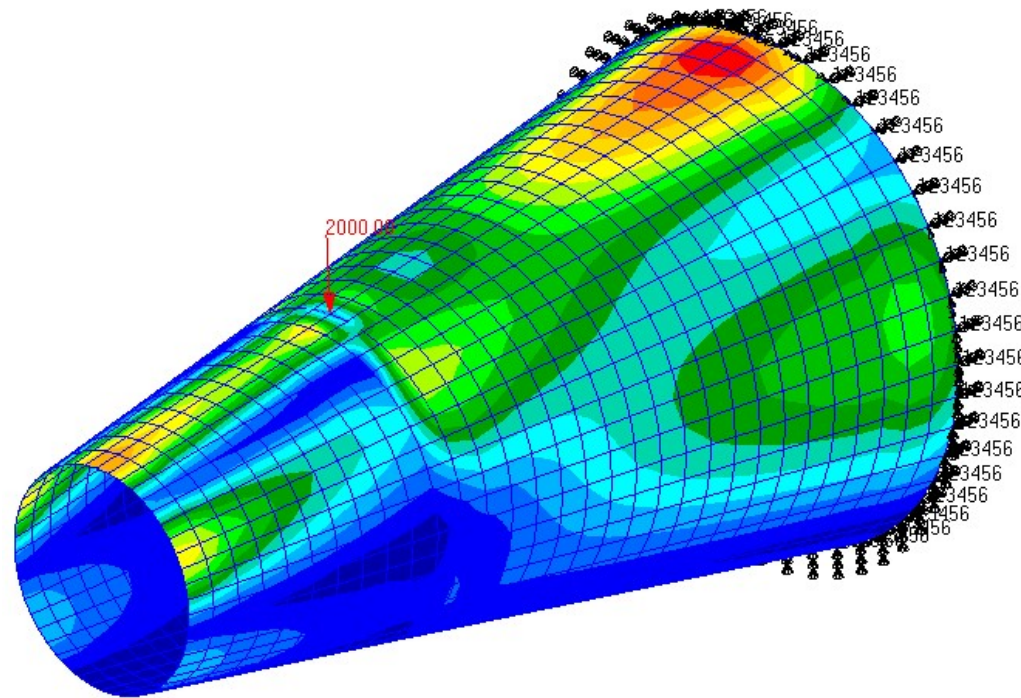
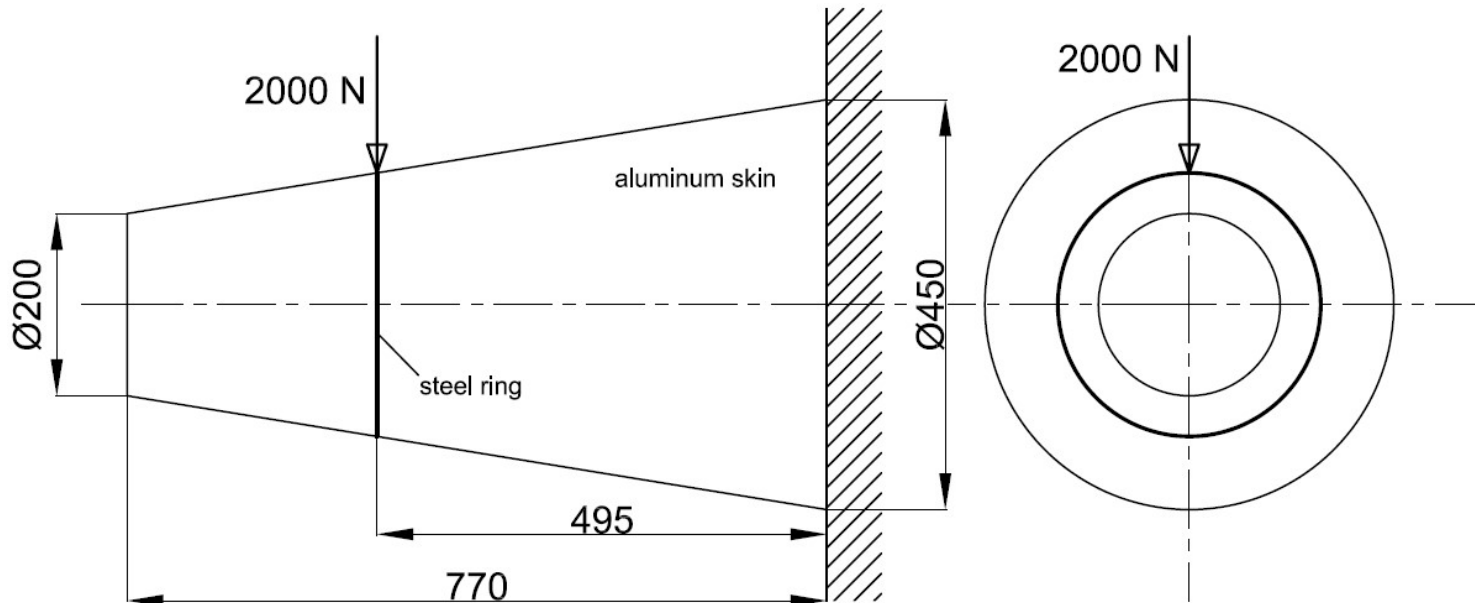


# Mechanics of Thin-walled Structures

## EXERCISE 1



# PROBLEM DESCRIPTION



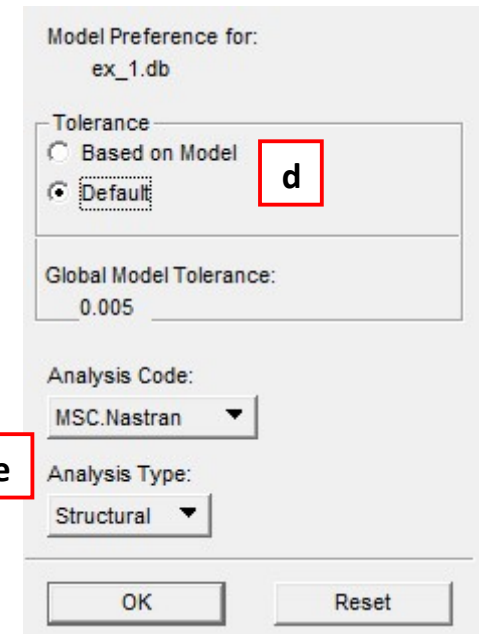
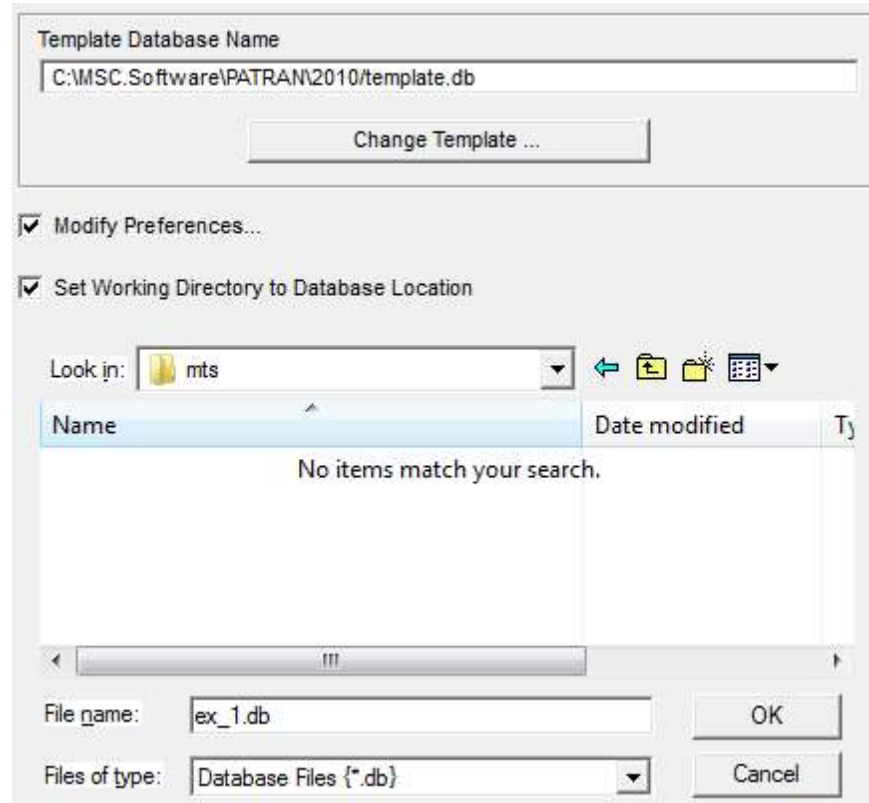
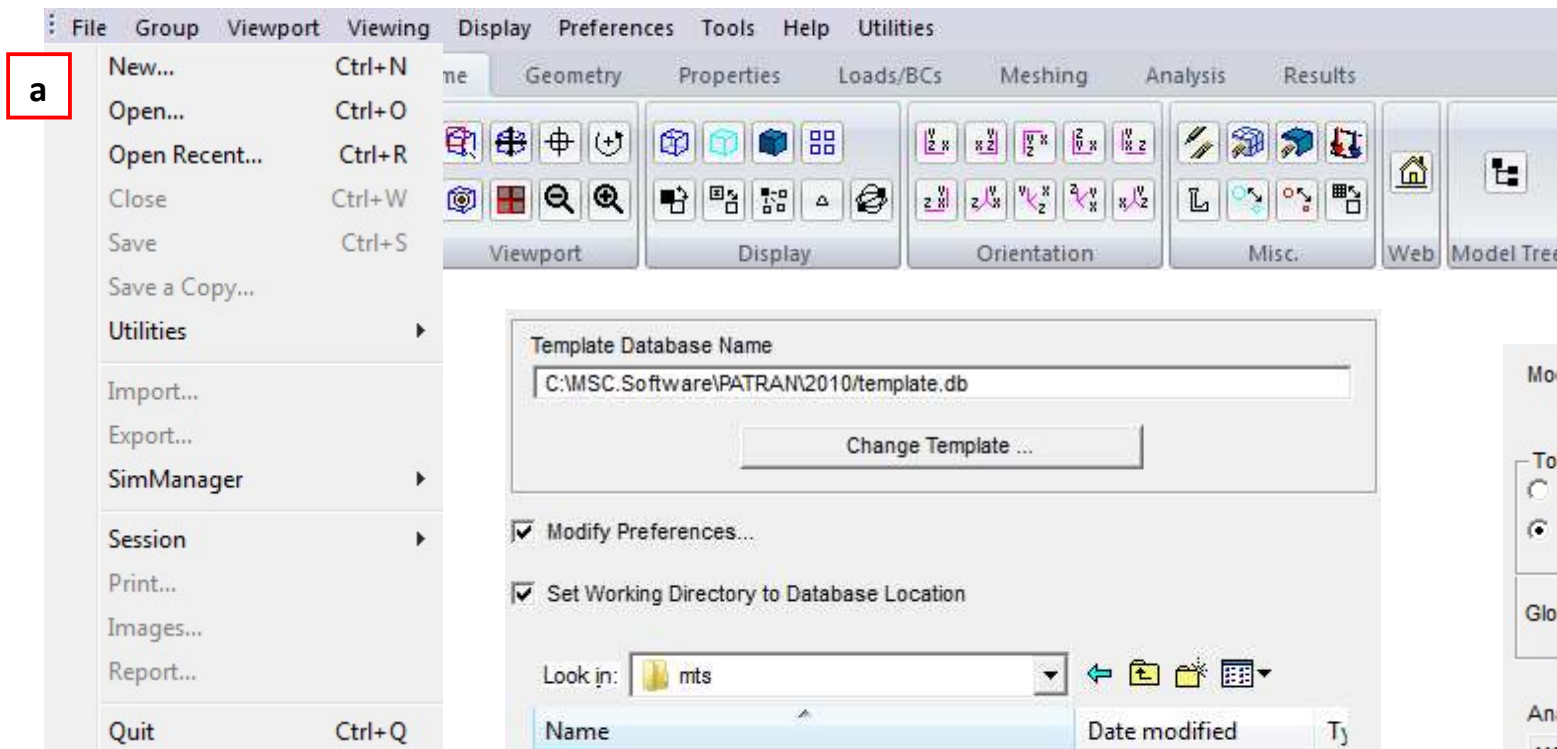
**Static, linear analysis** of thin-walled aluminum skin, reinforced by a stiffening ring, will be performed.

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.**

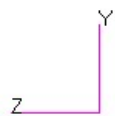
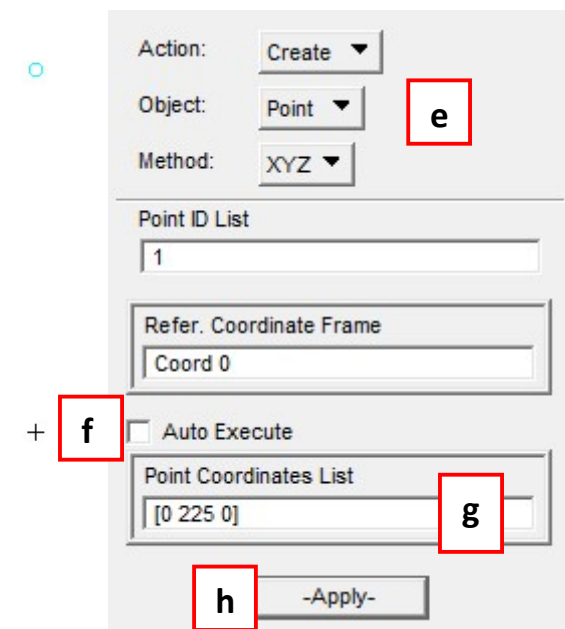
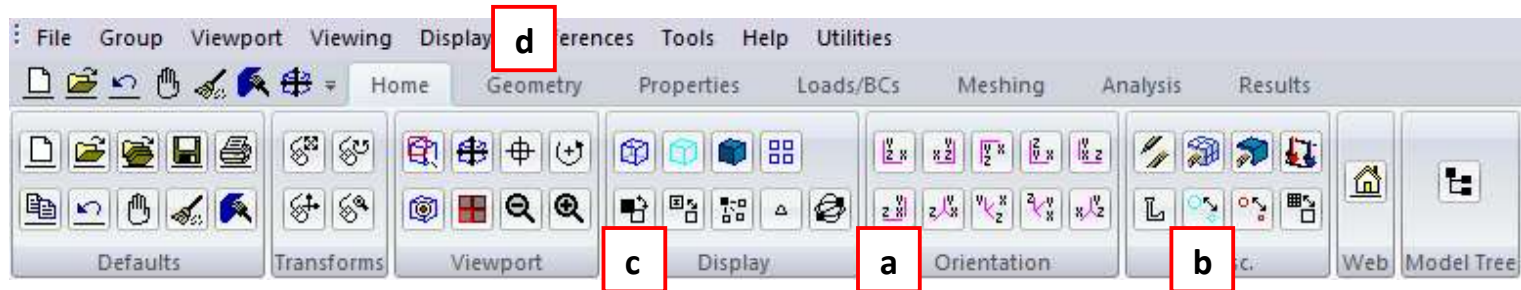
Two cases will be considered:

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**

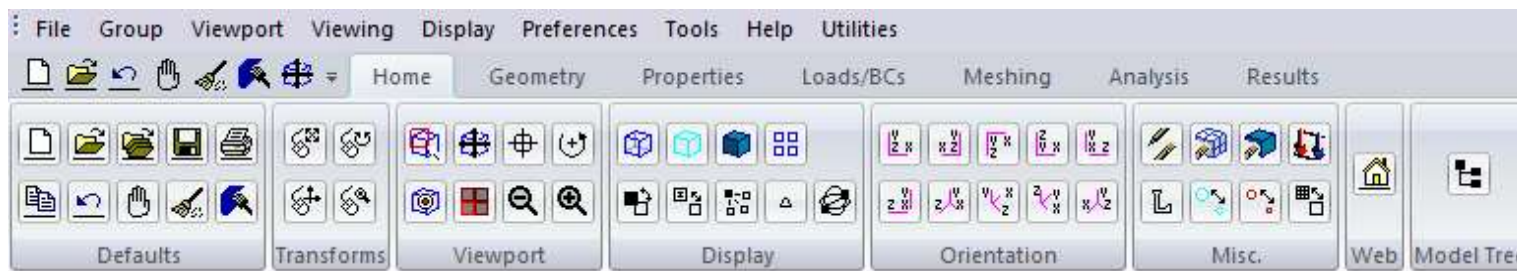


- Create a new database:
- a. **File / New...**
  - b. Enter **ex\_1.db** as the File name
  - c. Click **OK**
  - d. Select **Default**
  - e. Click **OK**



- Create geometry points:
- Click on the **Right side view** icon
  - Click on the **Point size** icon
  - Click on the **Cycle Background** icon
  - Click on the **Geometry** icon
  - Geometry: **Create/Point/XYZ**
  - Uncheck **Auto Execute**
  - Enter **[0 225 0]** as the Point Coordinates List
  - Click **Apply**
  - 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.



Point 2

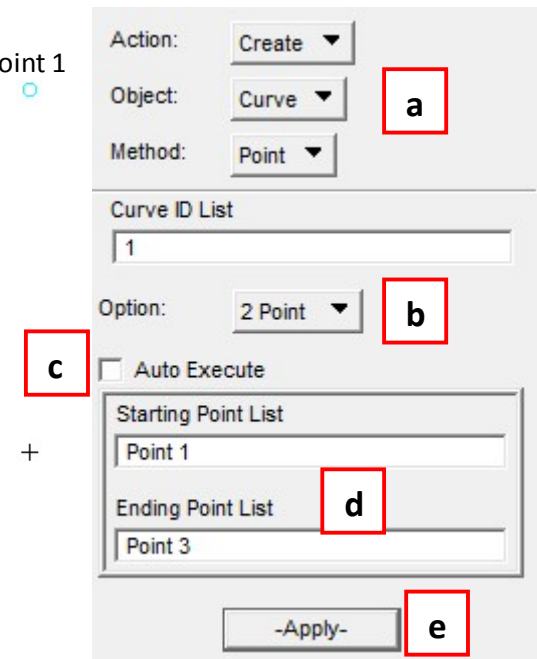
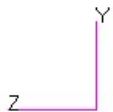
Point 3

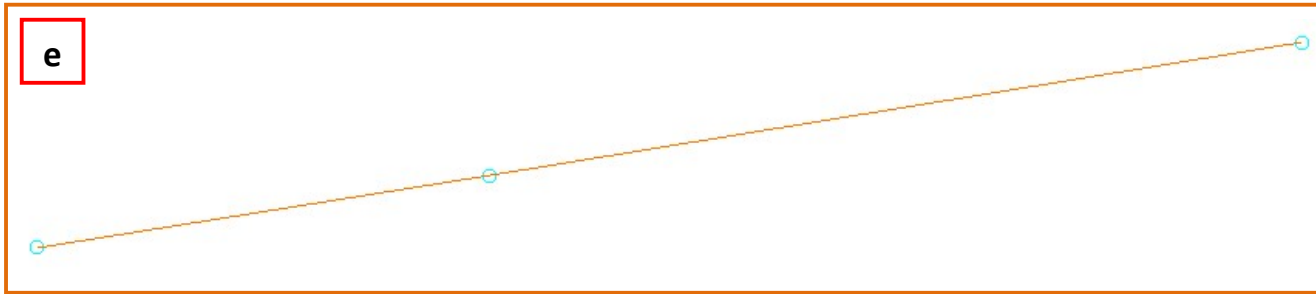
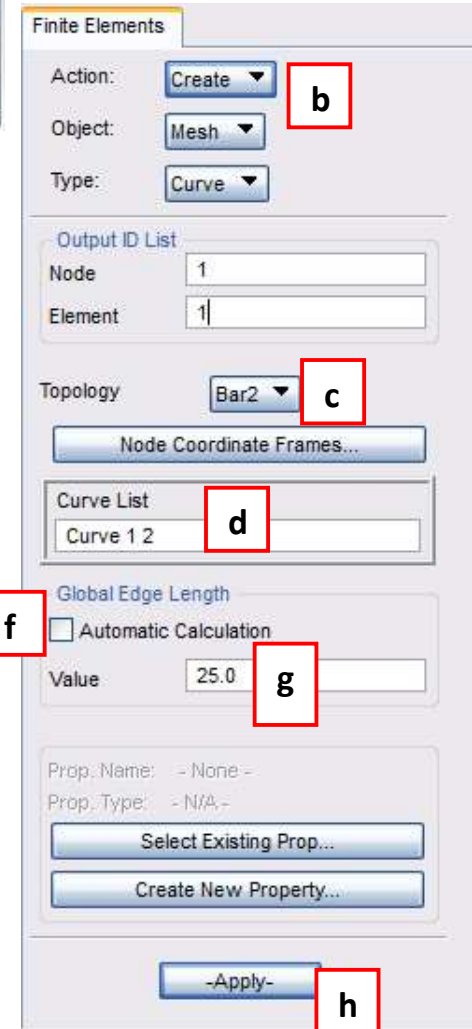
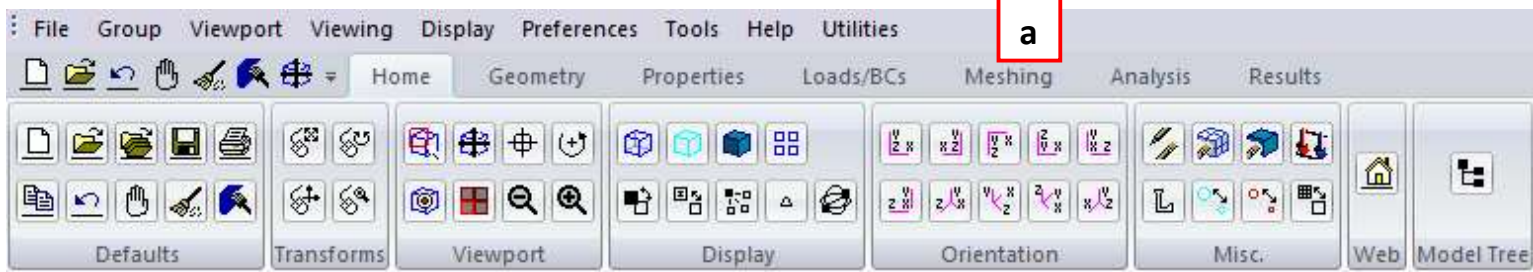
Point 1

Create curves:

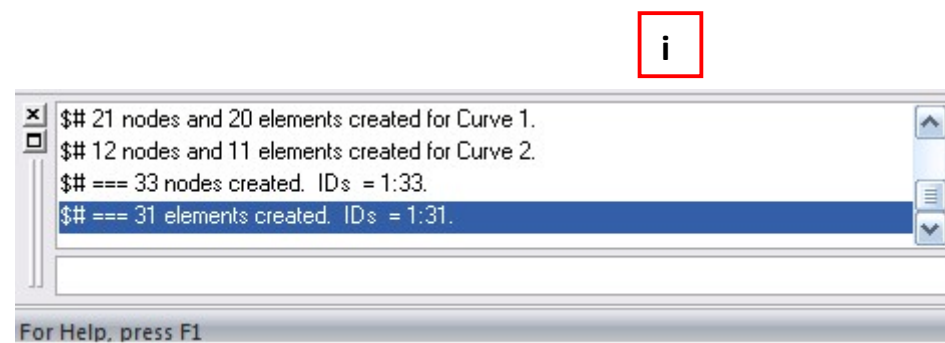
- a. Geometry: **Create/Curve/Point**
- b. Option: **2 Point**
- c. Uncheck **Auto Execute**
- d. Select **Point 1** as the starting point and **Point 3** as the ending point
- e. Click **Apply**
- f. Create one more curve using **Point 3** and **Point 2**

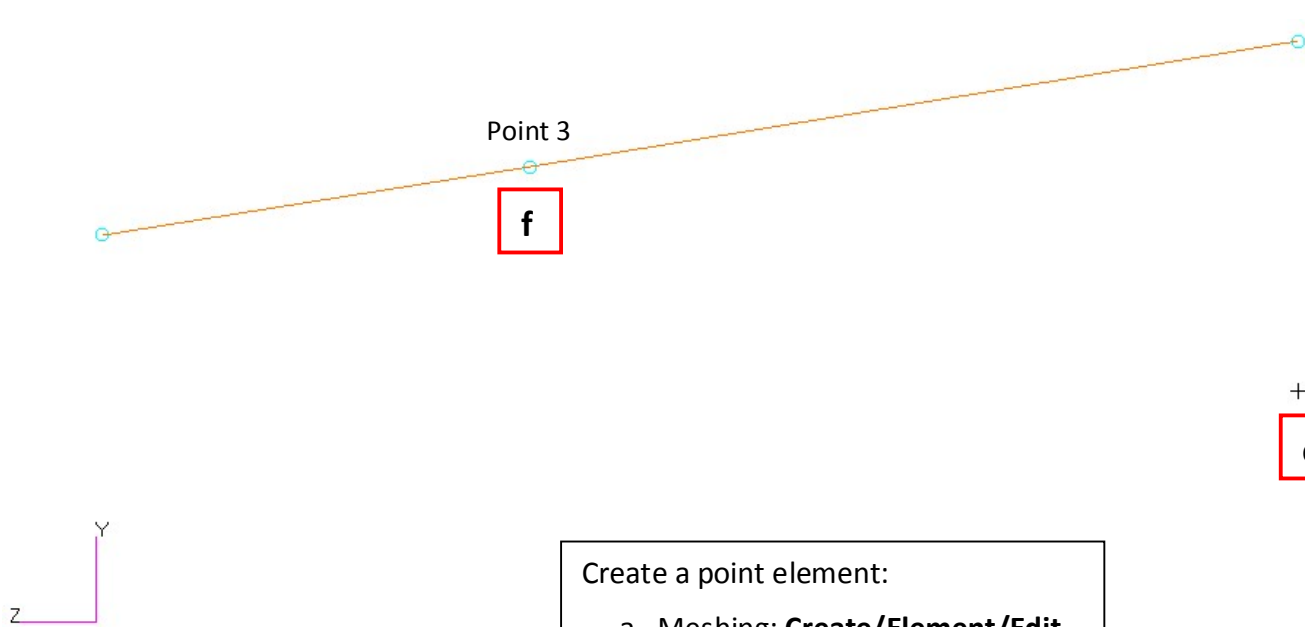
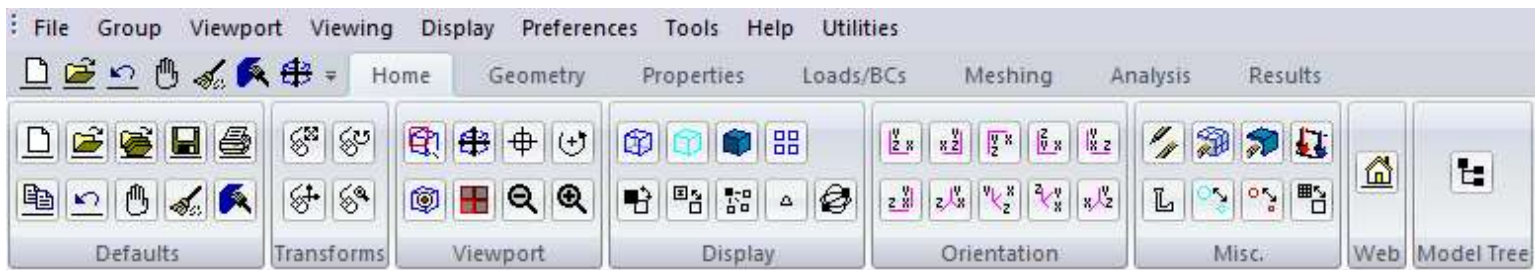
Remark: Points numbers may be different.





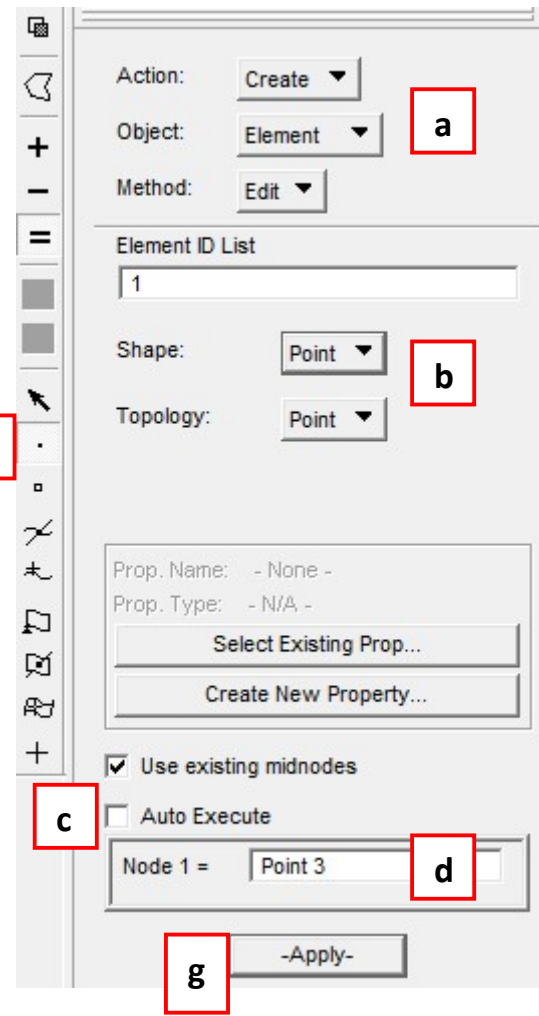
- Mesh the curves:
- Click on the **Meshing** tab
  - Meshing: **Create/Mesh/Curve**
  - Topology: **Bar2**
  - Click on the **Curve List** panel
  - Select all curves by clicking and dragging the mouse
  - Uncheck **Automatic Calculation**
  - Enter **25** as the Value of the Global Edge Length
  - Click **Apply**
  - Check message window whether elements were created

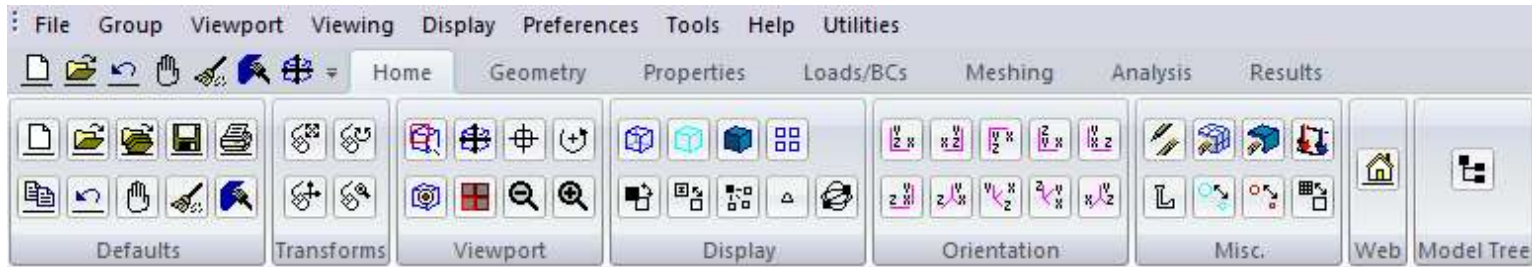




Create a point element:

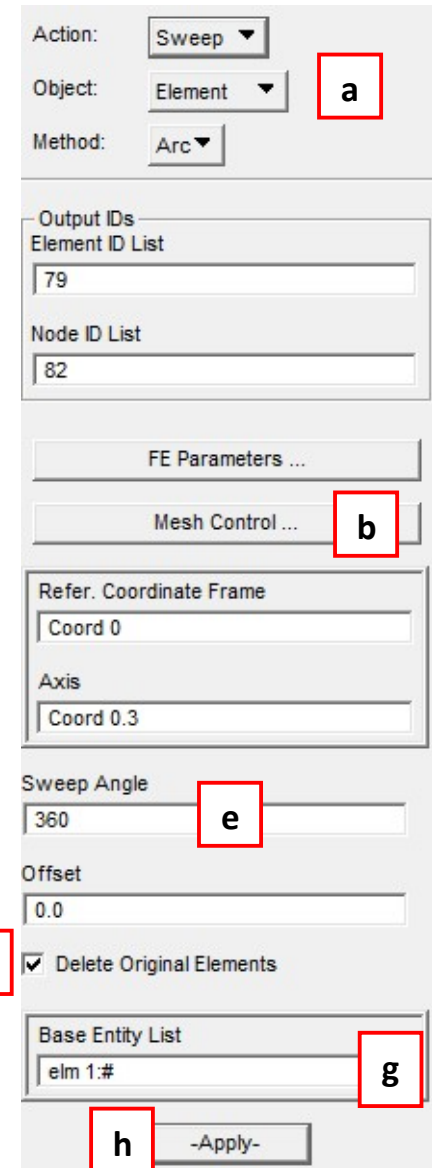
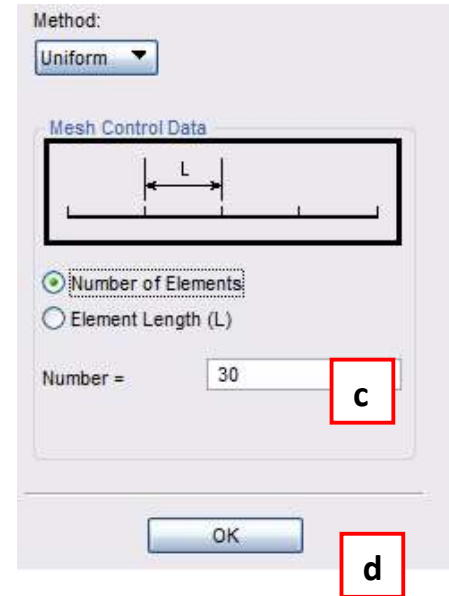
- a. Meshing: **Create/Element/Edit**
- b. Shape and Topology: **Point**
- c. Uncheck **Auto Execute**
- d. Click on the **Node 1** panel
- e. Select **Point** icon
- f. Select **Point 3**
- g. Click **Apply**



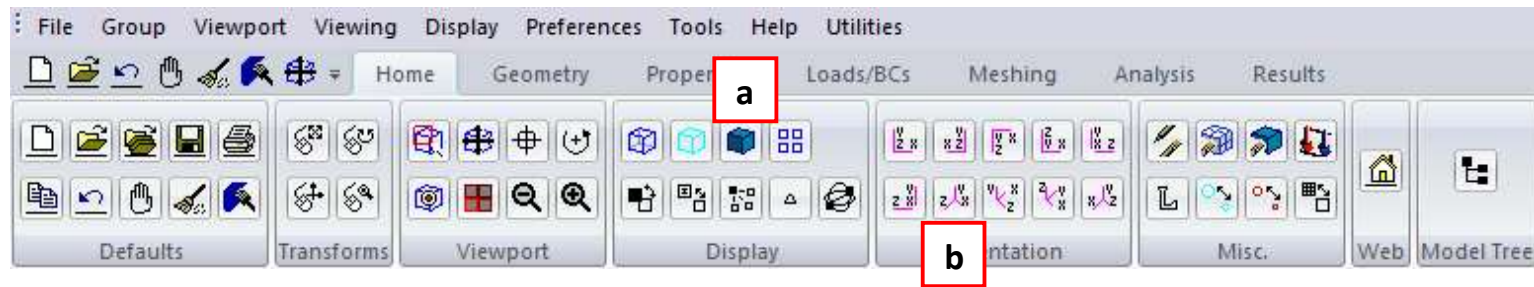


Sweep the elements to create the final mesh:

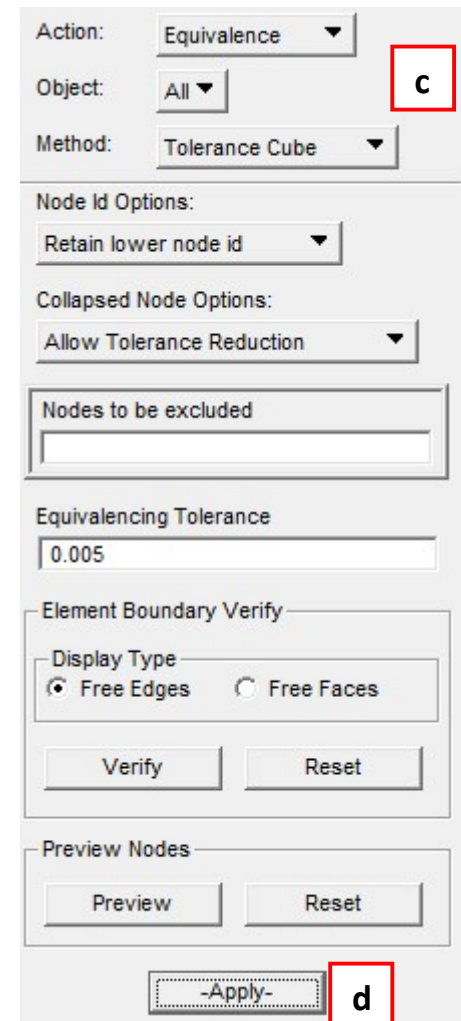
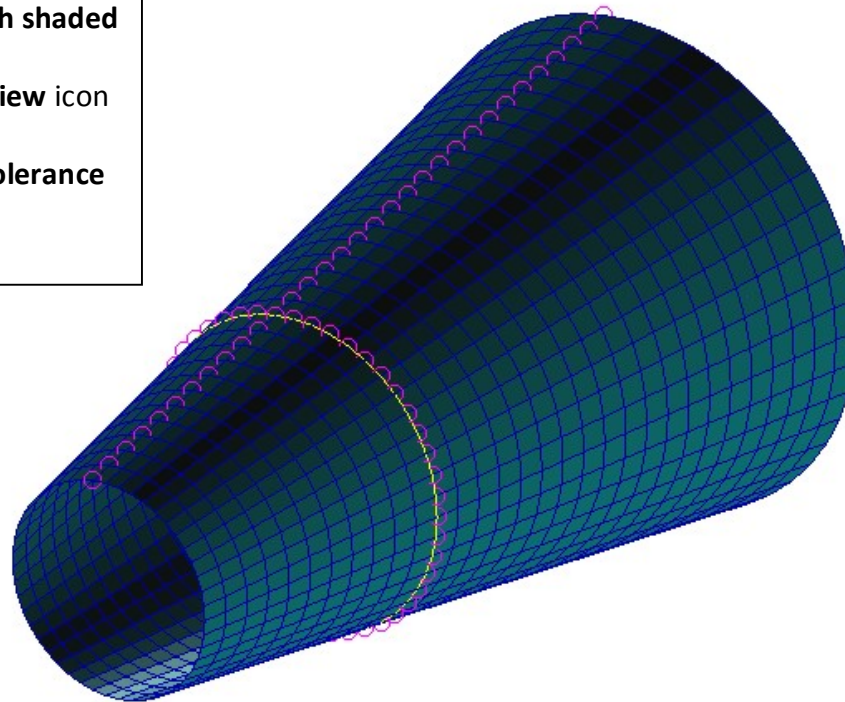
- a. Meshing: **Sweep/Element/Arc**
- b. Click **Mesh Control...**
- c. Enter **30** as the Number of Elements
- d. Click **OK**
- e. Enter **360** as the Sweep Angle
- f. Check **Delete Original Elements**
- g. In the **Base Entity List** enter **elm 1:#** to select all previously created elements
-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

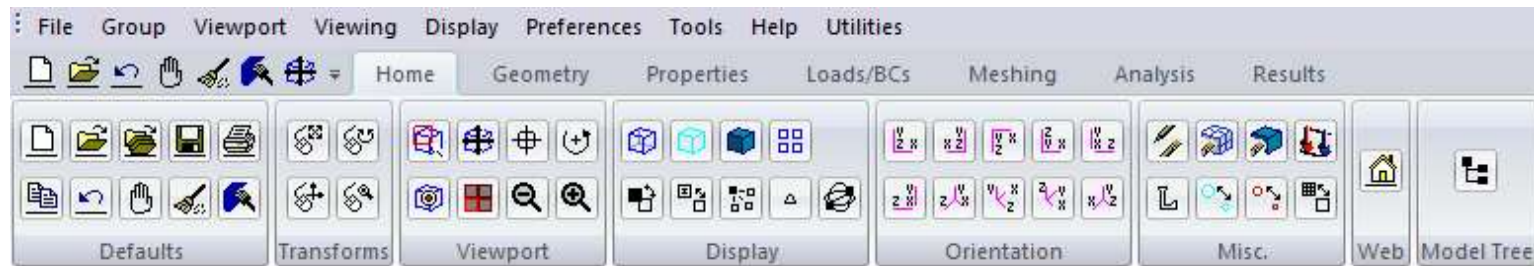






- Delete the duplicate nodes:
- Click on the **Smooth shaded** icon
  - Click on the **Iso 1 View** icon
  - Elements:  
**Equivalence/All/Tolerance Cube**
  - Click **Apply**

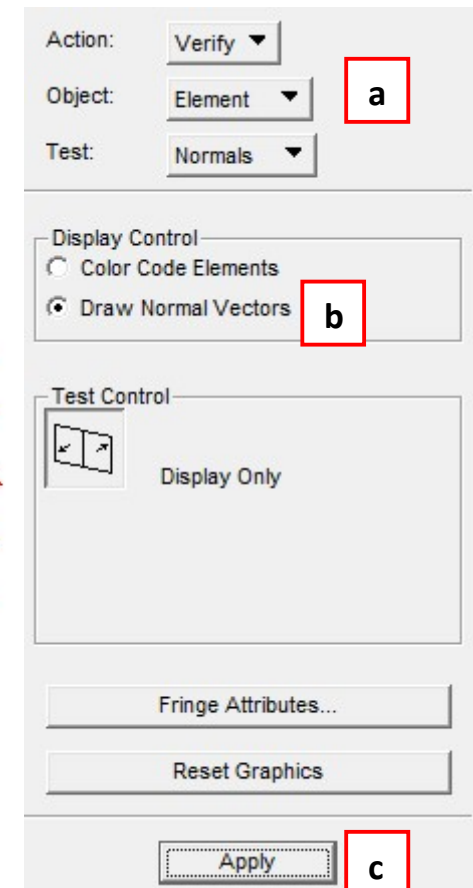
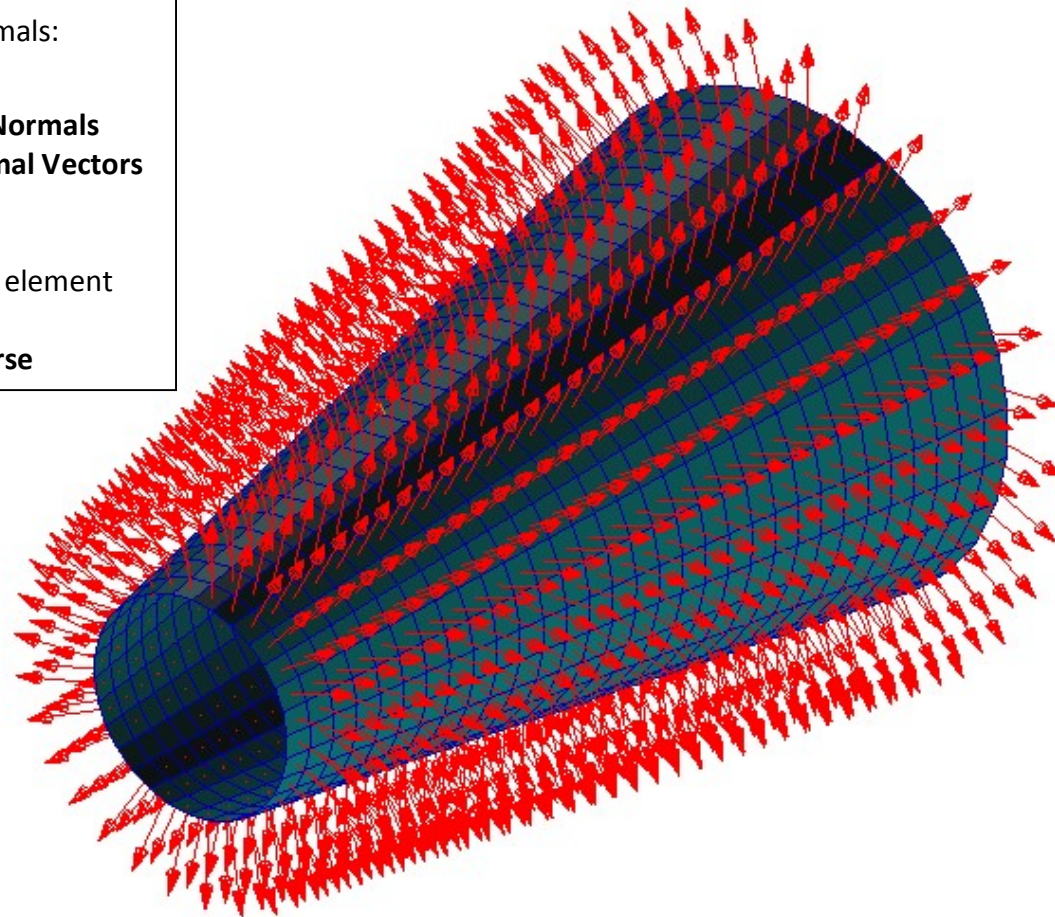




Verify the element normals:

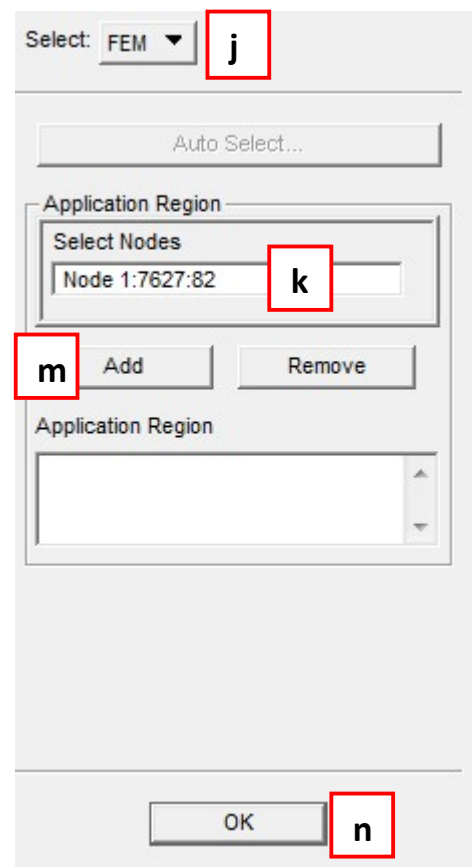
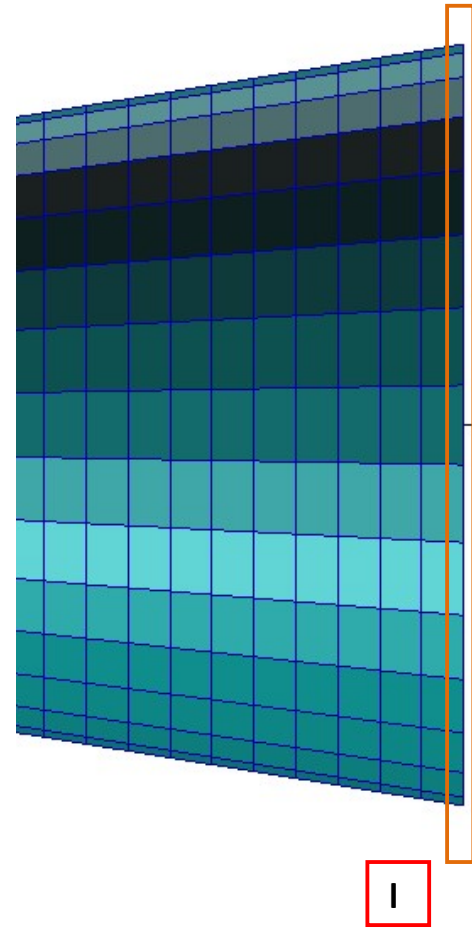
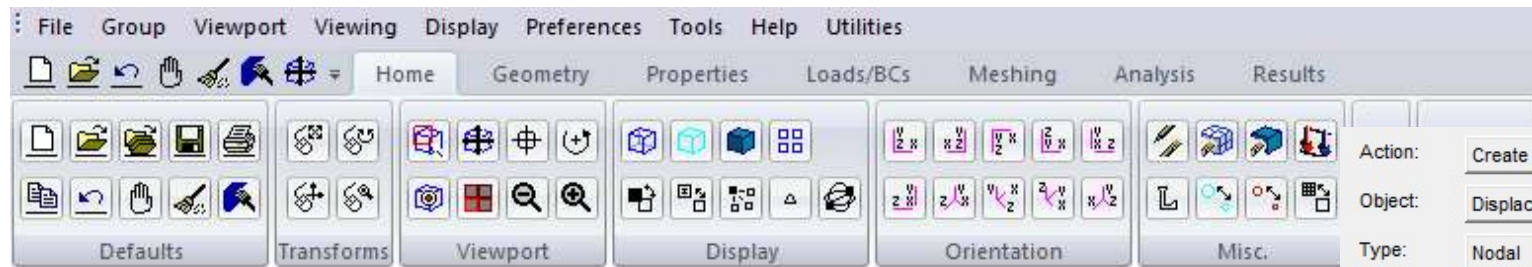
- a. Meshing:  
**Verify/Element/Normals**
- b. Select **Draw Normal Vectors**
- c. Click **Apply**

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

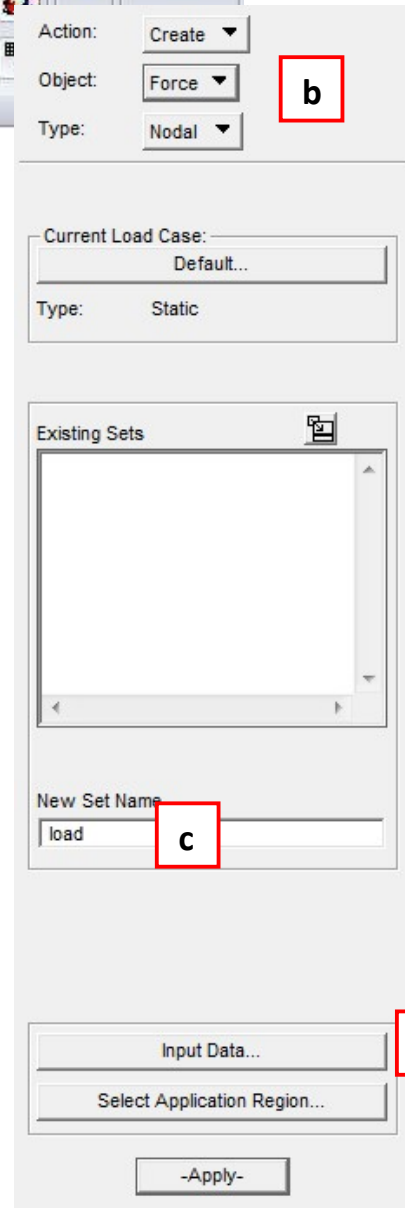
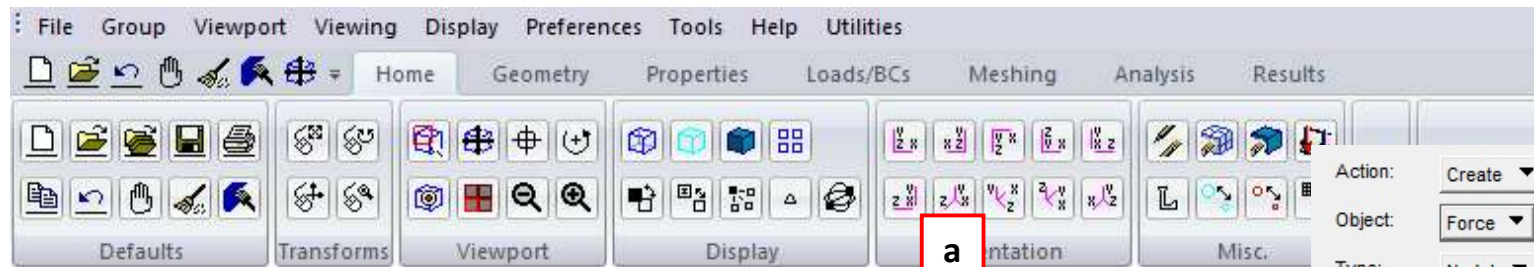


Apply the boundary conditions:

- Click on the **Right side view** icon
- Click on the **Fit view** icon
- Click on the **Loads/BCs** icon
- Loads/BCs:  
**Create/Displacement/Nodal**
- Enter **fix** as the New Set Name
- Click **Input Data...**
- Enter **<0,0,0>** for the Translations and for the Rotations
- Click **OK**

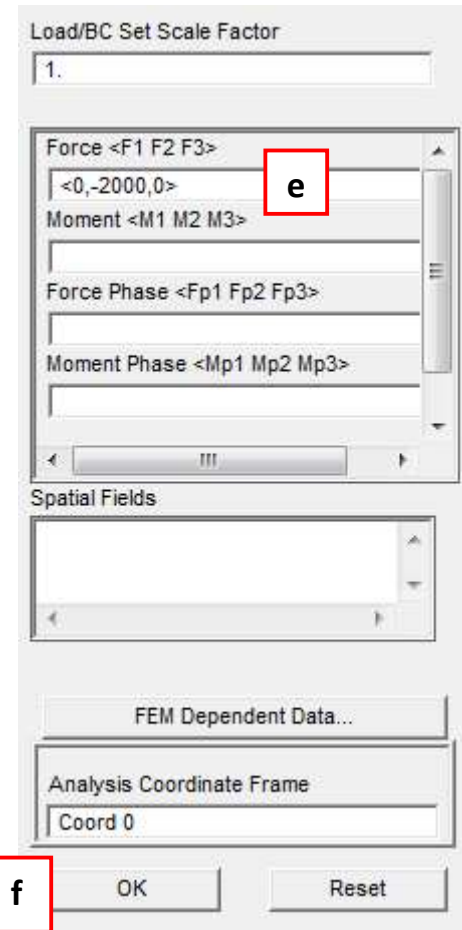


- i. Click **Select Application Region...**
- j. Select **FEM**
- k. Click on the **Select Nodes** panel
- l. Select nodes by clicking and dragging the mouse
- m. Click **Add**
- n. Click **OK**
- o. Click **Apply**



Apply the force:

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



g. Click **Select Application Region...**

h. Select **Geometry**

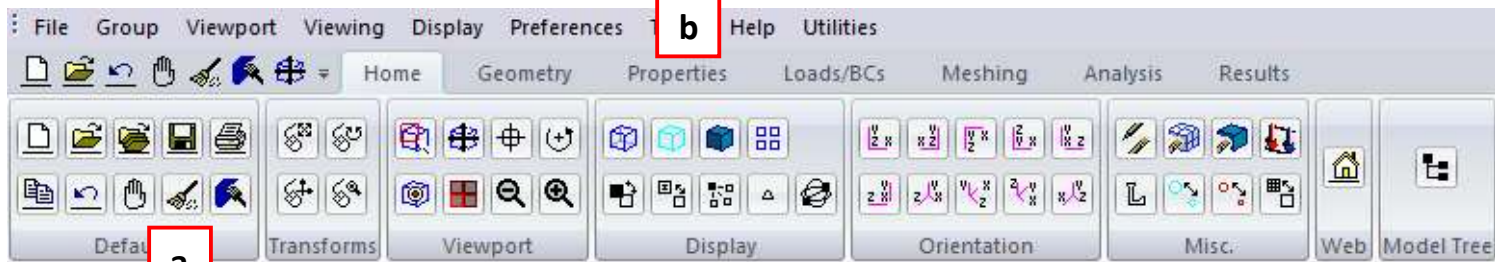
i. Click on the **Select Geometry Entities** panel

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:
- Click on the **Properties** tab
  - Click on **Isotropic** icon, then:  
**Create/Isotropic/Manual Input**
  - Enter **aluminum** as the Material Name
  - Click **Input Properties...**
  - Enter **73000** as Elastic Modulus and **0.33** as Poisson Ratio
  - Click **OK**
  - Click **Apply**
  - Repeat steps **f-k** to define the second, relatively stiff material e.g.: **ring\_mat**,  $E = 2e7$ ,  $\nu = 0.29$

Constitutive Model: Linear Elastic

Property Name	Value
Elastic Modulus =	73000 (labeled 'f')
Poisson Ratio =	.33
Shear Modulus =	
Density =	
Thermal Expan. Coeff =	
Structural Damping Coeff =	
Reference Temperature =	

Temperature Dep/Model Variable Fields:

Current Constitutive Models:

OK (labeled 'g')    Clear    Cancel

Action: Create (labeled 'c')

Object: Isotropic

Method: Manual Input

Existing Materials

Filter: \*

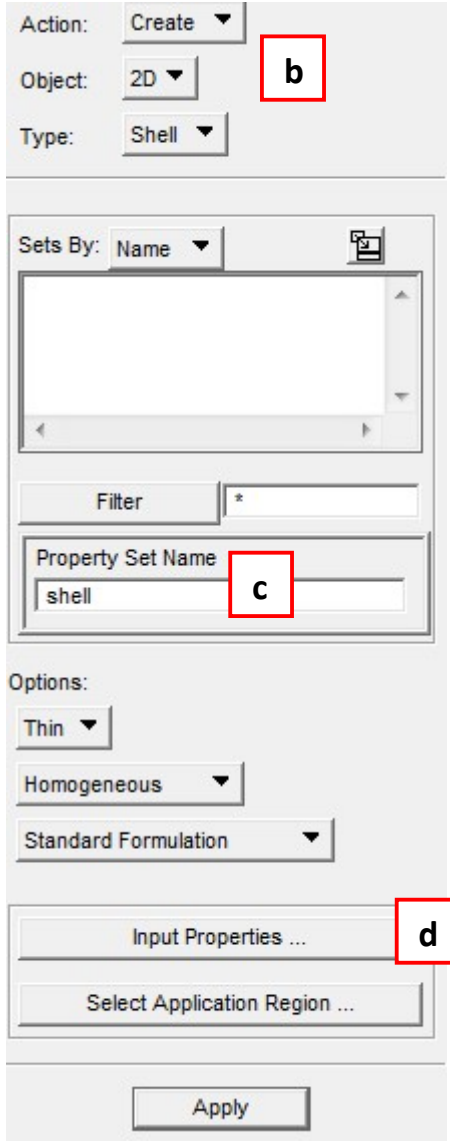
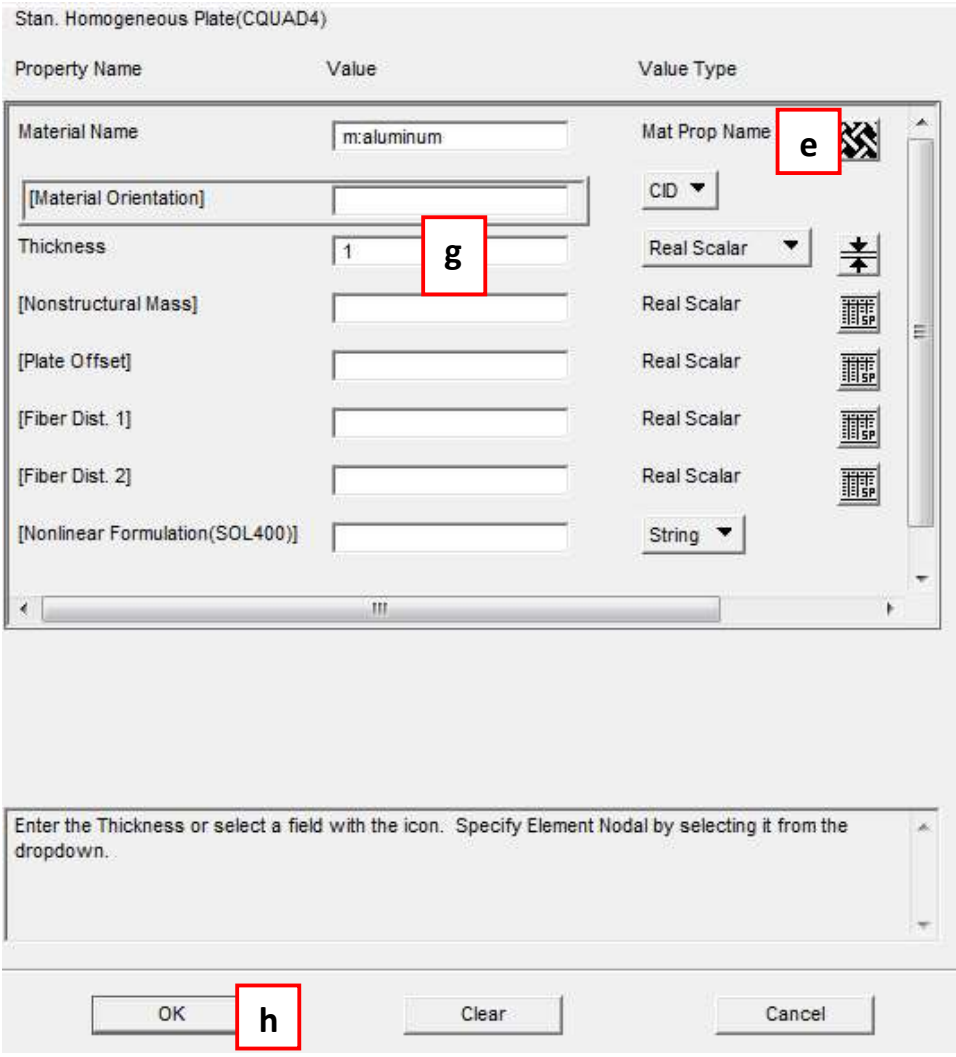
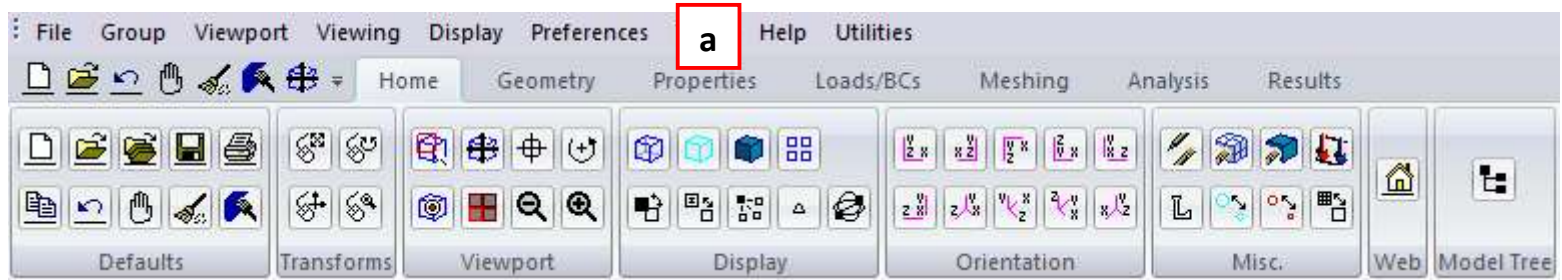
Material Name: aluminum (labeled 'd')

Description:

Input Properties ... (labeled 'e')

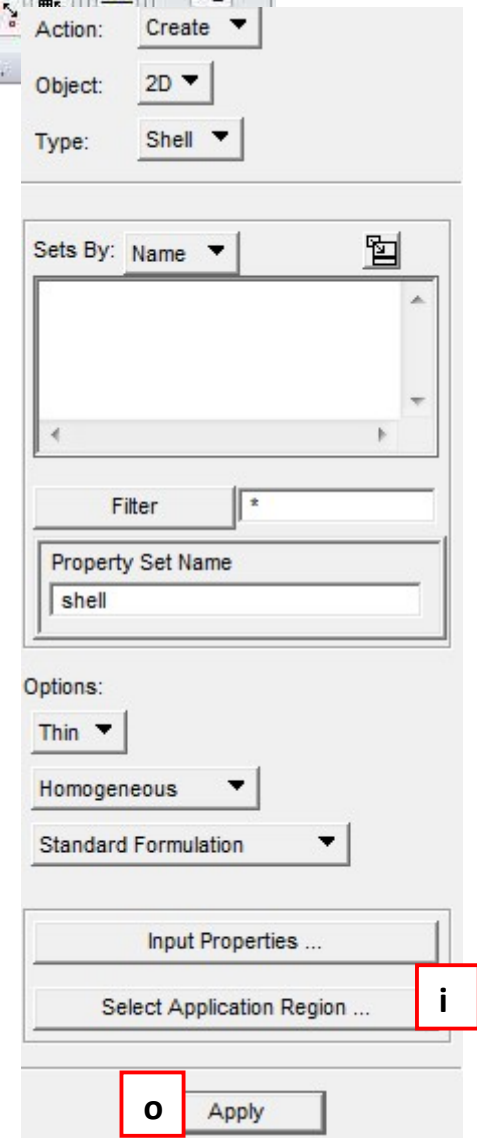
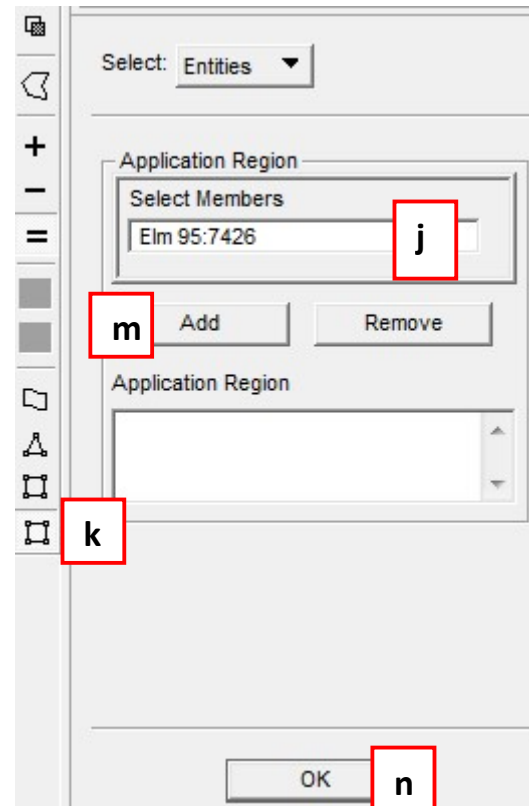
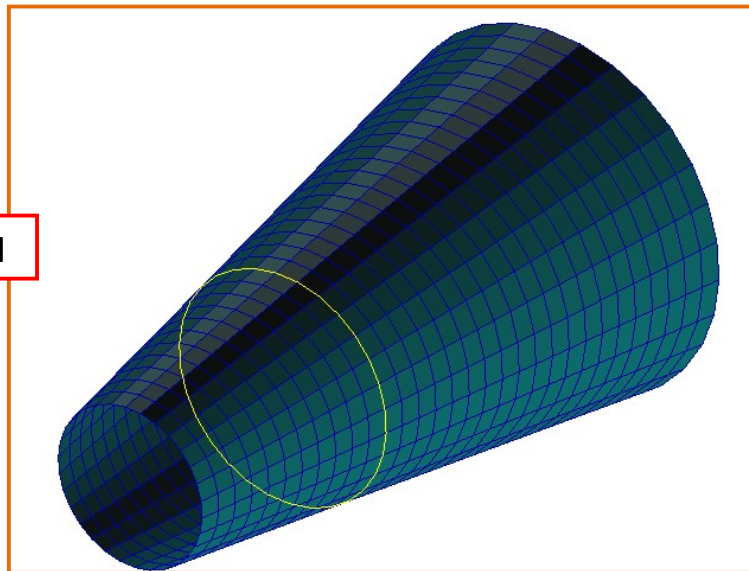
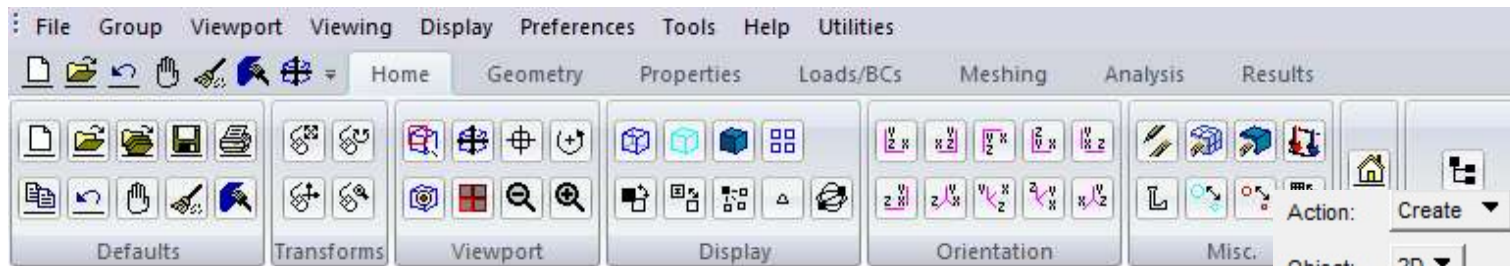
Change Material Status ...

Apply (labeled 'h')

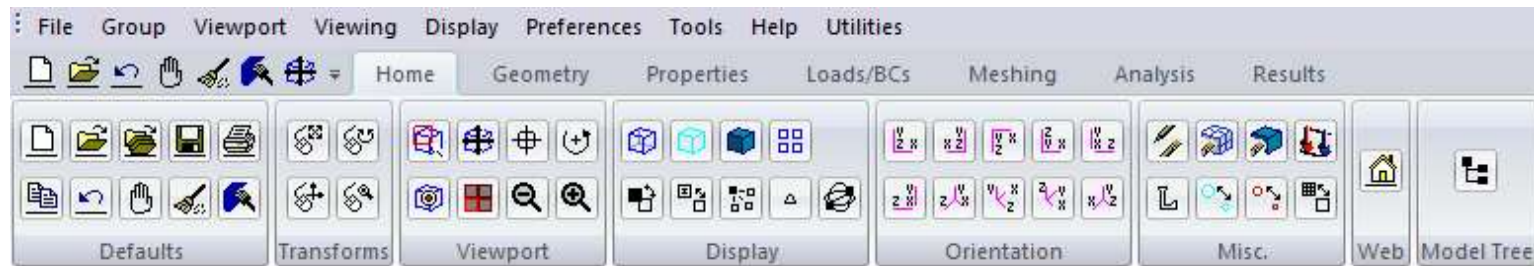


- Assign the properties (shell):
- a. Double-click **Properties** tab
  - b. Properties: **Create/2D/Shell**
  - c. Enter **shell** as the New Set Name
  - d. Click **Input Properties...**
  - e. Click on the **Mat Prop Name** icon
  - f. Select **aluminum**
  - g. Enter **1** as the Thickness
  - h.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



General Section Beam (CBAR)

Property Name	Value	Value Type
[Section Name]	na:	Properties
Material Name		Mat Prop Name
Bar Orientation		Vector
[Offset @ Node 1]		Vector
[Offset @ Node 2]		Vector
[Pinned DOFs @ Node 1]		String
[Pinned DOFs @ Node 2]		String
Area		Real Scalar

Action: Create

Object: 1D

Type: Beam

Sets By: Name

shell

Filter: \*

Property Set Name: ring

Options:

General Section

Standard Formulation

Input Properties ...

Select Application Region ...

Apply

Assign the properties (ring):

- Properties:  
**Create/1D/Beam**
- Enter **ring** as the New Set Name
- Click **Input Properties...**
- Click on the **Create Sections** icon

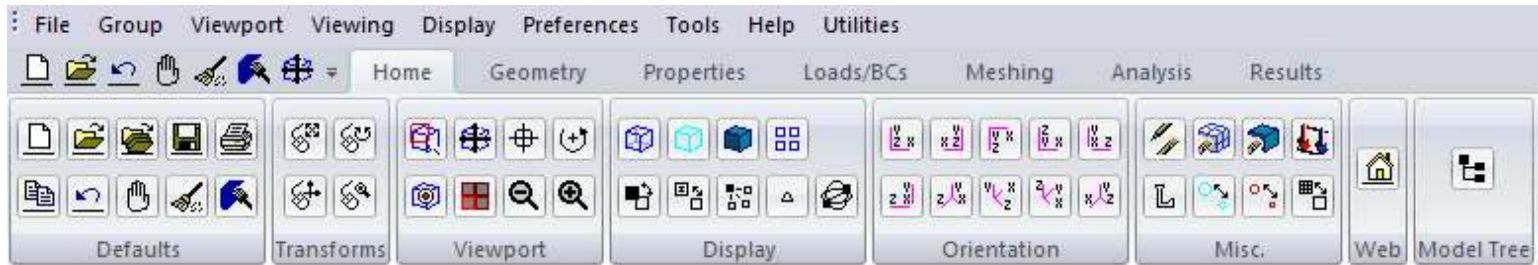




a

b

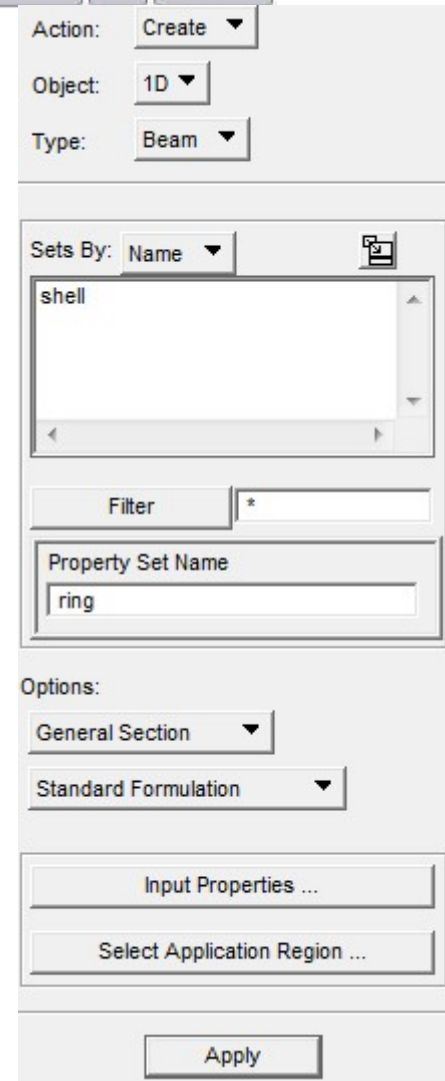
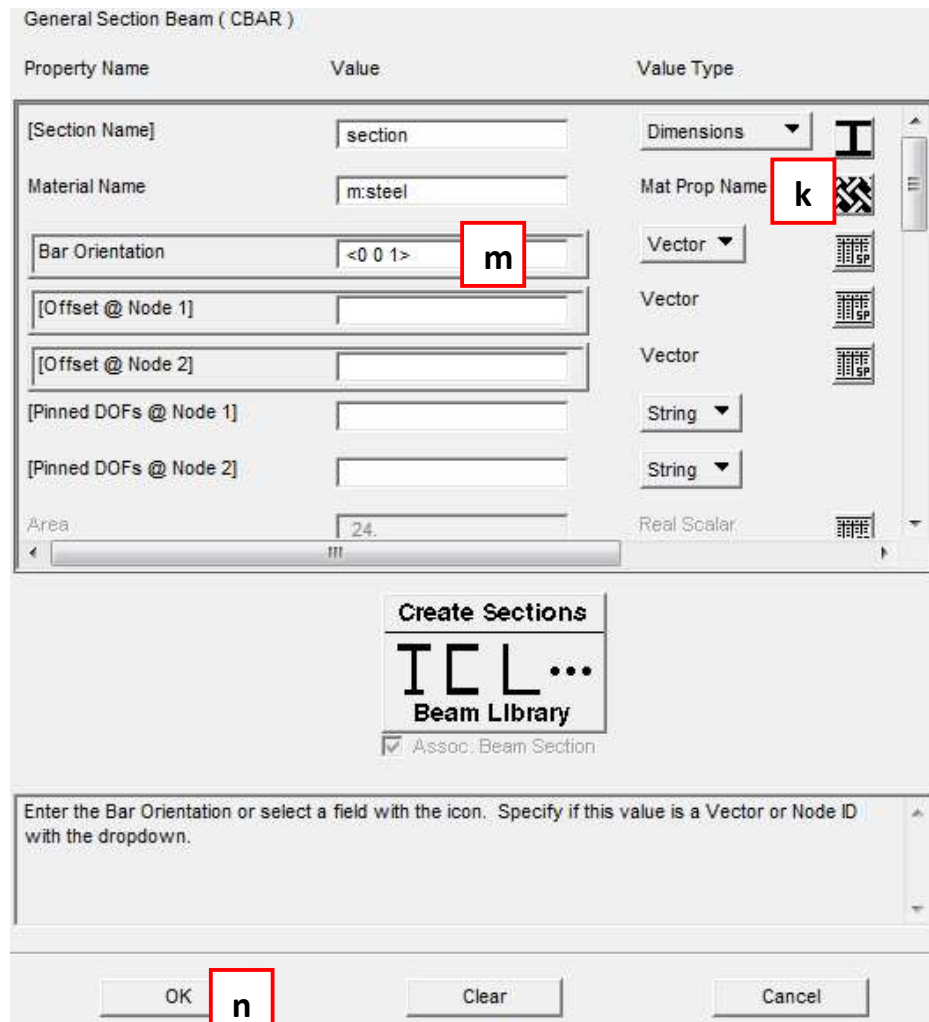
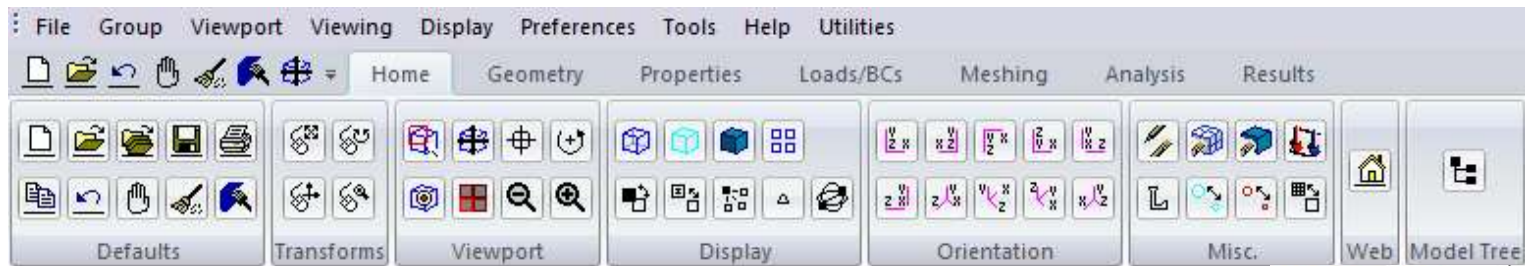
c

d

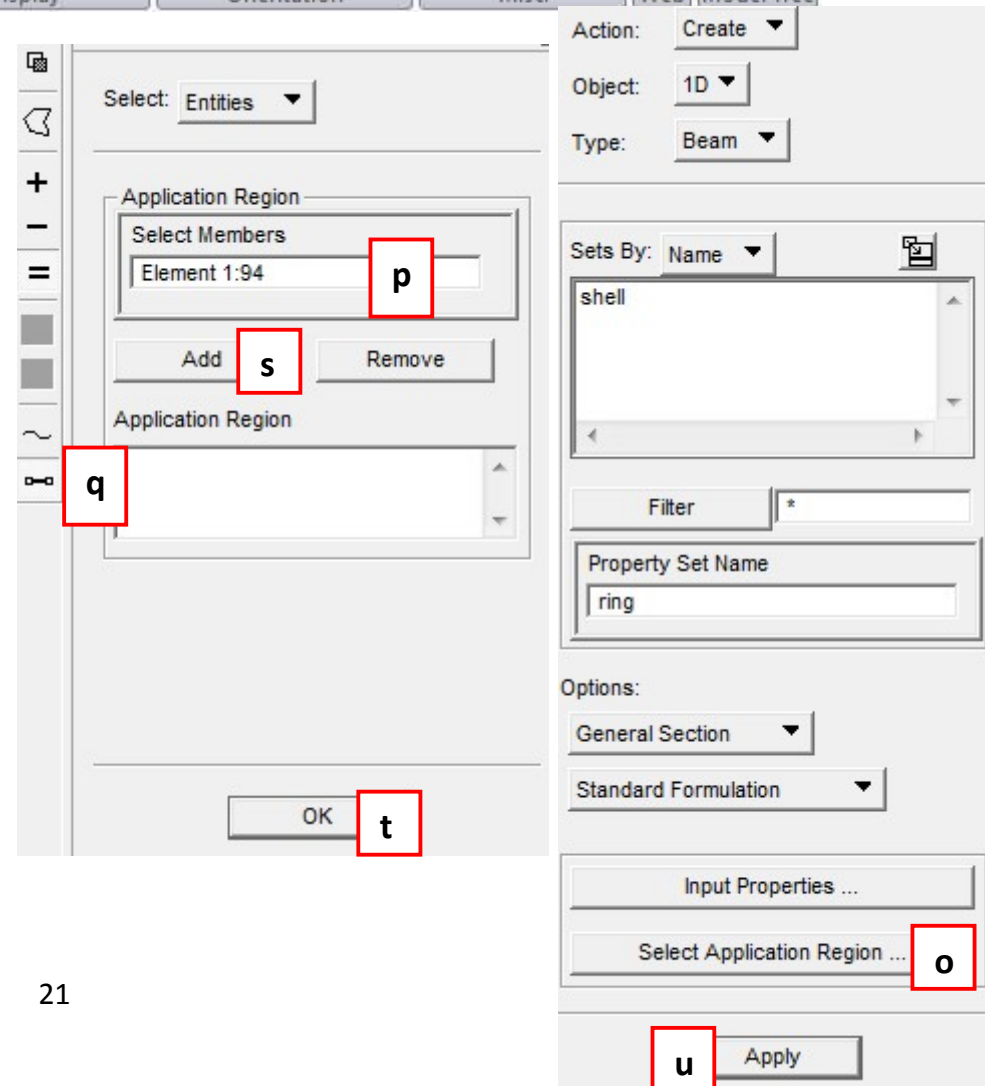
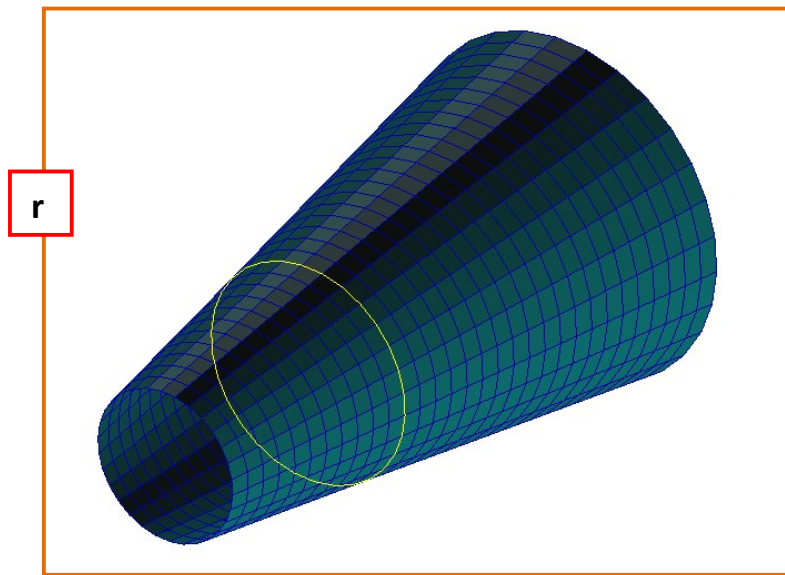
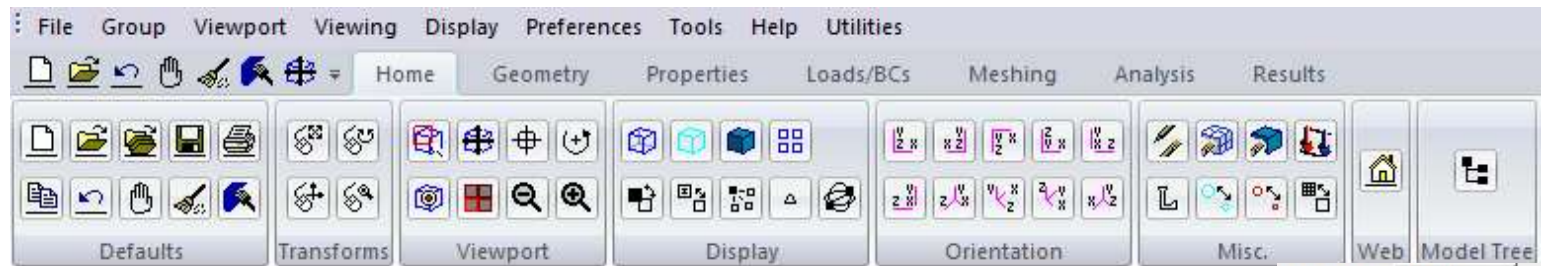


- e. Create Sections: **Create/Standard Shape/NASTRAN Standard**
- f. Enter **section** as the New Section Name
- g. Click 
- h. Click 
- i. Enter **12** as the value of W and **2** as the value of H
- j. Click **OK**

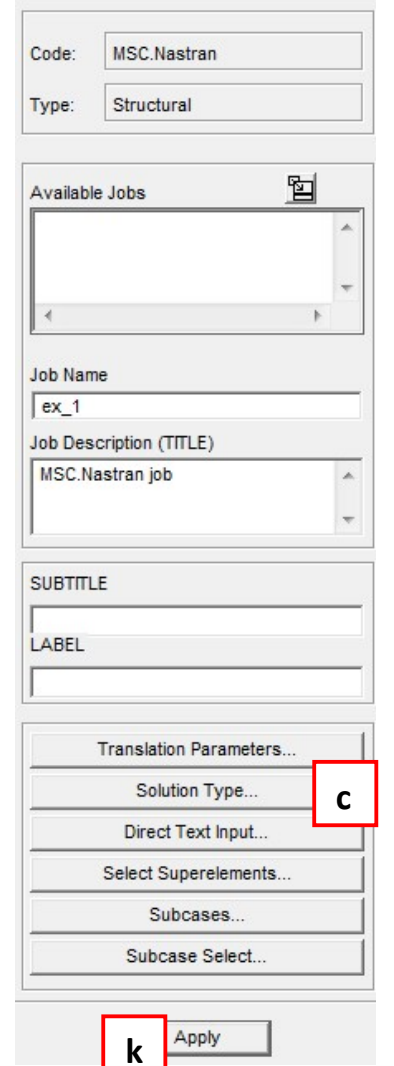
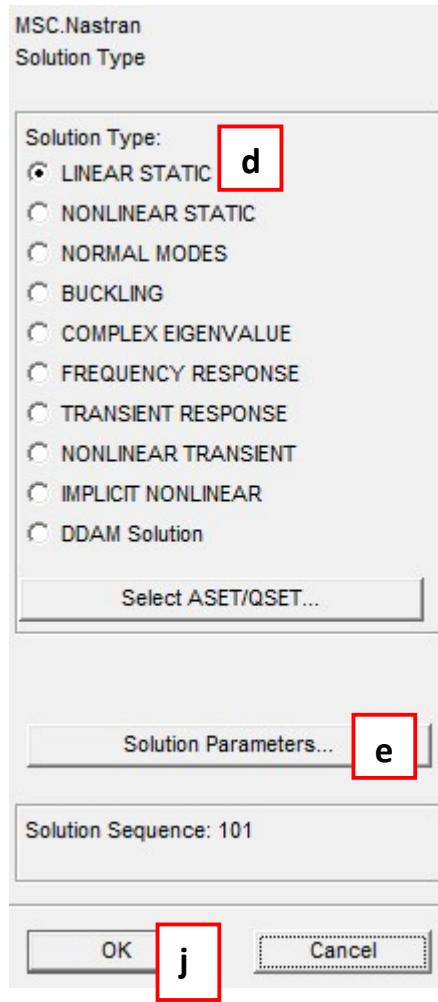
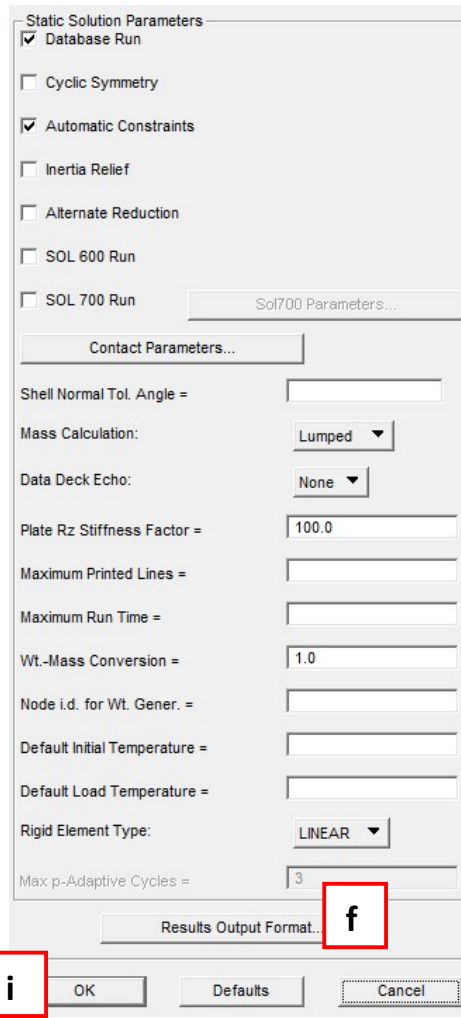
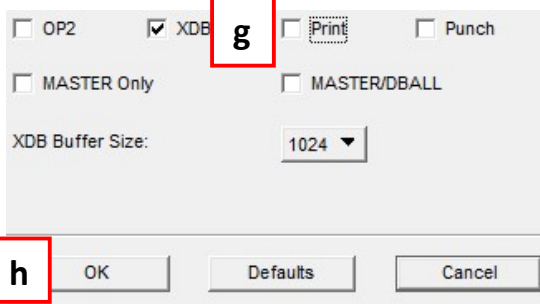
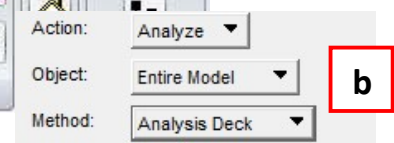
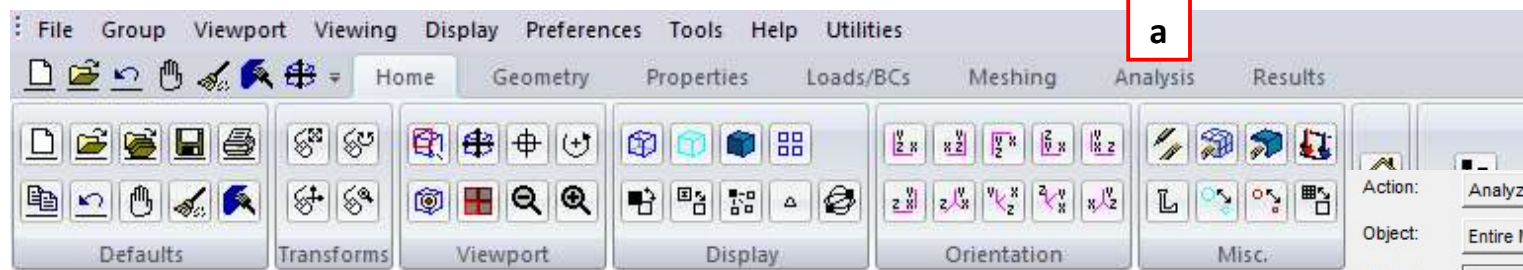
Action: **Create**  
 Object: **Standard Shape**  
 Method: **NASTRAN Standard**  
 Existing Sections: \*  
 New Section Name: **section**  
 W: **12**  
 H: **2**  
 Calculate/Display  
 Write to Report File  
 OK  
 Apply  
 Reset  
 Cancel



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

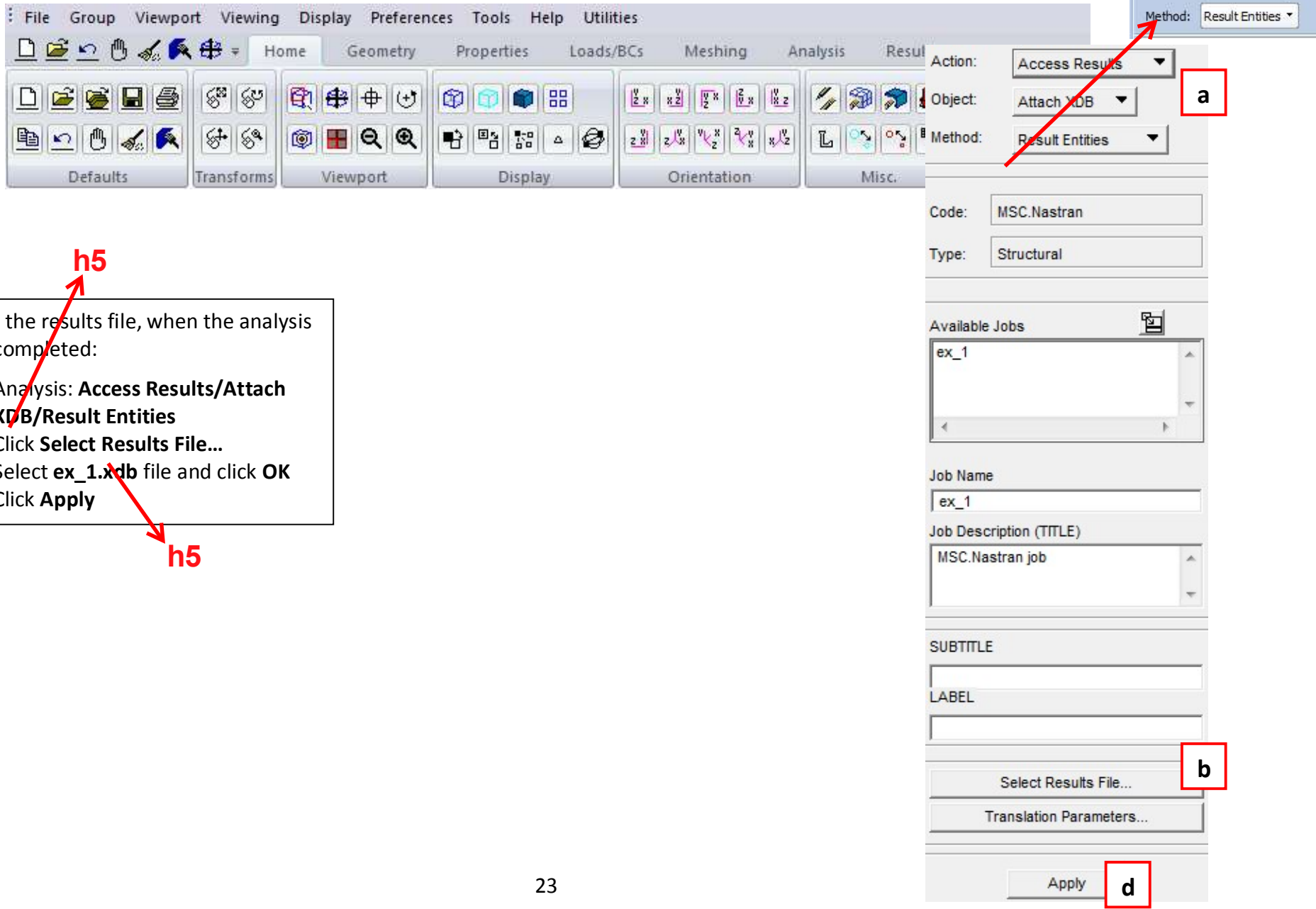


- o. Click **Select Application Region...**
- p. Click on the **Select Members** panel
- q. Select **Beam element** icon
- r. Select all beam elements by clicking and dragging the mouse
- s. Click **Add**
- t. Click **OK**
- u. Click **Apply**



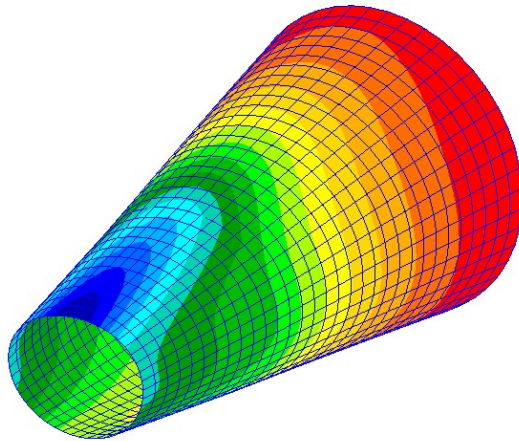
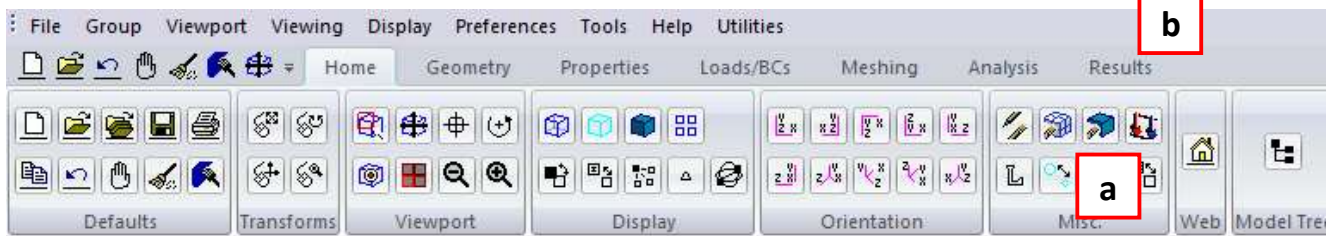
Run a linear analysis:

- 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:

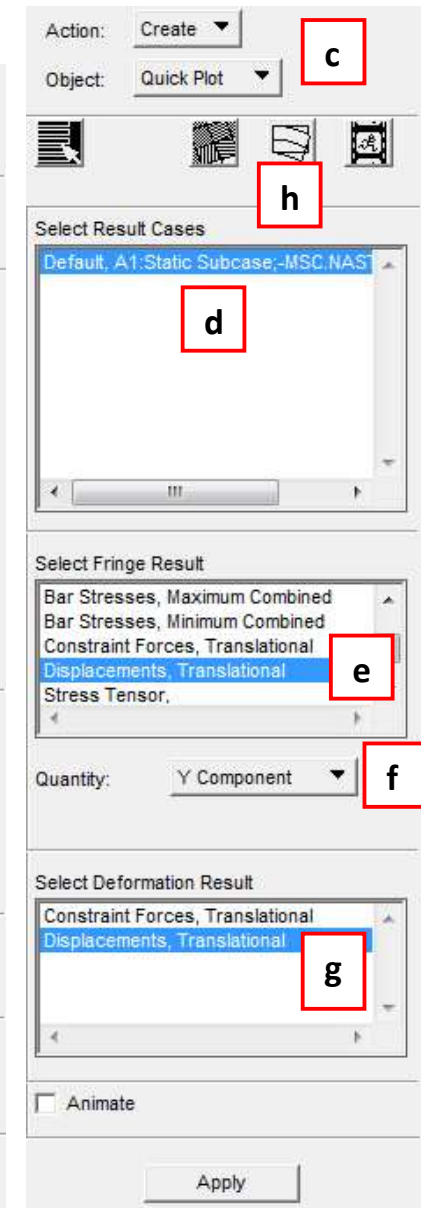
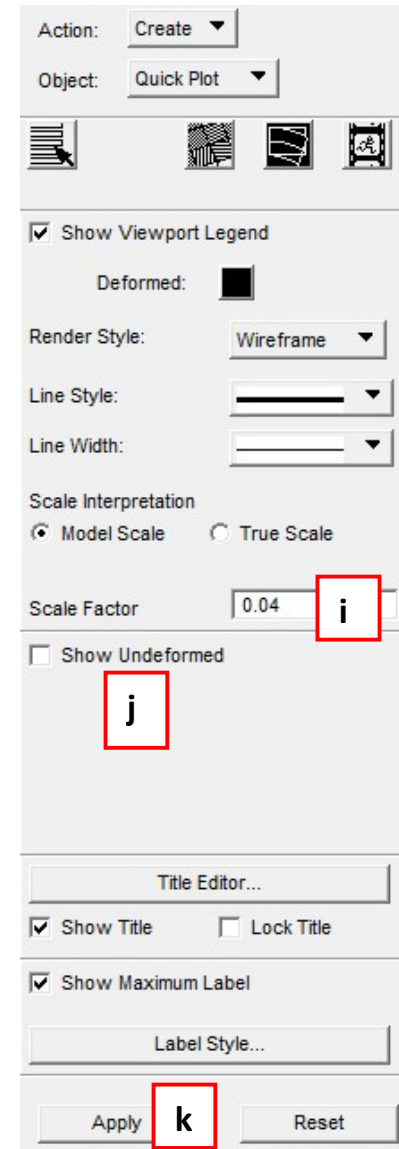
- a. Analysis: **Access Results/Attach XDB/Result Entities**
- b. Click **Select Results File...**
- c. Select **ex\_1.xdb** file and click **OK**
- d. Click **Apply**



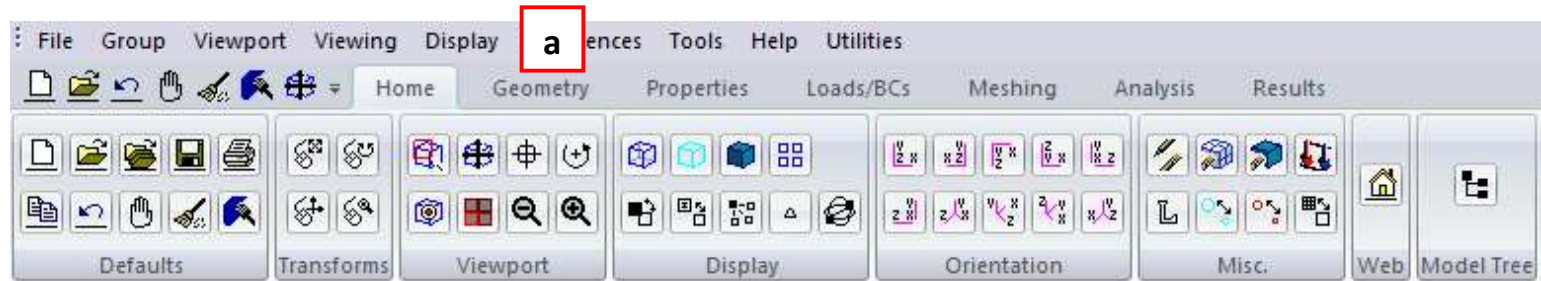
Post-process the results:

- 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...**

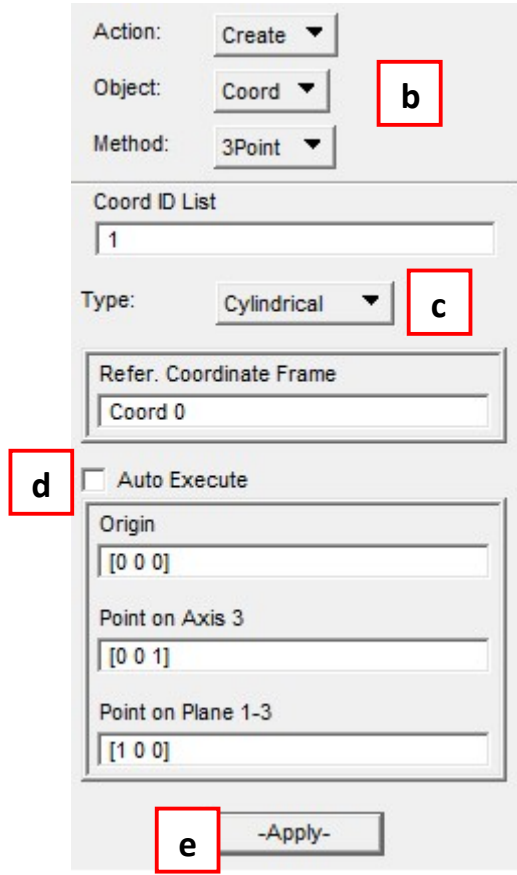
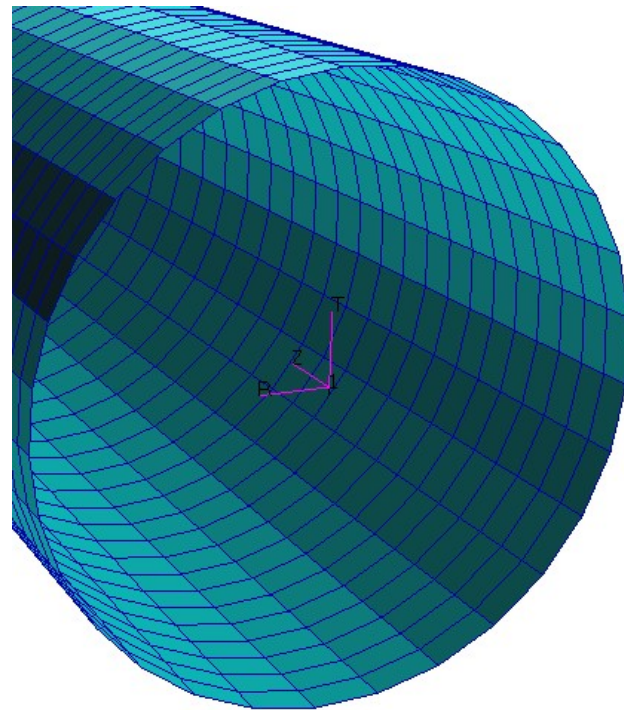


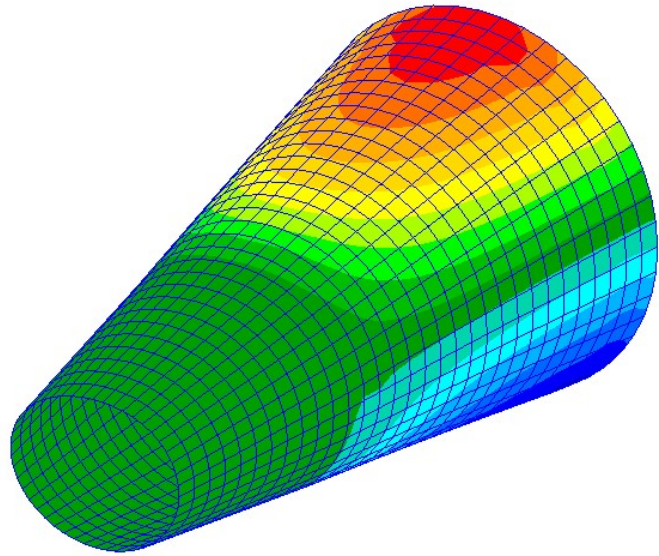
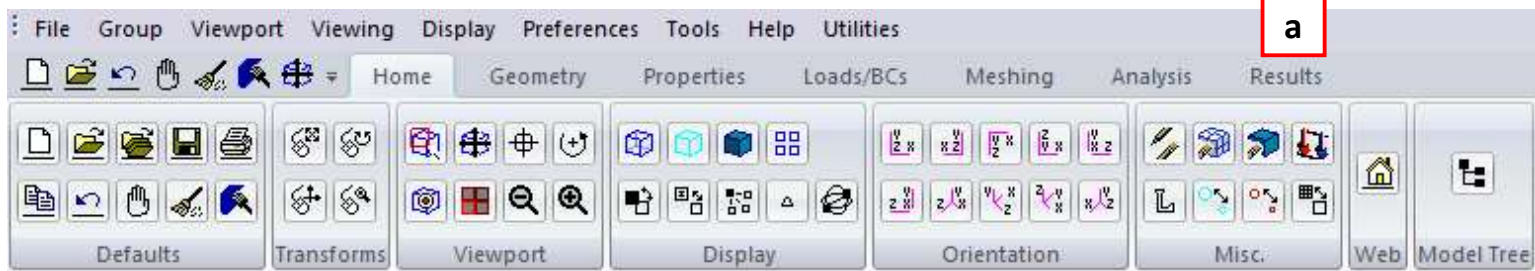




Create a cylindrical coordinate system:

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

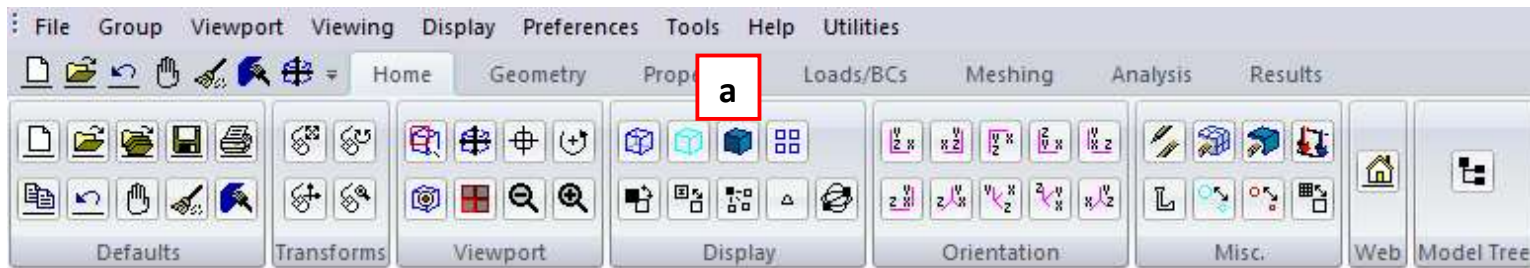




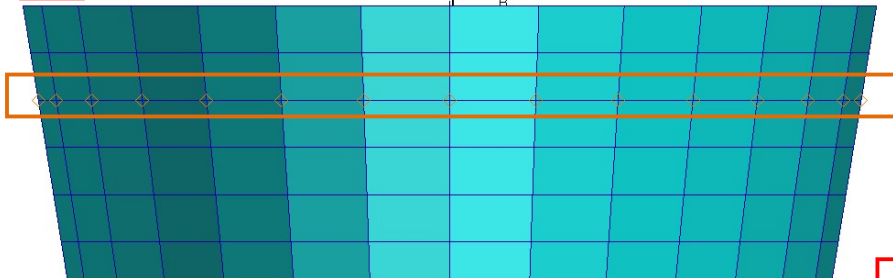
Action: Create **b**  
 Object: Fringe **b**  
 Coordinate Transformation: **g**  
 CID **g**  
 Select Coordinate Frame **h**  
 Coord 1 **h**  
 Scale Factor: 1.0  
 Filter Values: None  
 Averaging Definition:  
 Domain: All Entities  
 Method: Derive/Average  
 Extrapolation: Shape Fn.  
 Use PCL Expression  
 Define PCL Expression...  
 Existing Fringe Plots...  
 Save Fringe Plot As:  
**i** Apply Reset  
 Select Result Cases  
 Default, A1:Static Subcase;-MSC.NAS **c**  
 Select Fringe Result  
 Bar Stresses, Minimum Combined  
 Constraint Forces, Translational  
 Displacements, Translational  
 Stress Tensor, **d**  
 Position...(At Z2)  
 Quantity: Z Component **e**  
 Animate  
 Apply Reset

Post-process the results:

- Click on the **Results** tab
- Results: **Create/Fringe**
- Select Result Cases: **Default, A1:Static Subcase**
- Select Fringe Result: **Stress Tensor**
- Quantity: **Z Component** (axial stress)
- Click on the **Plot Options** icon
- Select **CID**
- Select the cylindrical coordinate system
- Click **Apply**



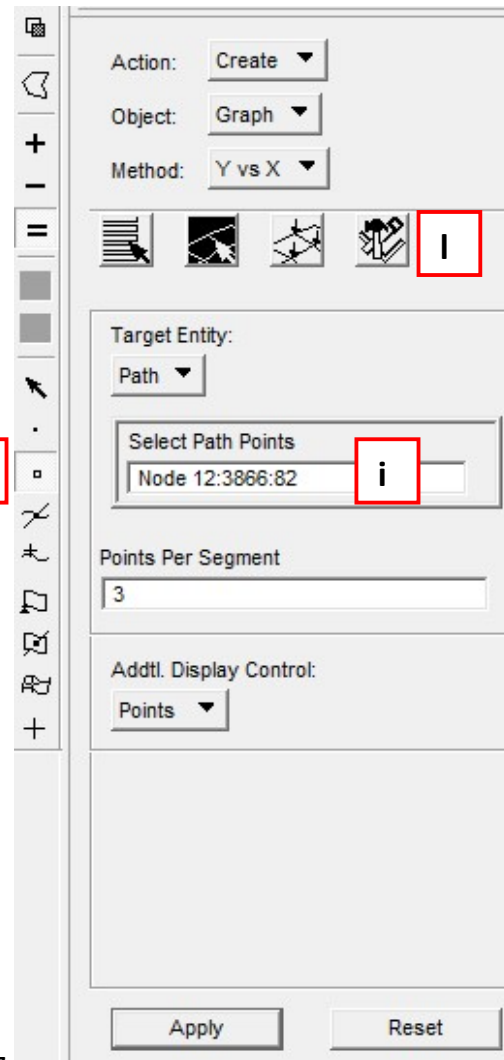
k



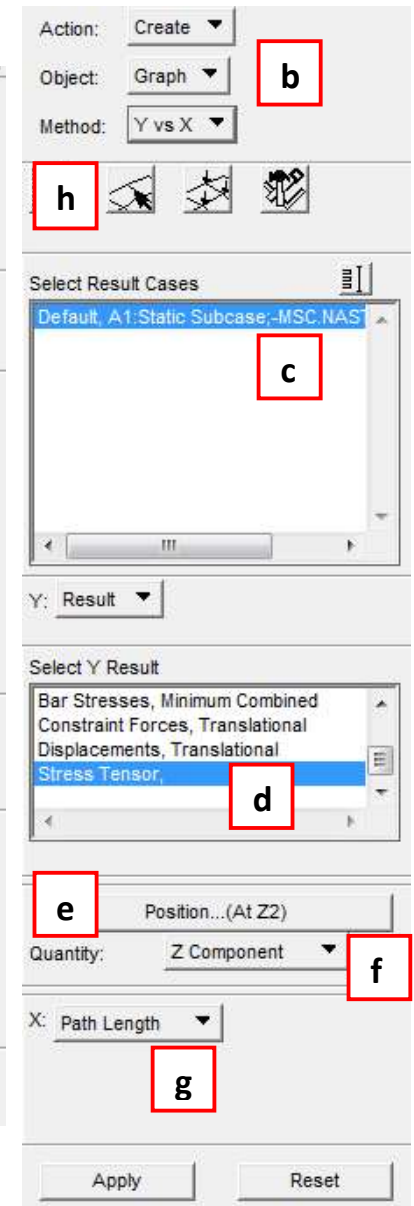
j

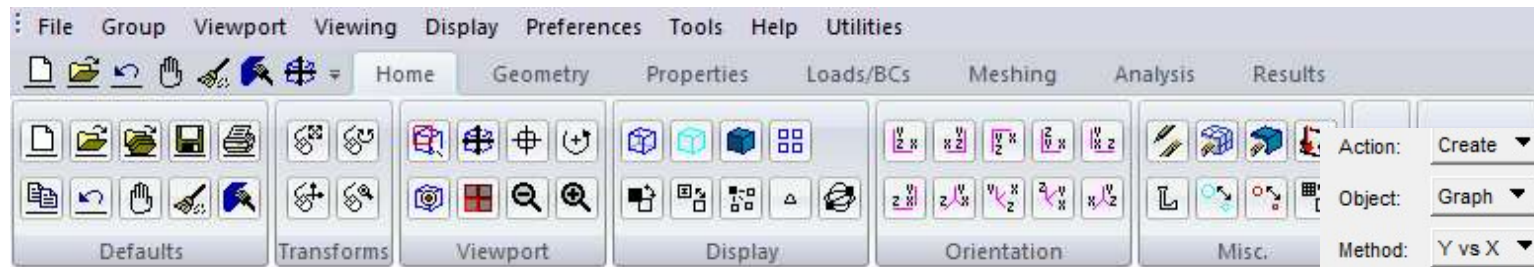
### Create a graph:

- 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



27





Action: Create  
 Object: Graph  
 Method: Y vs X

**p** [Select Results icon]

Coordinate Transformation:  
**m** CID

Select Coordinate Frame  
 Coord 1 **n**

Scale Factor: 1.0

Filter Values: None

Averaging Definition:  
 Domain: All Entities  
 Method: Derive/Average

Extrapolation: Shape Fn.

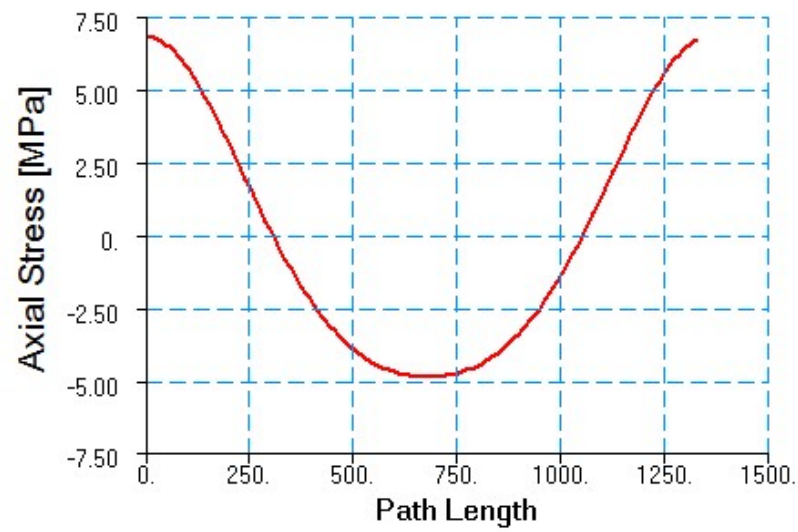
Use PCL Expression  
 Define PCL Expression...

Existing Graph Plots...

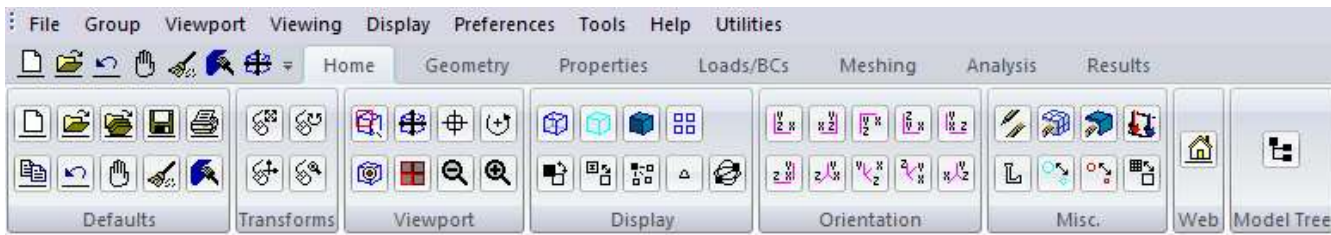
Save Graph Plot As:

**o** Apply    Reset

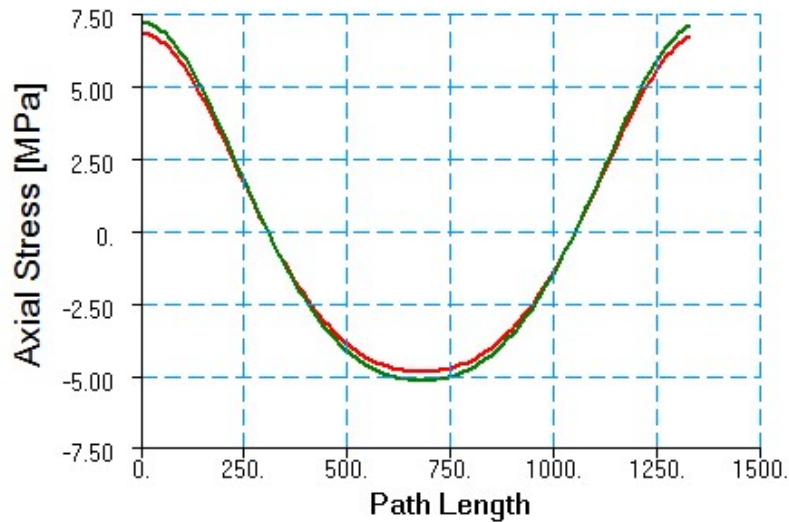
**LEGEND**  
 — At Z2 (top layer)



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



**LEGEND**  
 — At Z2 (top layer)  
 — At Z1 (bottom layer)



- 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

Action: Create  
 Object: Graph  
 Method: Y vs X

Curve Fit: Linear  
 Curve Style: —  
 Show Symbol

Show X Axis Label  
 X Axis Label: Path Length  
 X Axis Scale:  Linear  Log  
 X Axis Format...

Show Y Axis Label  
 Y Axis Label: Axial stress [MPa]  
 Y Axis Scale:  Linear  Log  
 Y Axis Format...

XY Window Name: 3  
 Append Curves in XY Window **s**

**t** Apply Reset

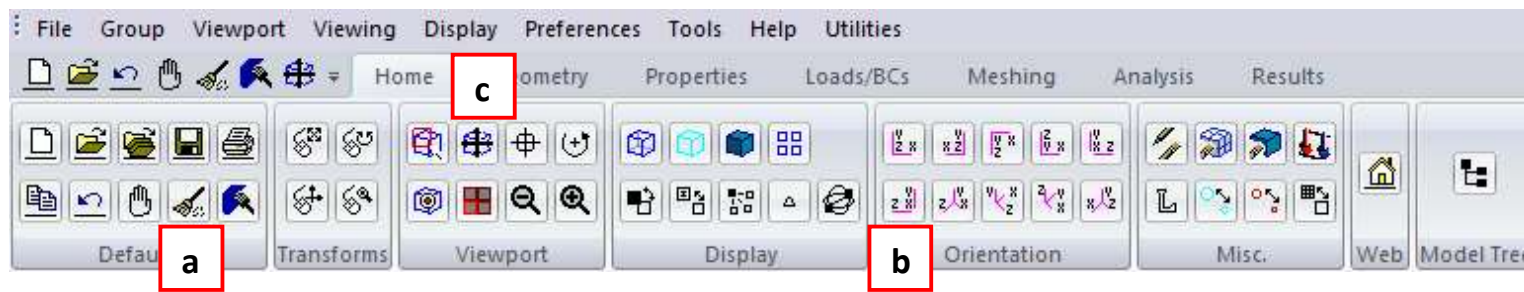
Action: Create  
 Object: Graph  
 Method: Y vs **r**

Select Result Cases  
 Default, A1-Static Subcase -MSC,NAS

Y: Result  
 Select Y Result  
 Bar Stresses, Maximum Combined  
 Bar Stresses, Minimum Combined  
 Constraint Forces, Translational  
 Displacements, Translational  
 Stress Tensor

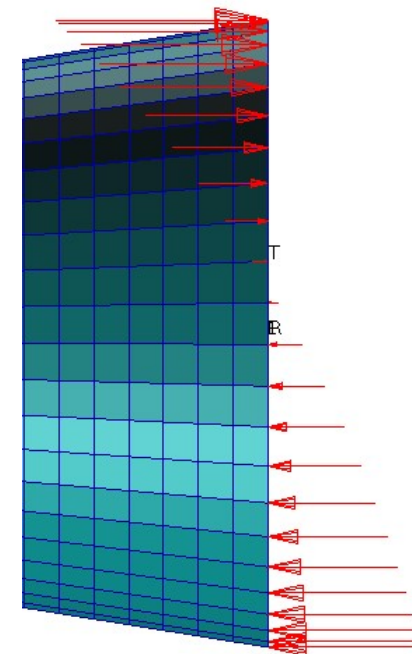
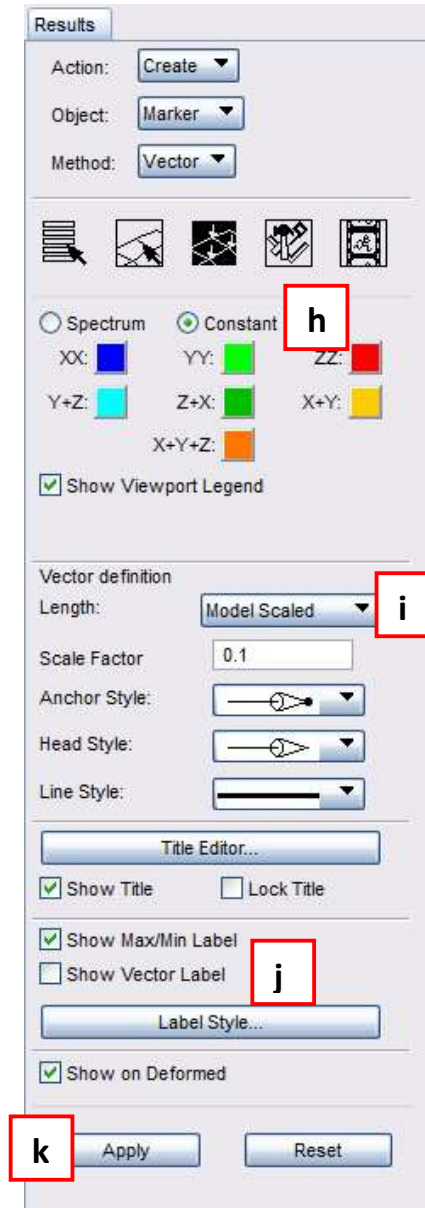
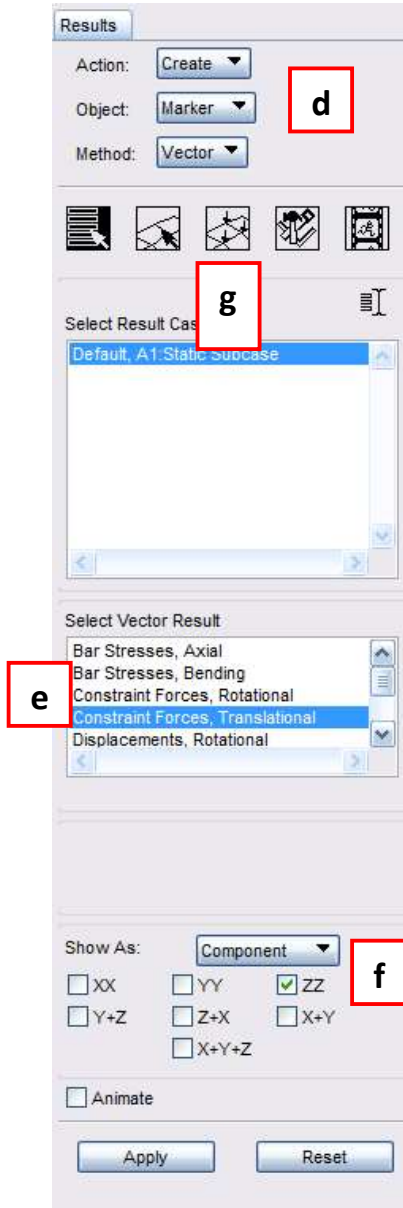
Position...(At Z1) **q**  
 Quantity: Z Component  
 X: Path Length

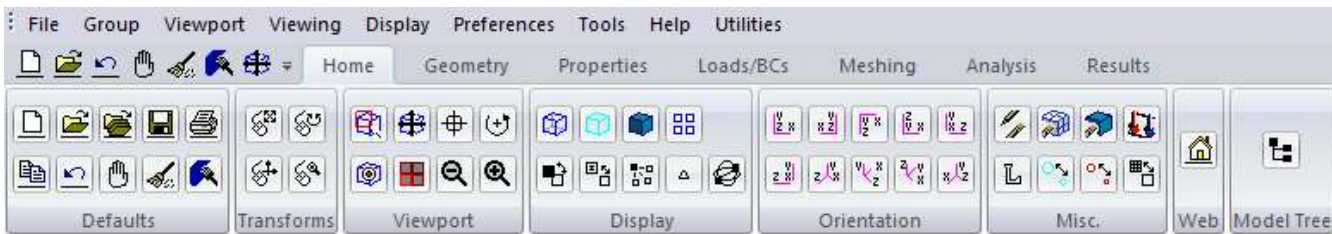
Apply Reset



Compare obtained graph with vector results:

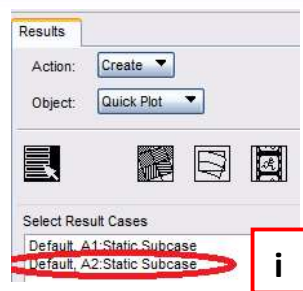
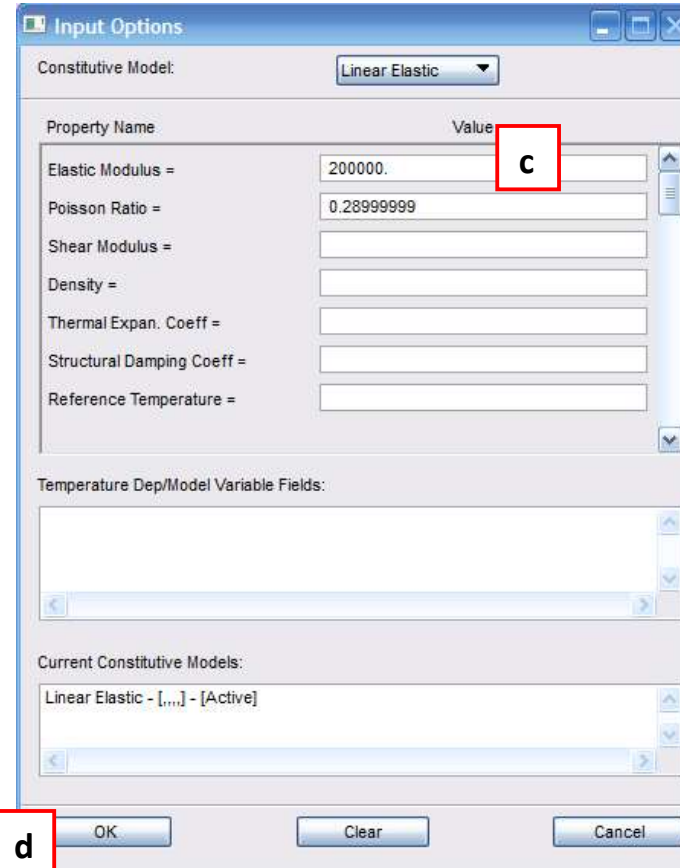
- Click on the **Reset graphics** icon
- Select **Right side view**
- Fit window**
- Click on **Results/Create/Marker/Vector**
- Select **Constraint Forces, Translational** for the Vector Result
- Change option to **Component** and toggle on only **ZZ** option
- Click on **Display Attributes**
- Toggle **Constant** vector color option and choose color for **ZZ** component
- Change **Length** to **Model Scaled**
- Toggle off **Show Vector Label**
- Click **Apply**



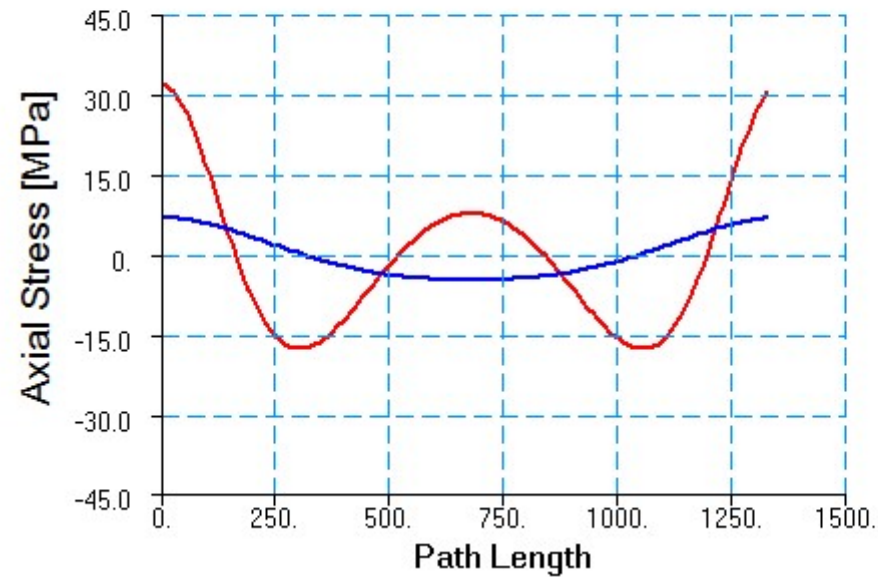
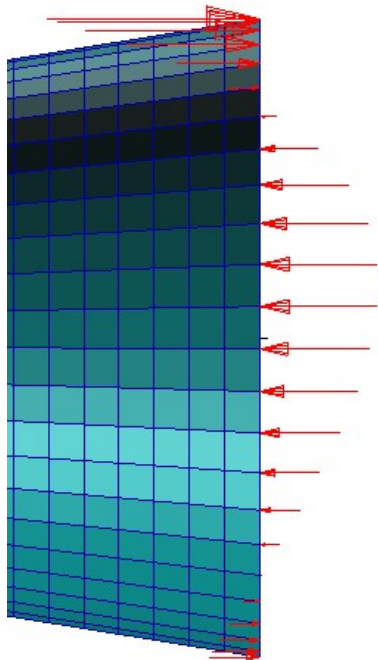
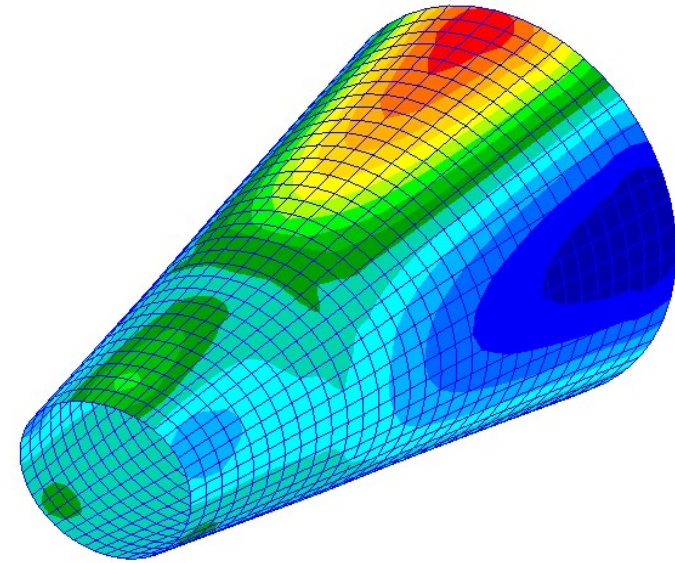
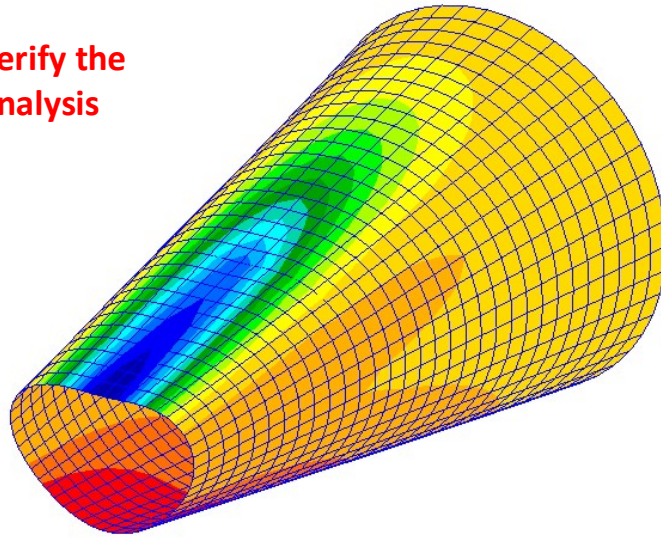


Change ring material properties, run second analysis and compare results with recently obtained:

- Go to **Properties** tab and select **Modify/Isotropic**
- Choose **ring\_mat**
- Change Young's Modulus to **2e5** (note that previous material was 100 times stiffer!)
- Click **OK**
- Click **Apply**
- In the **Analysis** tab, select **Analyze/Entire Model/Full Run**
- Change **Job Name** to **ex\_1b**
- Select **Apply**
- After analysis is done, **attach ex\_1b.xdb** file and **plot results** in the same manner as previously. Results referring to the second analysis will be tagged as **A2: Static subcase**



Use this reference to verify the results of the second analysis



What qualitative changes can be noticed?

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