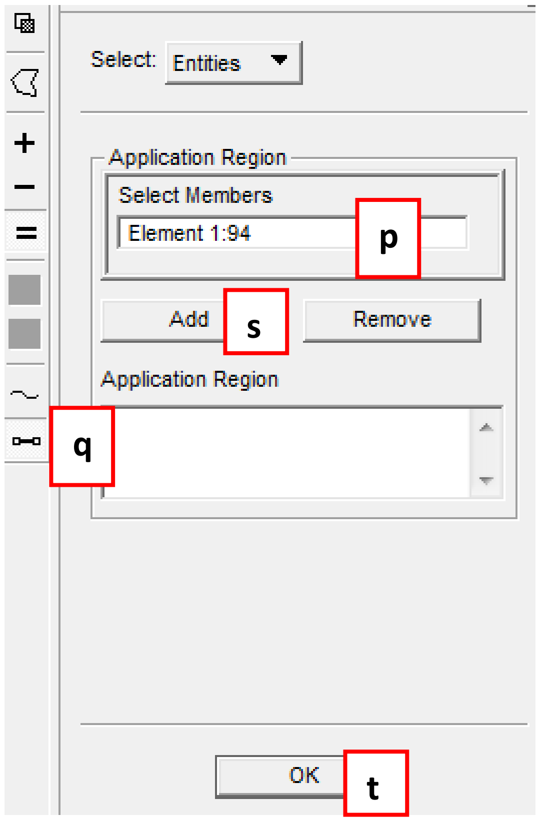




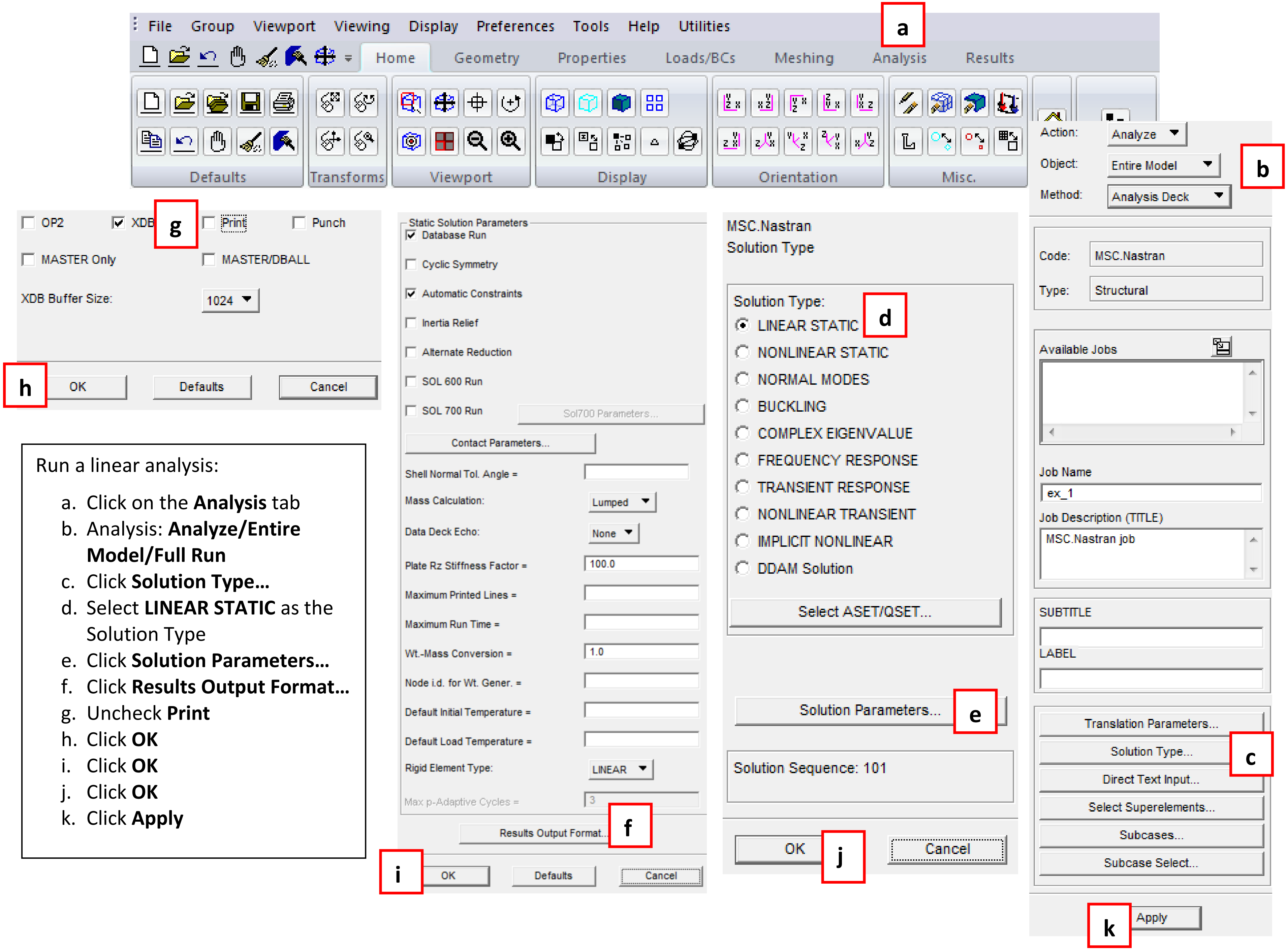
k. Click on the **Mat Prop Name** icon
l. Select **ring_mat**
m. Enter **<0 0 1>** for the Bar Orientation
n. Click **OK**





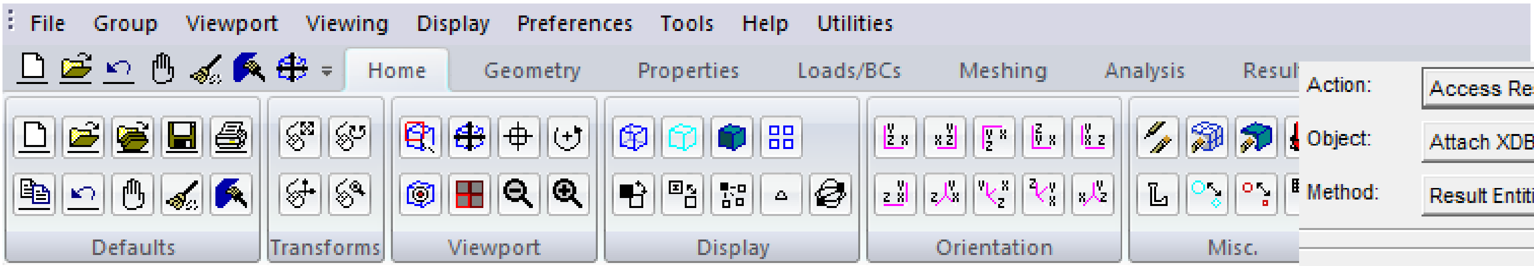


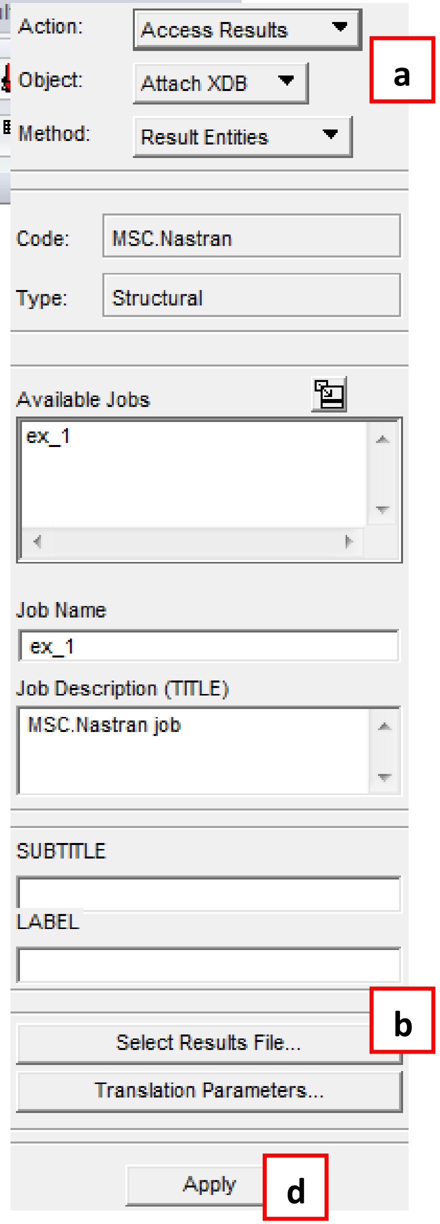
o. Click **Select Application Region…**
p. Click on the **Select Members** panel
q. Select **Beam element** icon
r. Select all beam elements by clicking and dragging the mouse
s. Click **Add**
t. Click **OK**
u. Click **Apply**



Run a linear analysis:

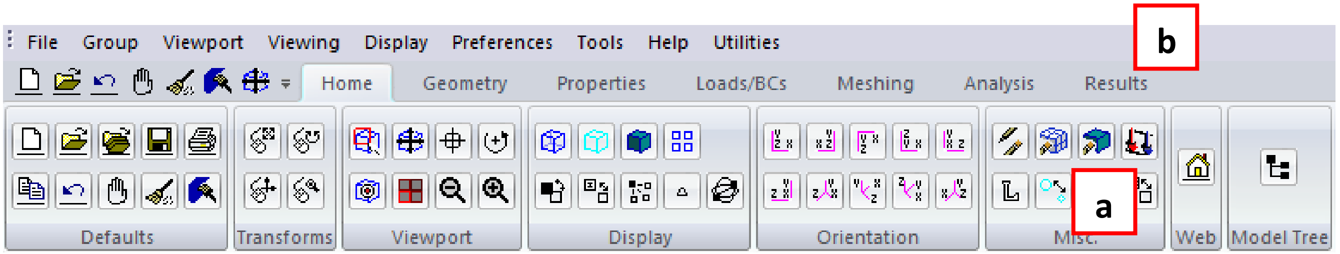
a. Click on the **Analysis** tab
b. Analysis: **Analyze/Entire Model/Full Run**
c. Click **Solution Type...**
d. Select **LINEAR STATIC** as the Solution Type
e. Click **Solution Parameters...**
f. Click **Results Output Format...**
g. Uncheck **Print**
h. Click **OK**
i. Click **OK**
j. Click **OK**
k. Click **Apply**

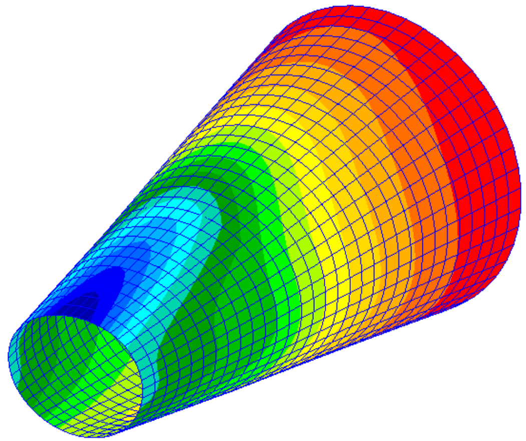




Attach the results file, when the analysis job is completed:

a. Analysis: **Access Results/Attach XDB/Result Entities**
b. Click **Select Results File…**
c. Select **ex_1.xdb** file and click **OK**
d. Click **Apply**

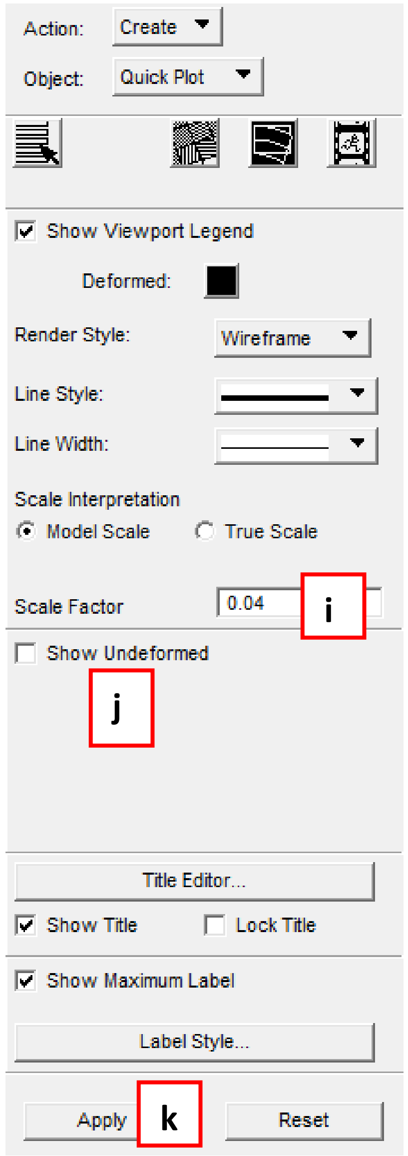


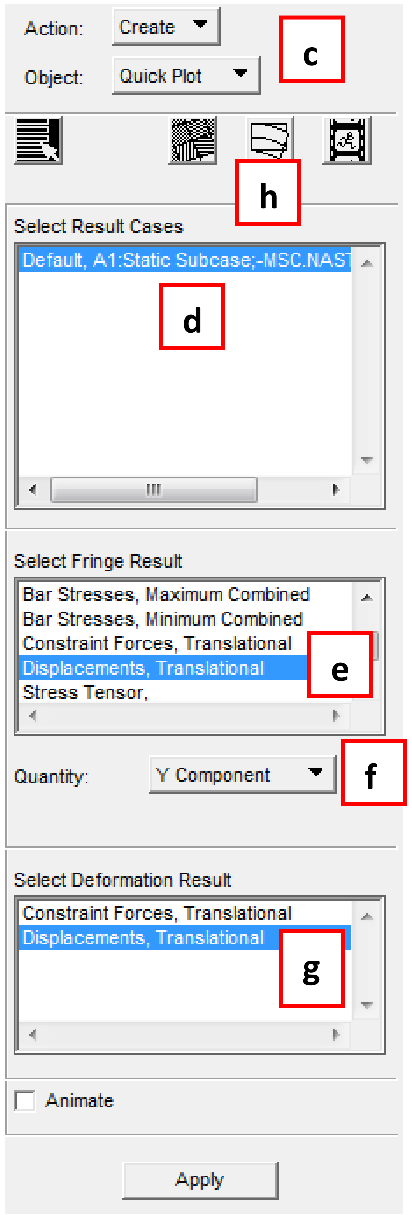


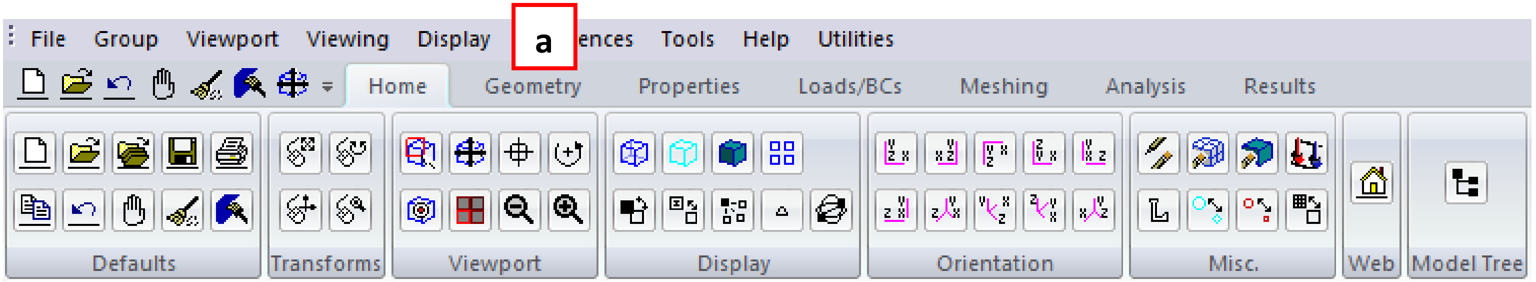
Post-process the results:

a. Click on the **Plot/Erase Geometry** icon
b. Click on the **Results** tab
c. Results: **Create/Quick Plot**
d. Select Result Cases: **Default, A1:Static Subcase**
e. Select Fringe Result: **Displacements, Translational**
f. Quantity: **Y Component**
g. Select Deformation Result: **Displacements, Translational**
h. Click on the **Deform Attributes** icon
i. Enter **0.04** as the Scale Factor
j. Uncheck **Show Undeformed**
k. Click **Apply**

Remark: To capture the plot use **File / Images…**

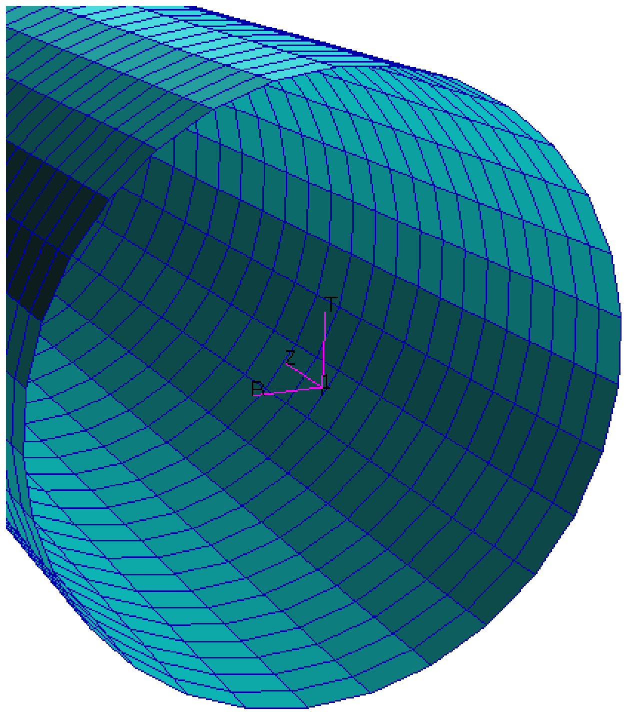


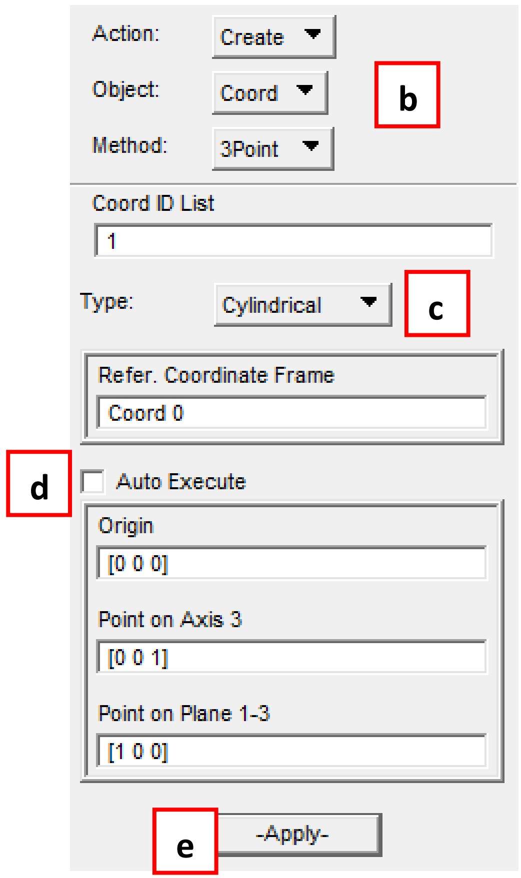


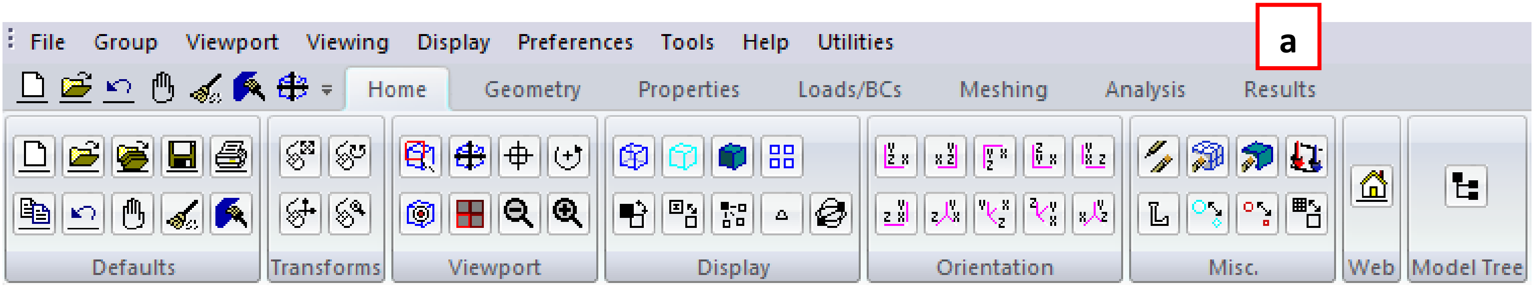


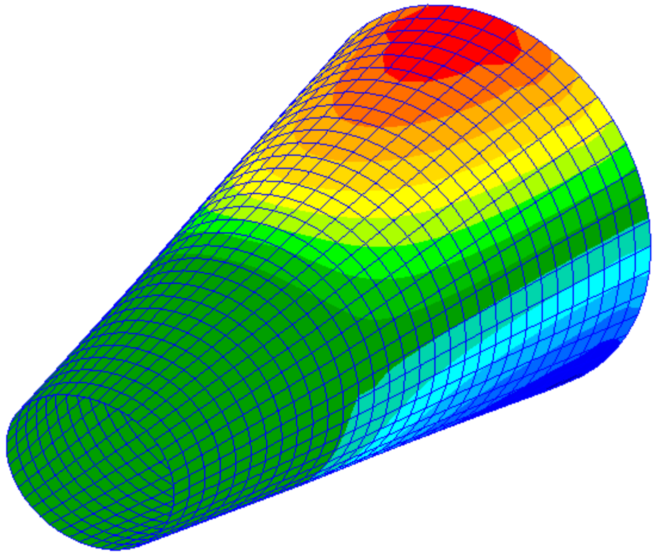
Create a cylindrical coordinate system:

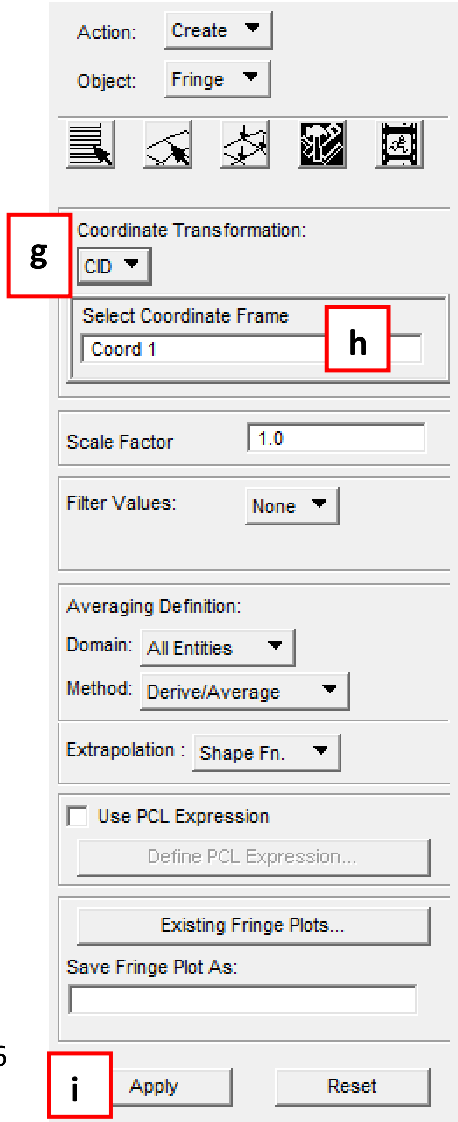
a. Click on the **Geometry** tab
b. Geometry: **Create/Coord/3Point**
c. Type: **Cylindrical**
d. Uncheck **Auto Execute**
e. Click **Apply**

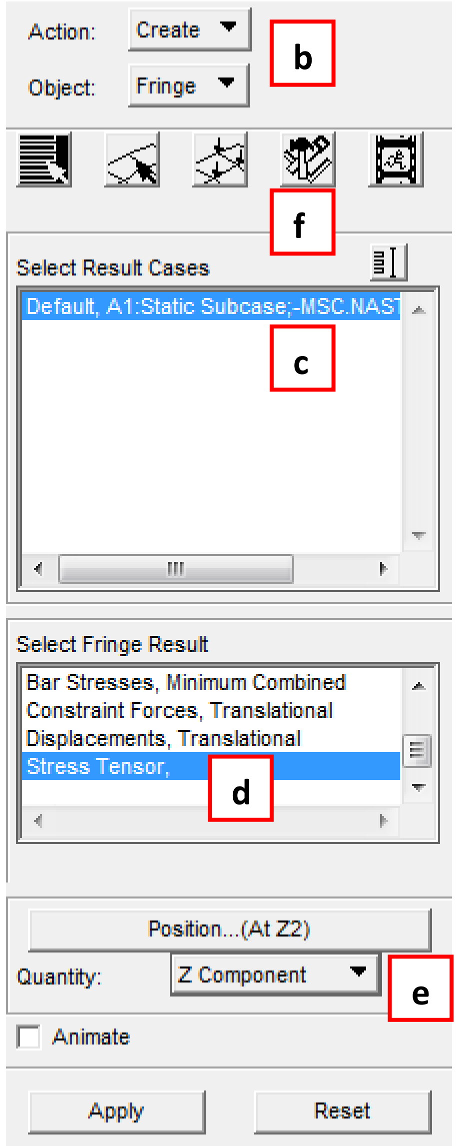






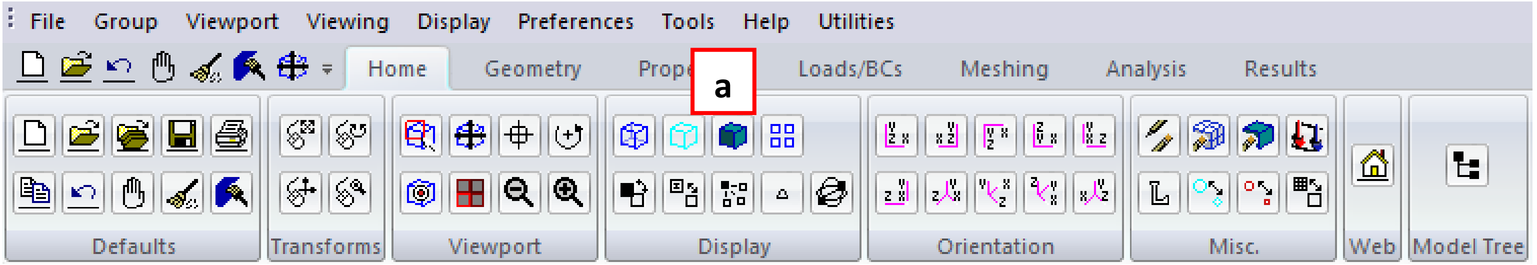


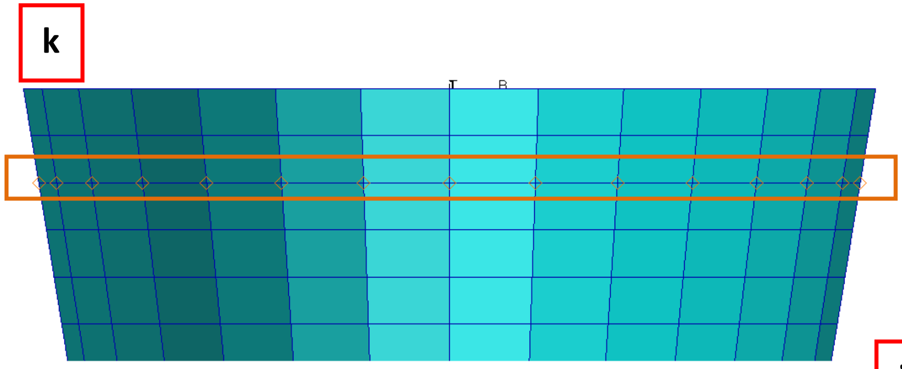


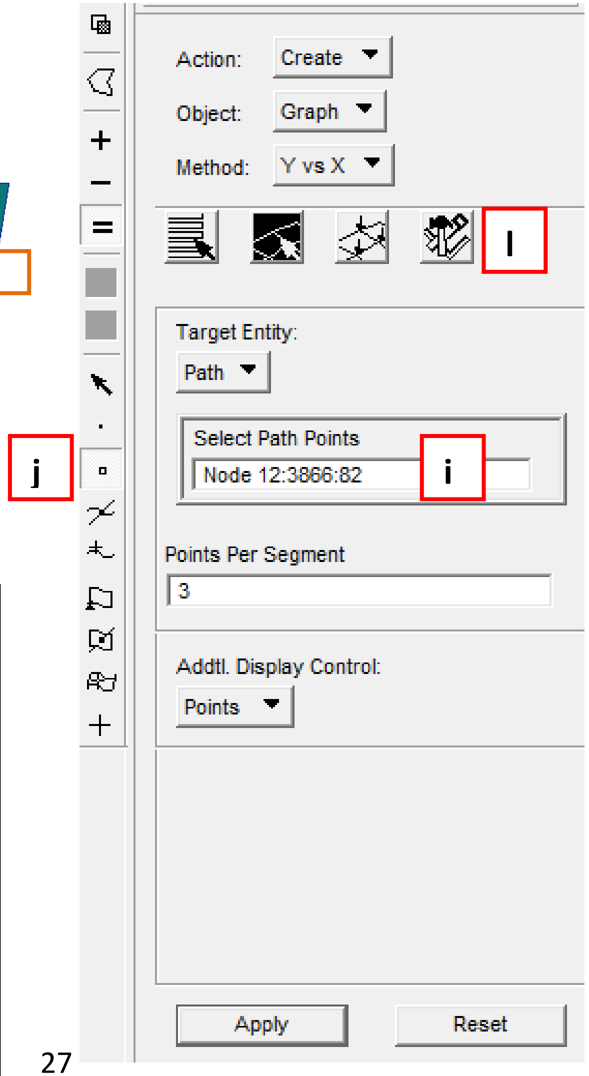


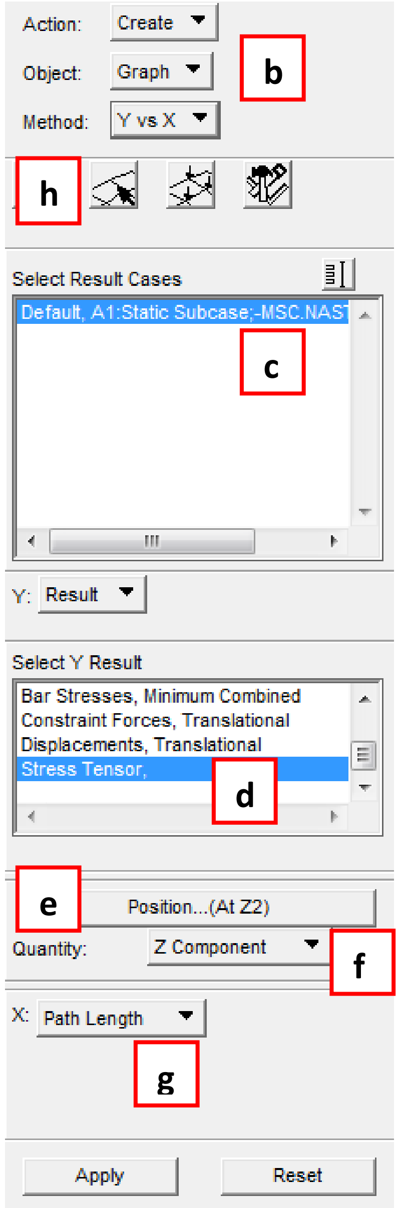
Post-process the results:

a. Click on the **Results** tab
b. Results: **Create/Fringe**
c. Select Result Cases: **Default, A1:Static Subcase**
d. Select Fringe Result: **Stress Tensor**
e. Quantity: **Z Component** (axial stress)
f. Click on the **Plot Options** icon
g. Select **CID**
h. Select the cylindrical coordinate system
i. Click **Apply**



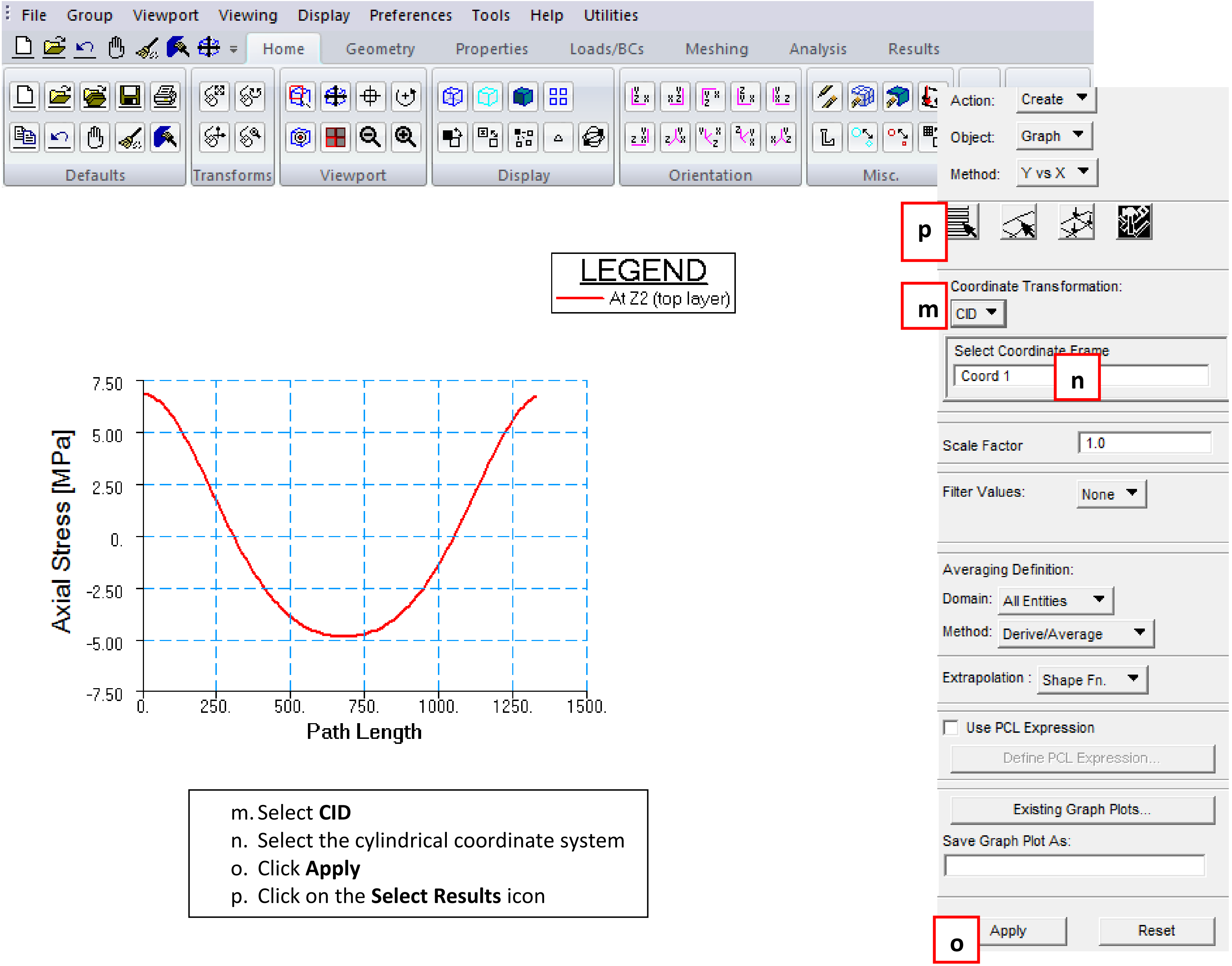




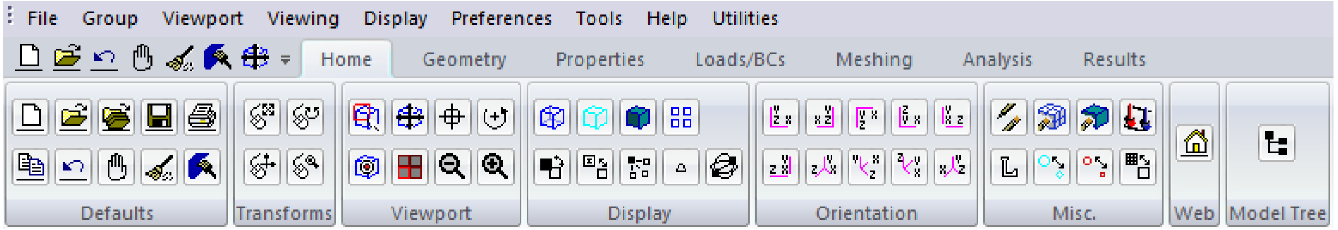


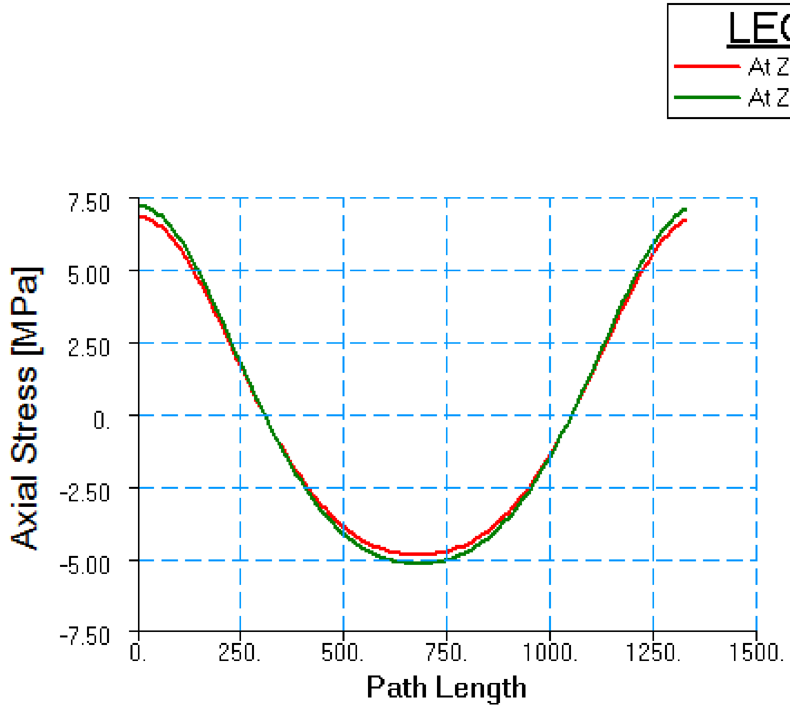
Create a graph:

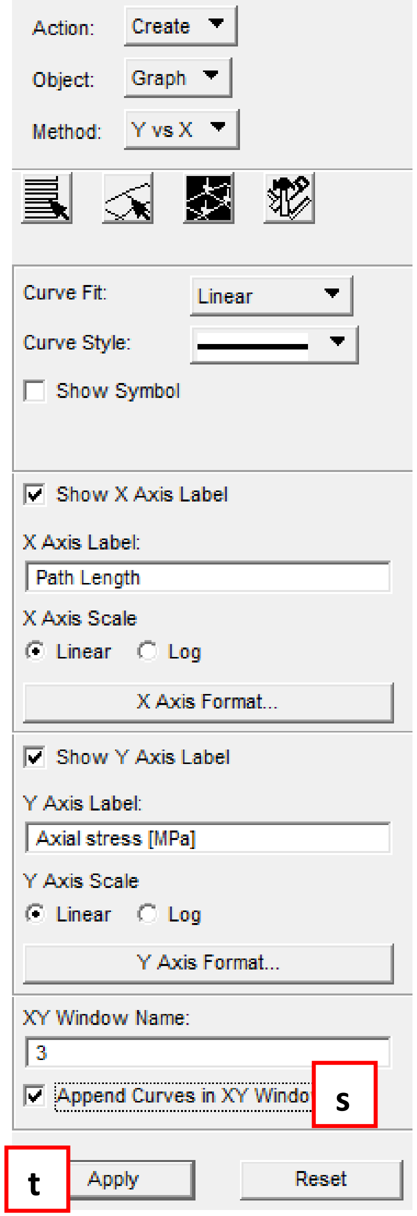
a. Click on **smooth shaded** display icon
b. Results: **Create/Graph/Y vs X**
c. Select Result Cases: **Default, A1:Static Subcase**
d. Select Fringe Result: **Stress Tensor**
e. Position: **At Z2** (top layer)
f. Quantity: **Z Component** (axial stress)
g. X: **Path Length**
h. Click on the **Target Entities** icon
i. Click on the **Select Path Points** panel
j. Select **Node** icon
k. Select nodes by clicking and dragging the mouse
l. Click on the **Plot Options** icon

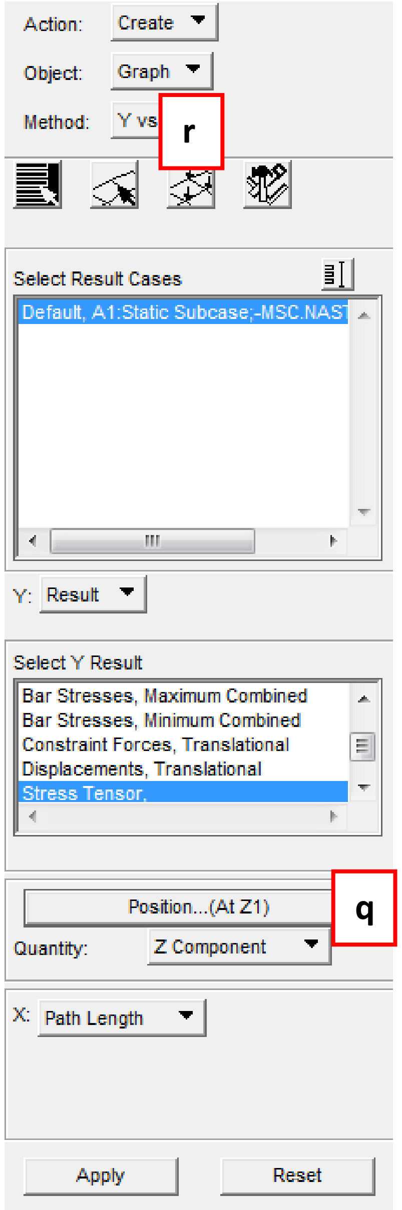


m. Select **CID**
n. Select the cylindrical coordinate system
o. Click **Apply**
p. Click on the **Select Results** icon





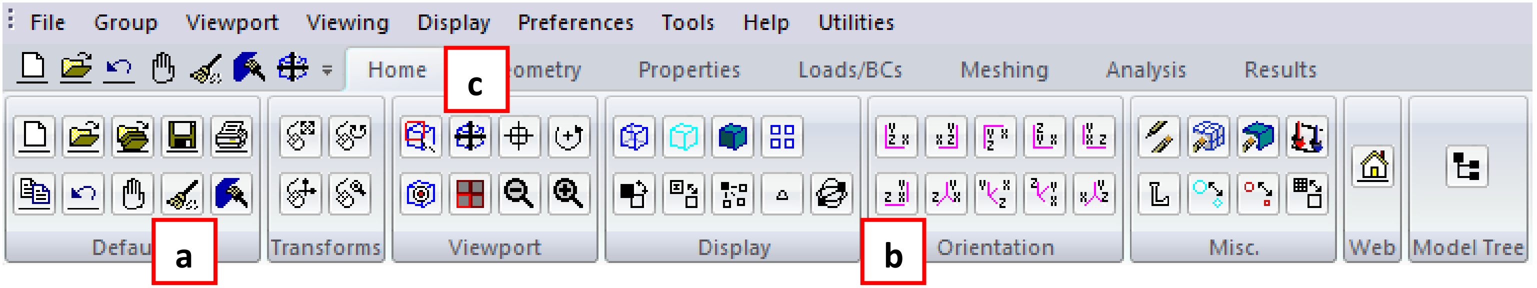




q. Change *Position* to **At Z1** (bottom layer)
r. Click on the **Display Attributes** icon
s. Check **Append Curves in XY Window**
t. Click **Apply**

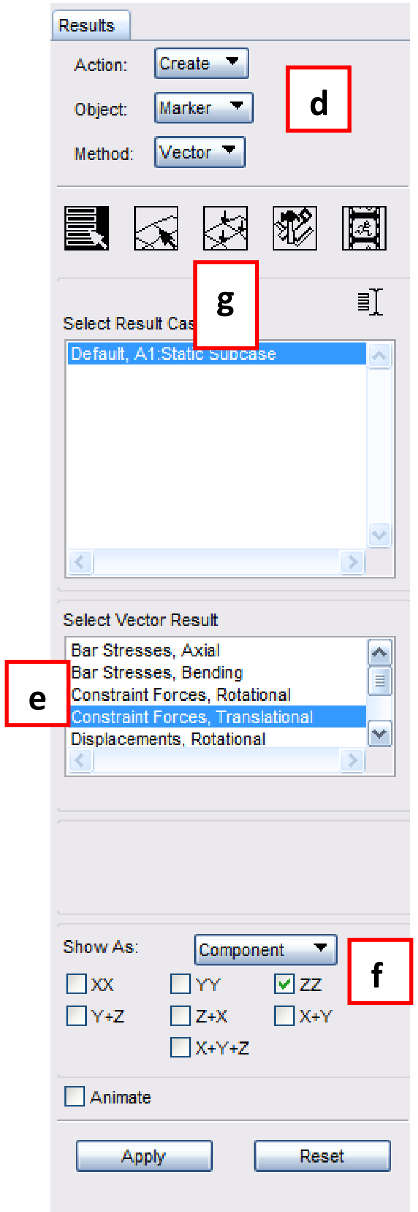
Why do results for both layers differ?

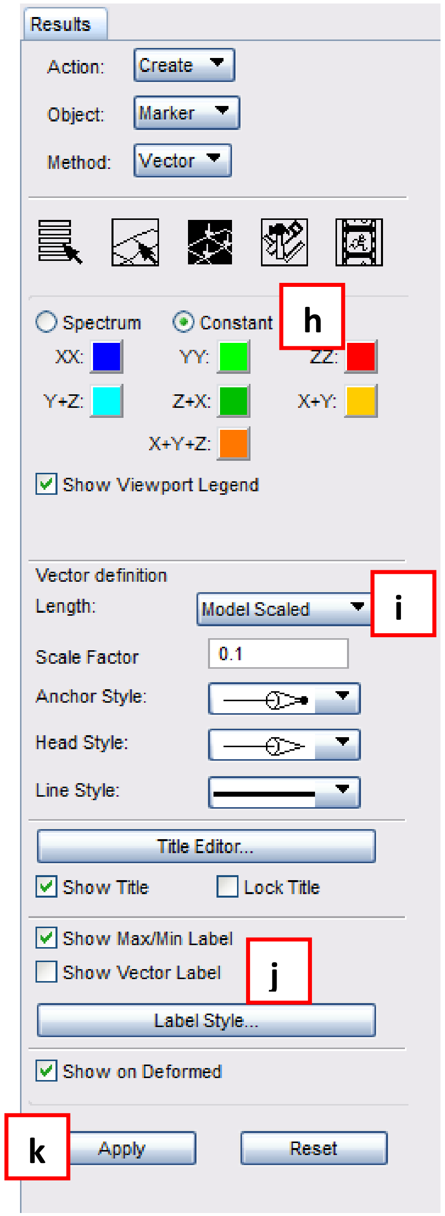
Compare foregoing graphs with the constraint forces distribution from the next slide

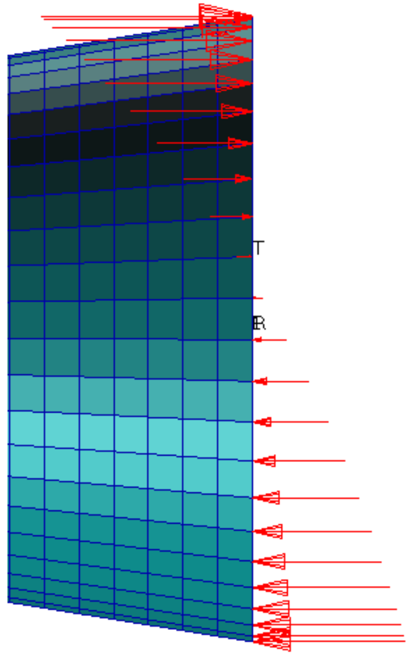


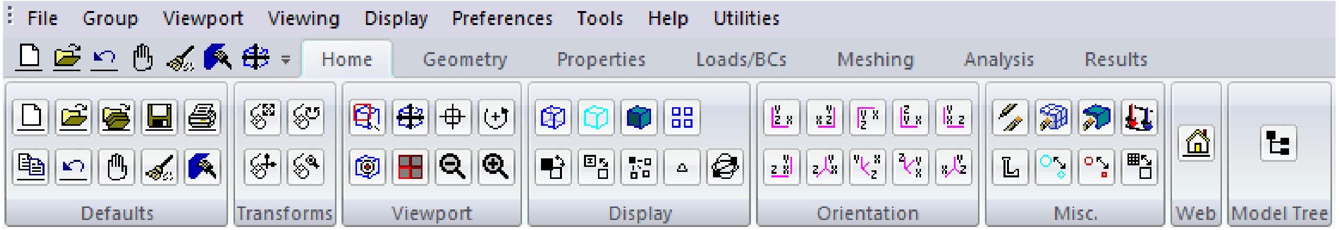
Compare obtained graph with vector results:

a. Click on the **Reset graphics** icon
b. Select **Right side view**
c. **Fit** window
d. Click on **Results/Create/Marker/Vector**
e. Select **Constraint Forces, Translational** for the Vector Result
f. Change option to **Component** and toggle on only **ZZ** option
g. Click on **Display Attributes**
h. Toggle **Constant** vector color option and choose color for ZZ component
i. Change **Length** to **Model Scaled**
j. Toggle off **Show Vector Label**
k. Click **Apply**







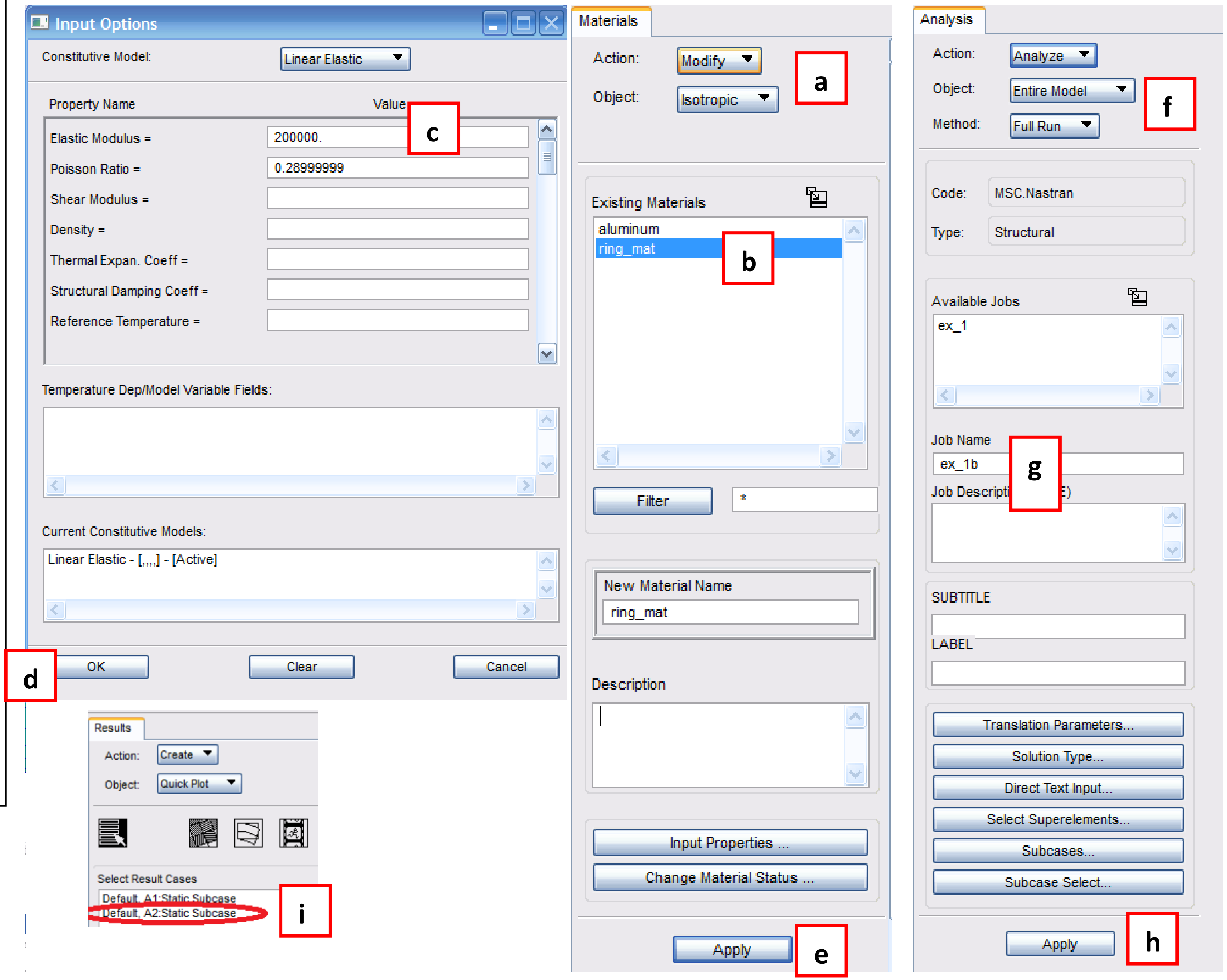


ANALYSIS NO. 2

- change of the material properties

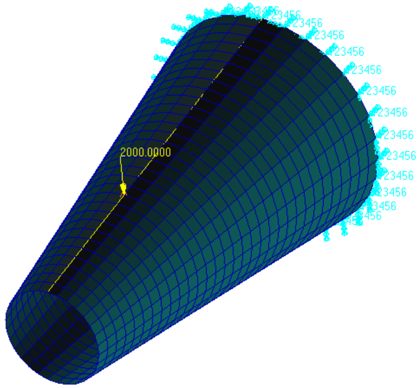
Change ring material properties, run second analysis and compare results with recently obtained:

a. Go to **Properties** tab and select **Modify/Isotropic**
b. Choose **ring_mat**
c. Change Young's Modulus to **2e5** (note that previous material was 100 times stiffer!)
d. Click **OK**
e. Click **Apply**
f. In the **Analysis** tab, select **Analyze/Entire Model/Full Run**
g. Change **Job Name** to **ex_1b**
h. Select **Apply**
i. After analysis is done, ***attach* ex_1b.xdb** file and ***plot results*** in the same manner as previously. Results refering to the second analysis will be tagged as **A2: Static subcase**



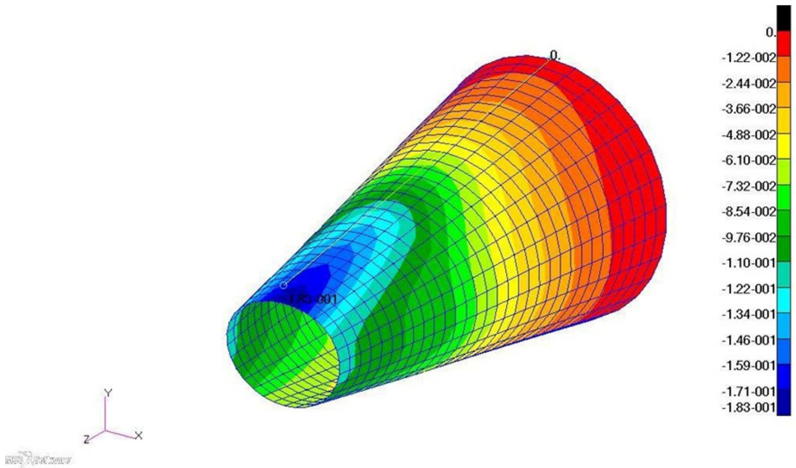
LIST OF REQUIRED PLOTS (total number of plots = <u>10</u>):

1) FE MODEL WITH LOADS AND BOUNDARY CONDITIONS (1 plot)

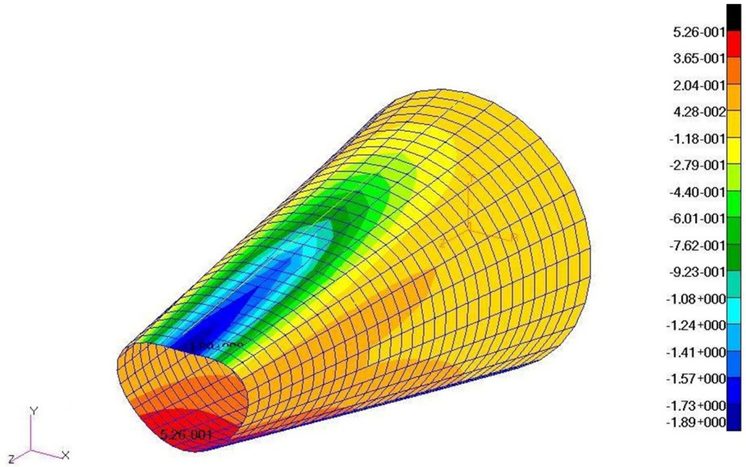


Use this reference to verify the results of both analyses.

2) VERTICAL TRANSLATIONAL DISPLACEMENT IN Y DIRECTION (2 plots)
- for 1st and 2nd analysis separately

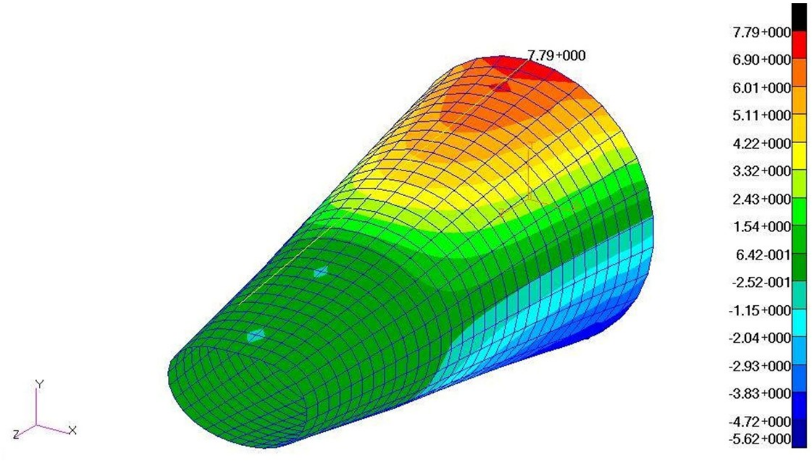


Analysis no. 1

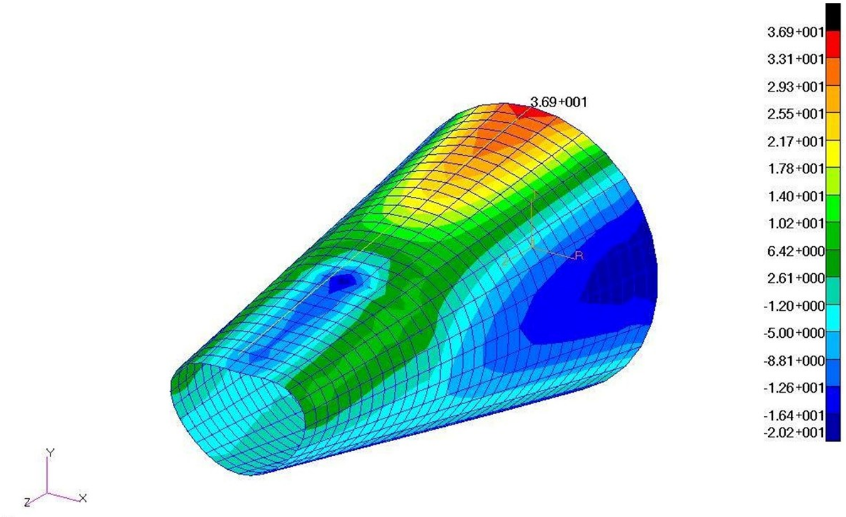


Analysis no. 2

3) AXIAL STRESS "sigma_z" in cylindrical CS (2 plots)
- for 1st and 2nd analysis separately

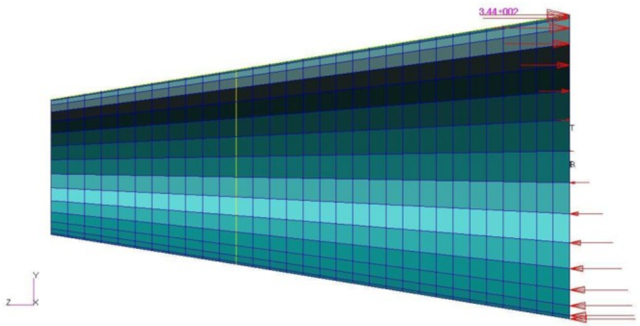


Analysis no. 1

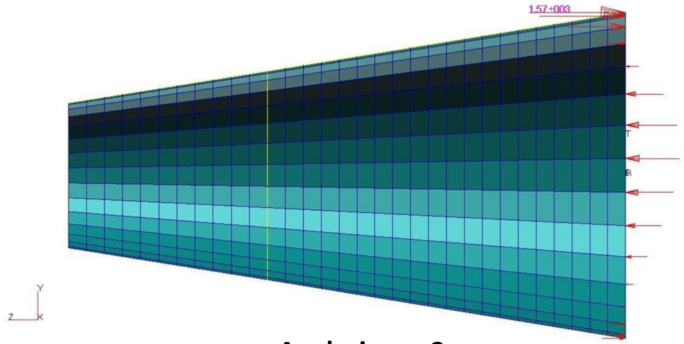


Analysis no. 2

4) CONSTRAINT FORCES, TRANSLATIONAL, COMPONENT ZZ PLOT (2 plots)
- for 1st and 2nd analysis separately



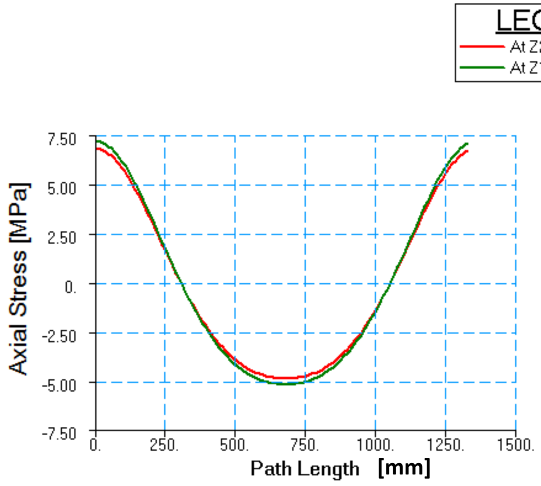
Analysis no. 1



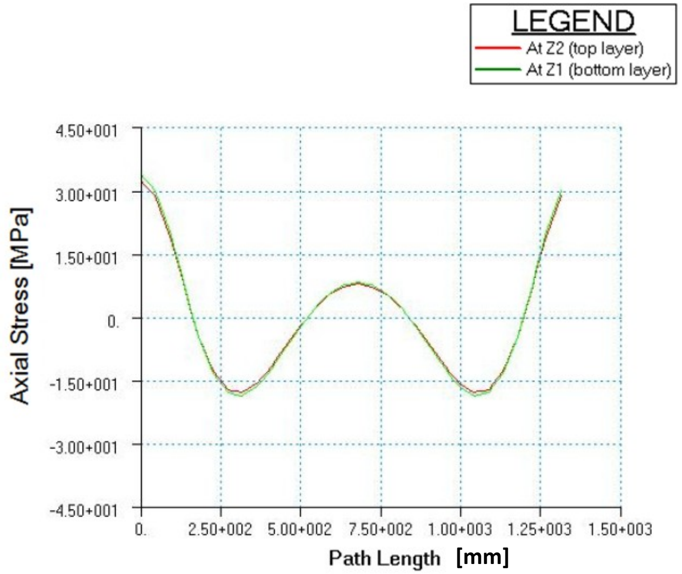
Analysis no. 2

5) YX GRAPH OF AXIAL STRESS "sigma_z" on top and bottom layer in cylindrical CS (2 plots)

- for 1st and 2nd analysis separately



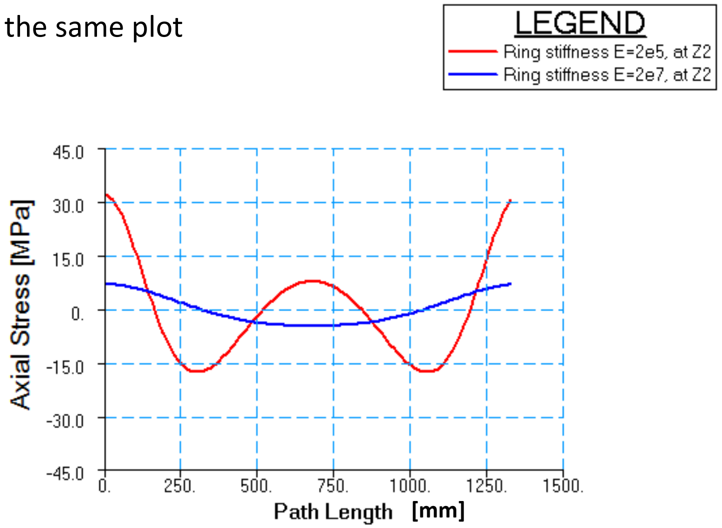
Analysis no. 1



Analysis no. 2

6) YX GRAPH OF AXIAL STRESS "sigma_z" on top OR bottom layer in cylindrical CS (1 plot)

- curve from 1st and curve from 2nd analysis on the same plot



What qualitative changes can be noticed?

Which calculation yield results more similar to the theoretical solution? Why?