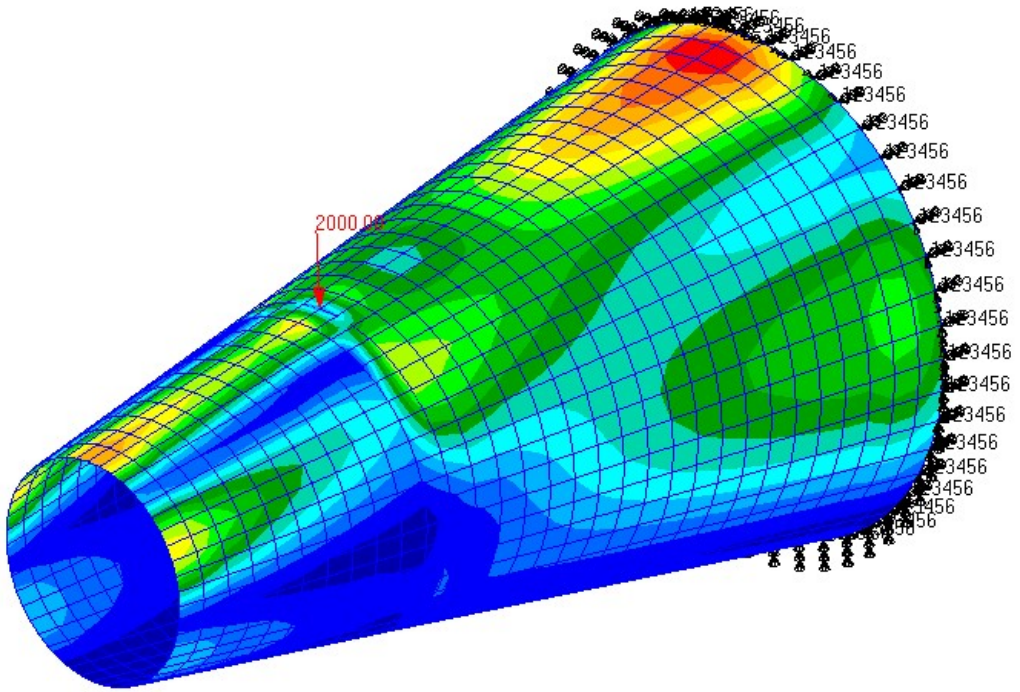
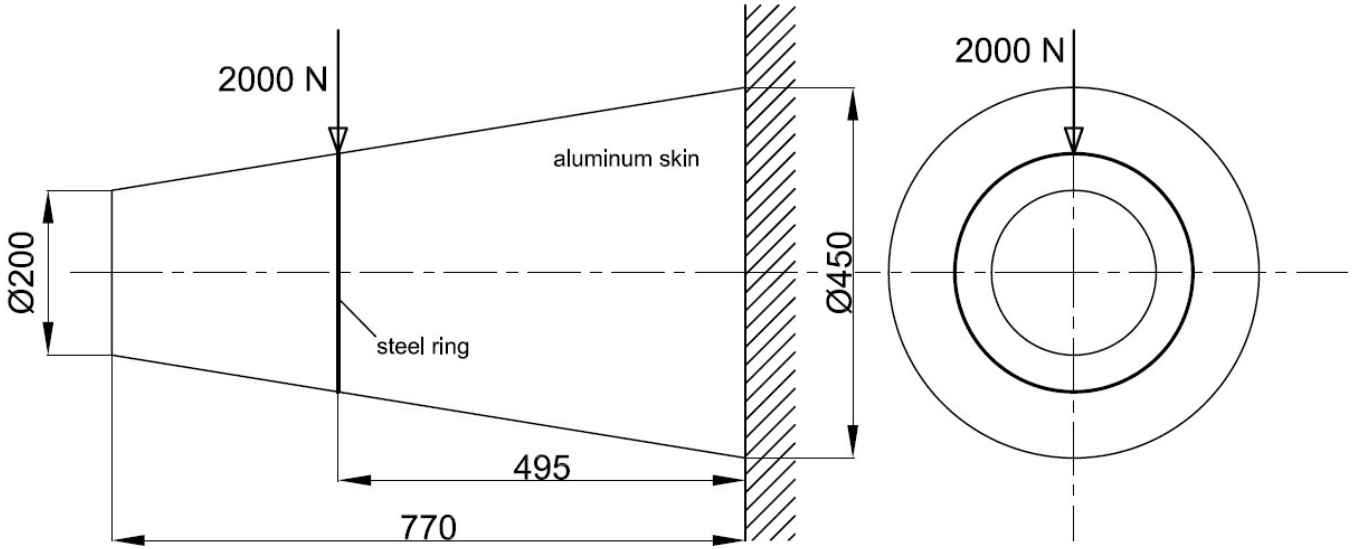
Mechanics of Thin-walled Structures



PROBLEM DESCRIPTION



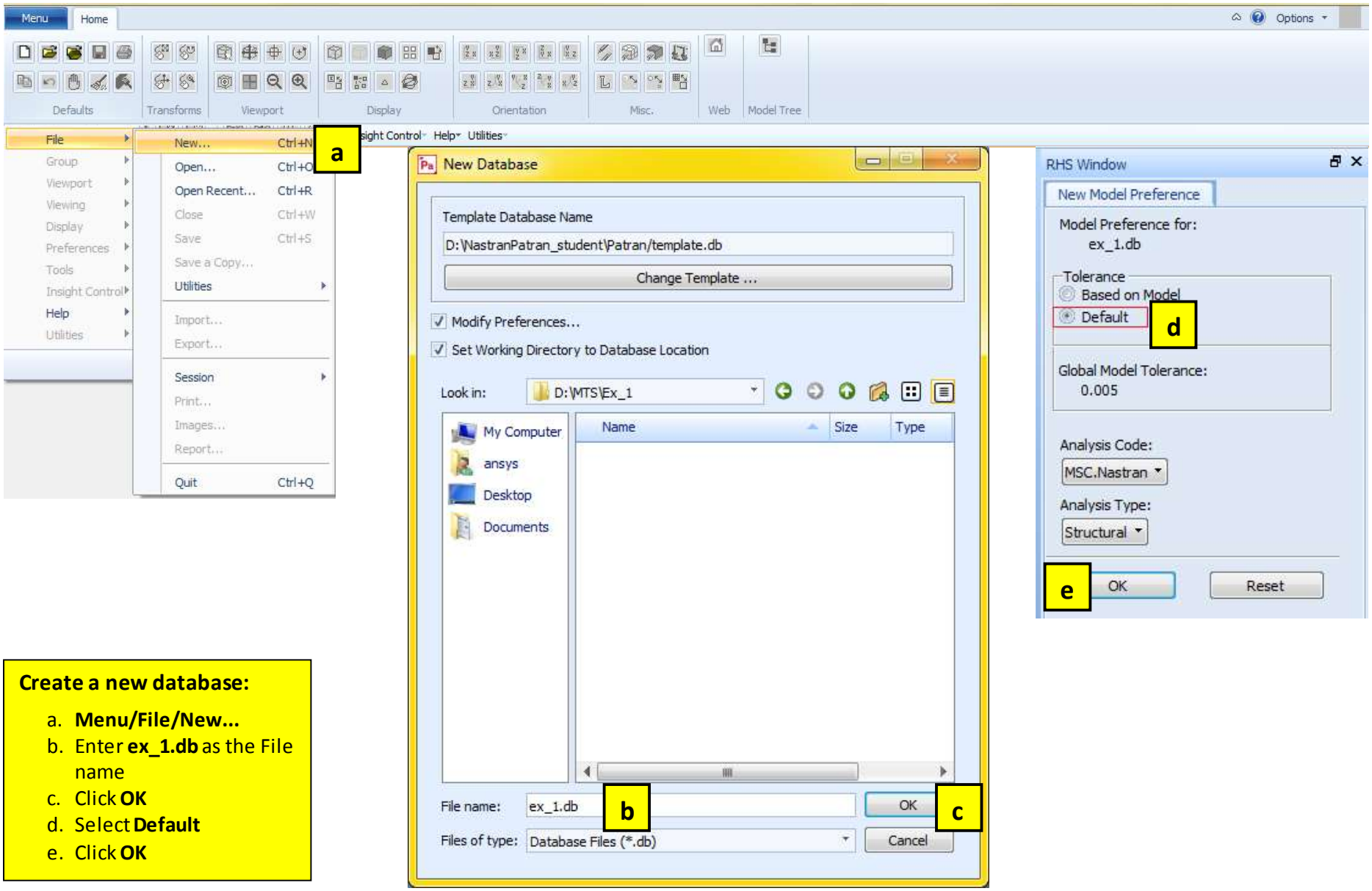
Static, linear analysis of thin-walled aluminum skin, reinforced by a stiffening ring, will be performed.

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:

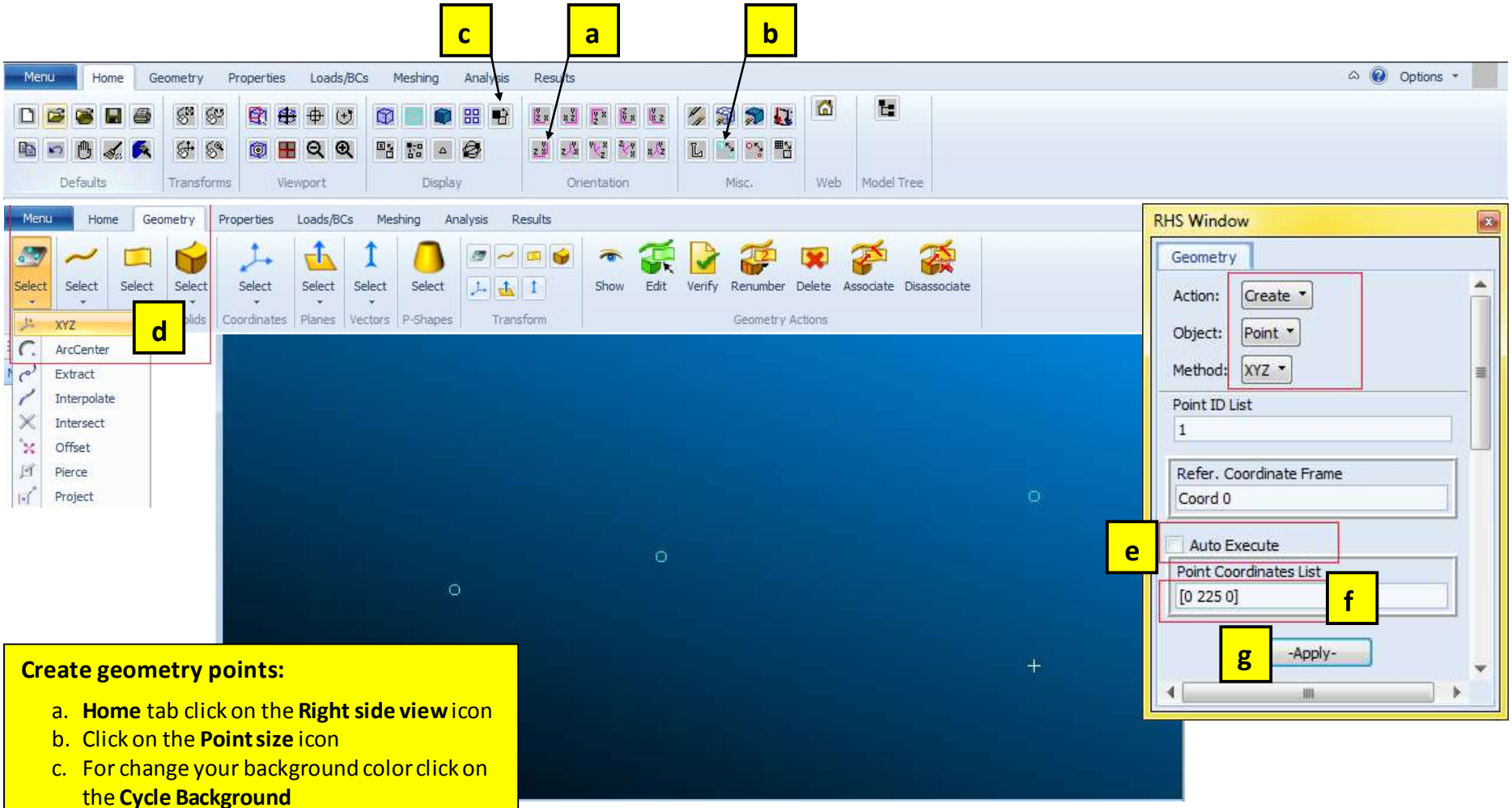
1. When stiffening ring is (relatively) **rigid**
2. When stiffening ring is made of **steel** (and its deformations influence the deformation of the aluminum skin)

Units: mm, N, MPa



Create a new database:

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

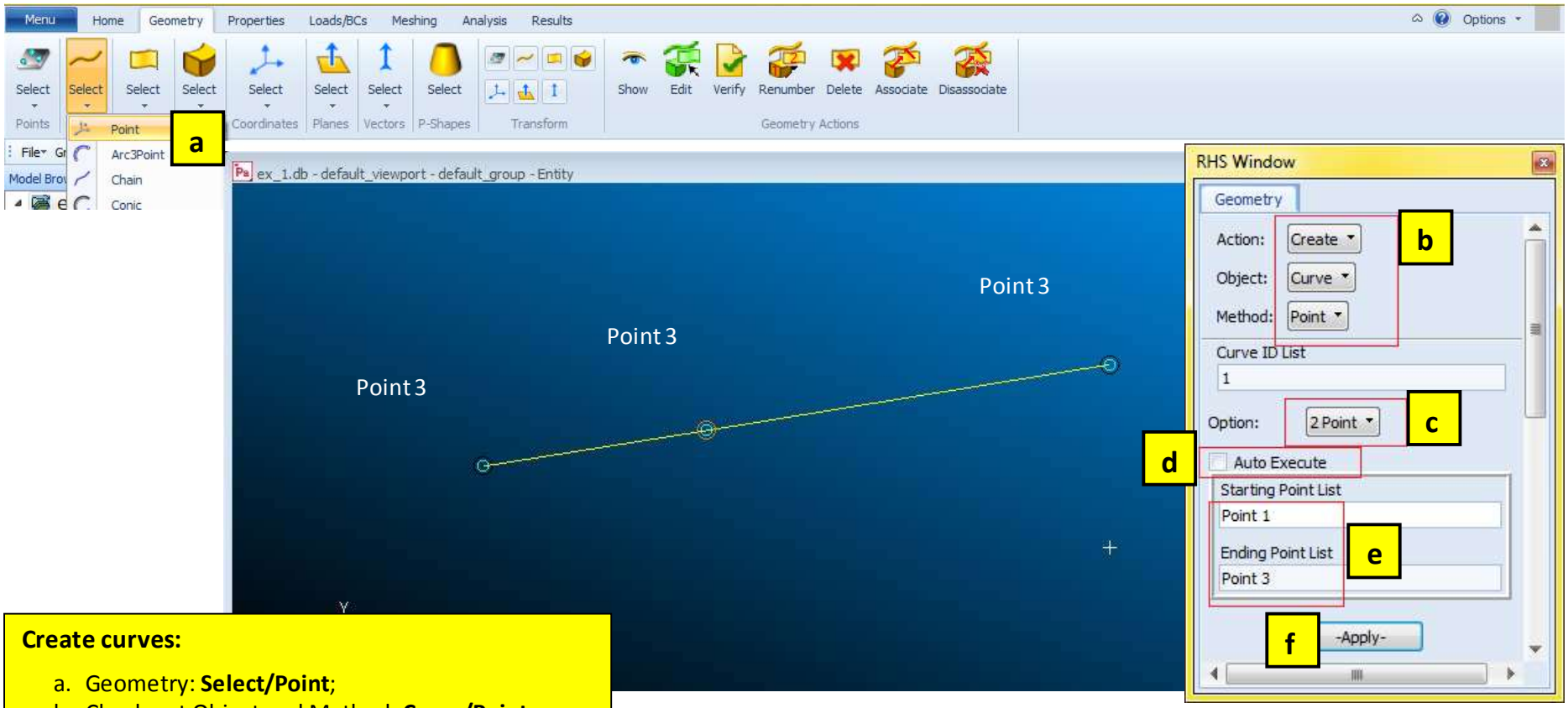


Create geometry points:

a. **Home** tab click on the **Right side view** icon
b. Click on the **Point size** icon
c. For change your background color click on the **Cycle Background**
d. Geometry icon: **Select/XYZ** *(Points icon)*
e. Uncheck **Auto Execute**
f. Enter **[0 225 0]** as the Point Coordinates List
g. Click **Apply**
h. Create two more points using coordinates: **[0 100 770]**, **[0 144 495]**

WARNING!

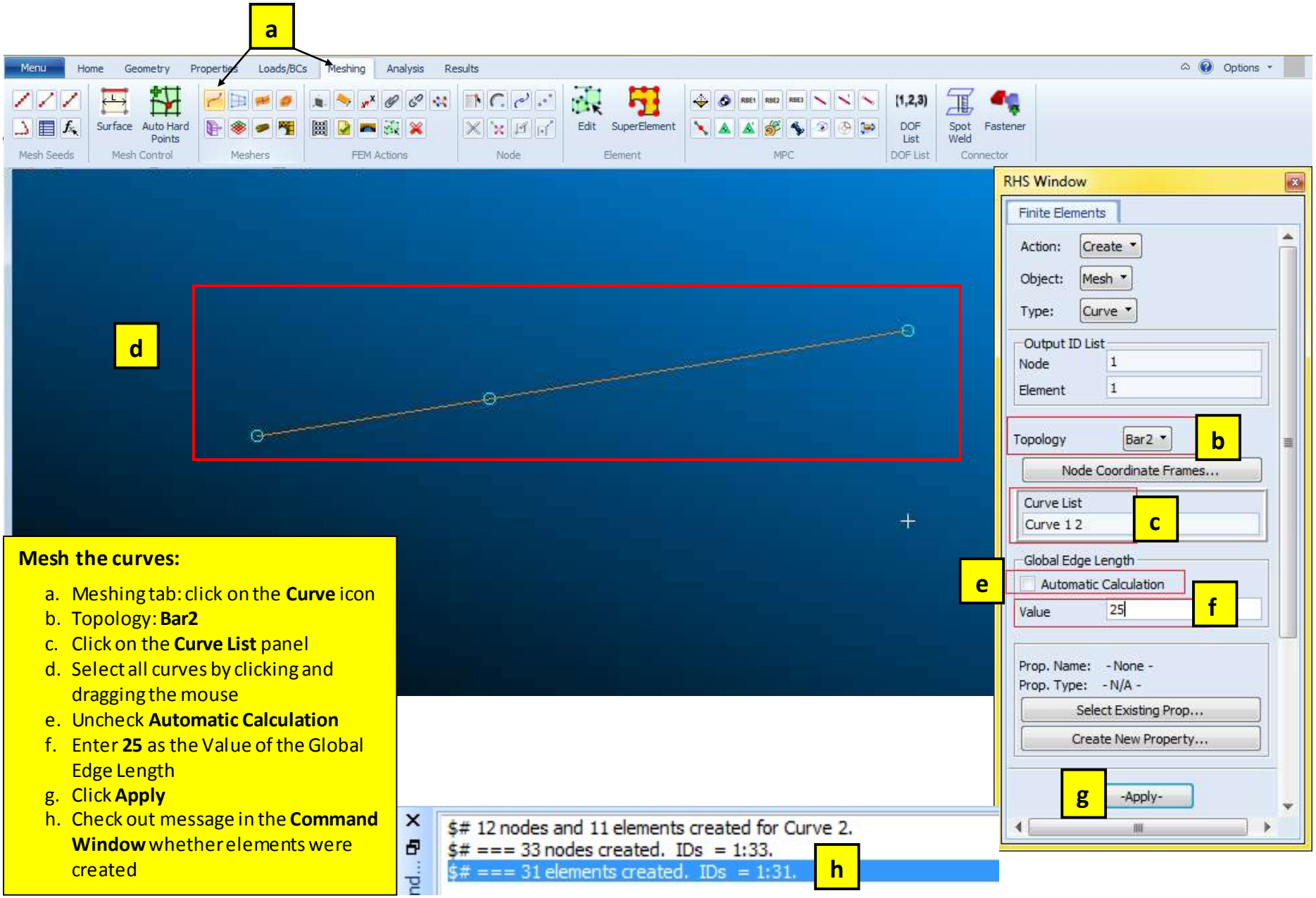
The **Auto Execute** option will automatically apply the operation when the last field is filled in.



Create curves:

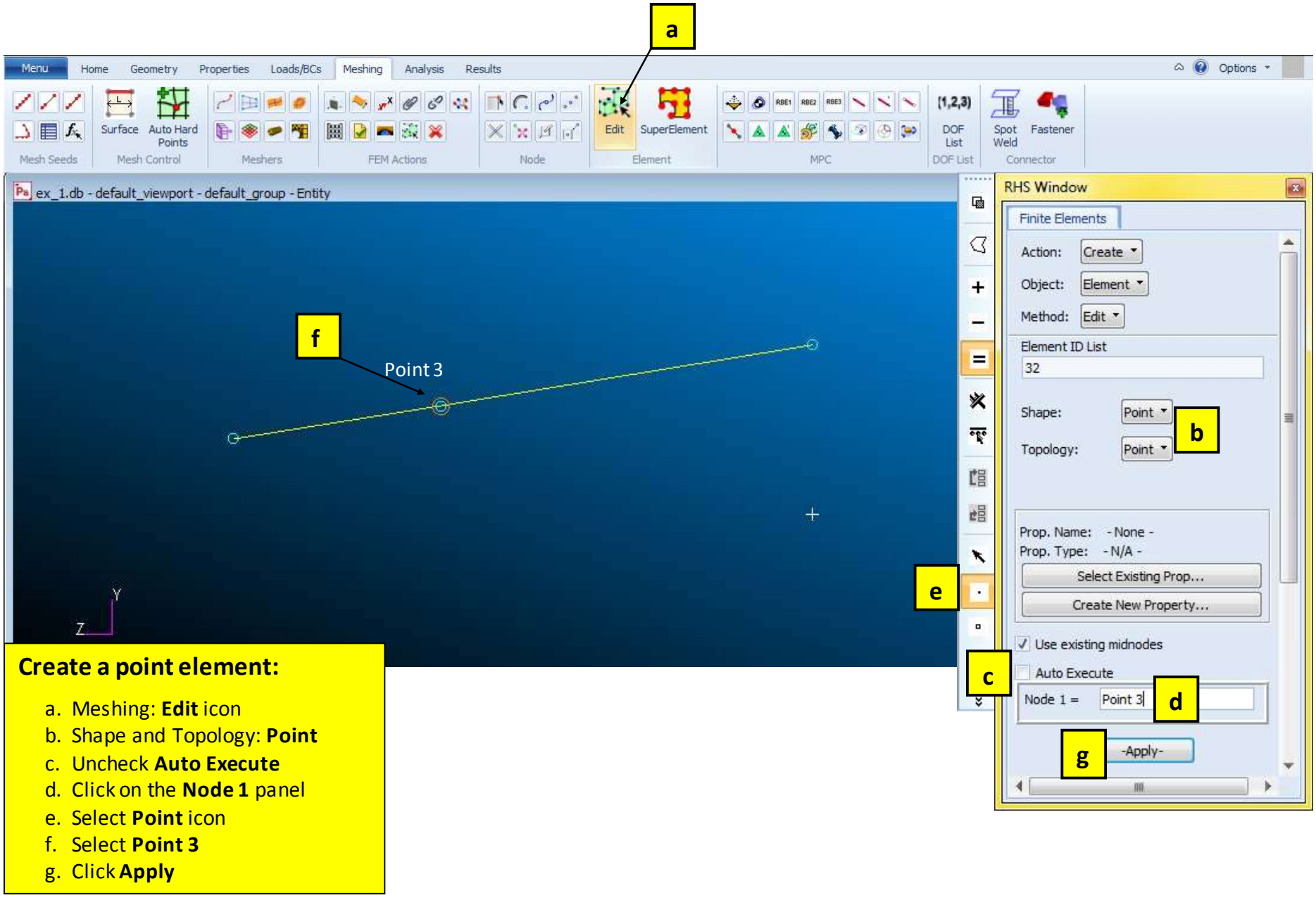
a. Geometry: **Select/Point**;
b. Check out Object and Method: **Curve/Point**
c. Option: **2 Point**;
d. Uncheck **Auto Execute**;
e. Select **Point 1** as the starting point and **Point 3** as the ending point;
f. Click **Apply**;
g. Create one more curve using **Point 3** and **Point 2**.

Remark: Points numbers may be different.



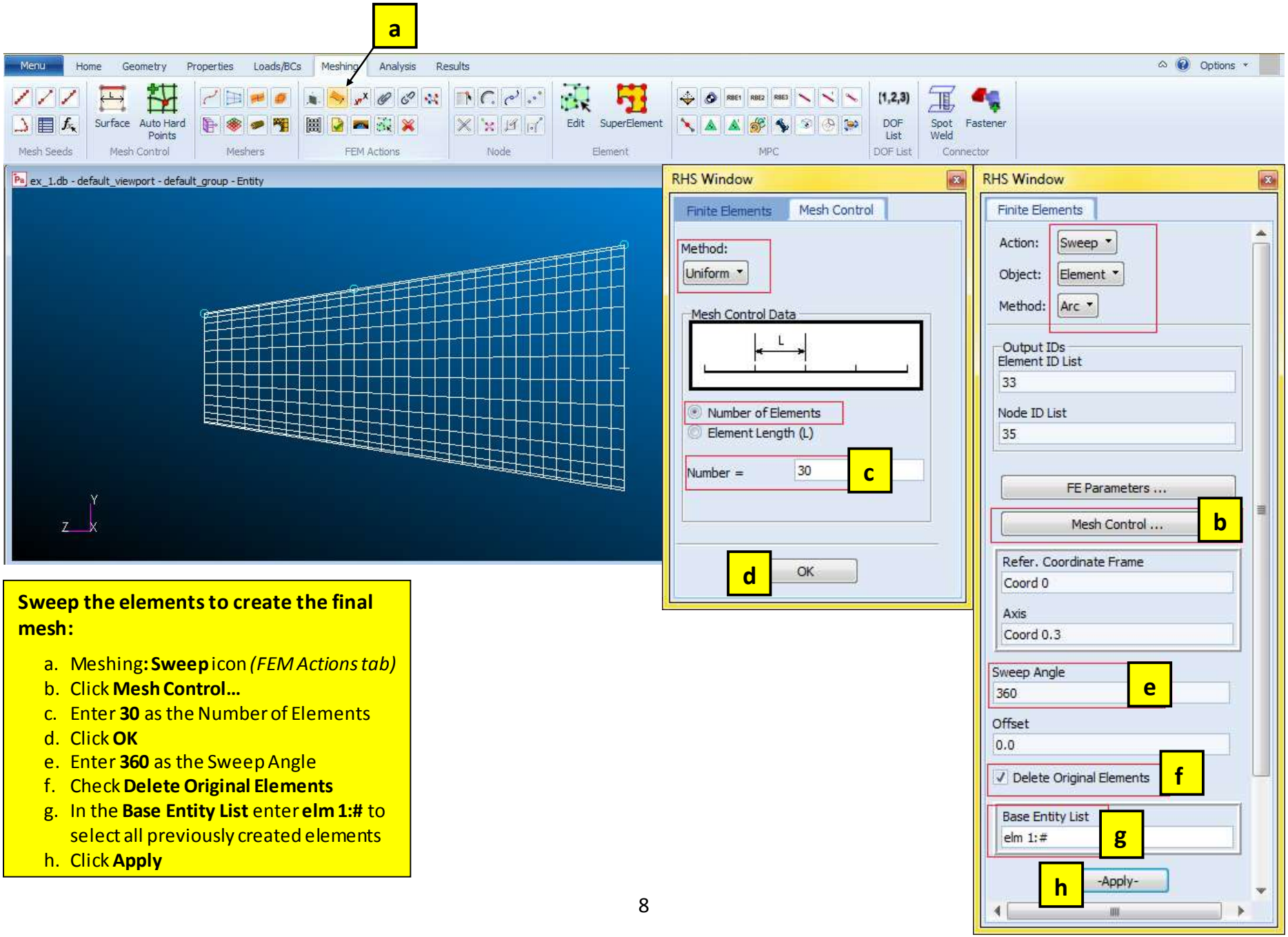
Mesh the curves:

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



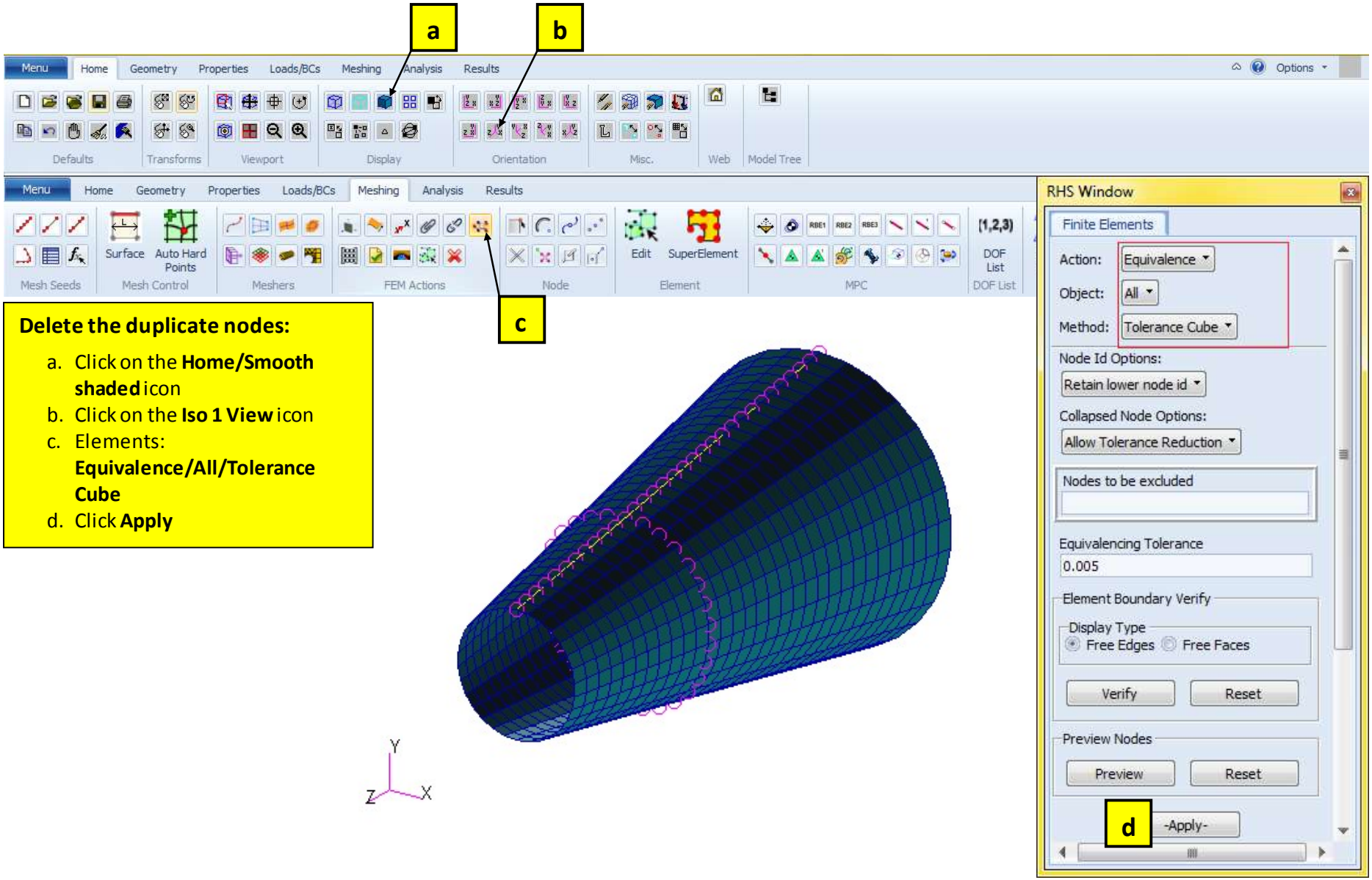
Create a point element:

a. Meshing: **Edit** icon
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**



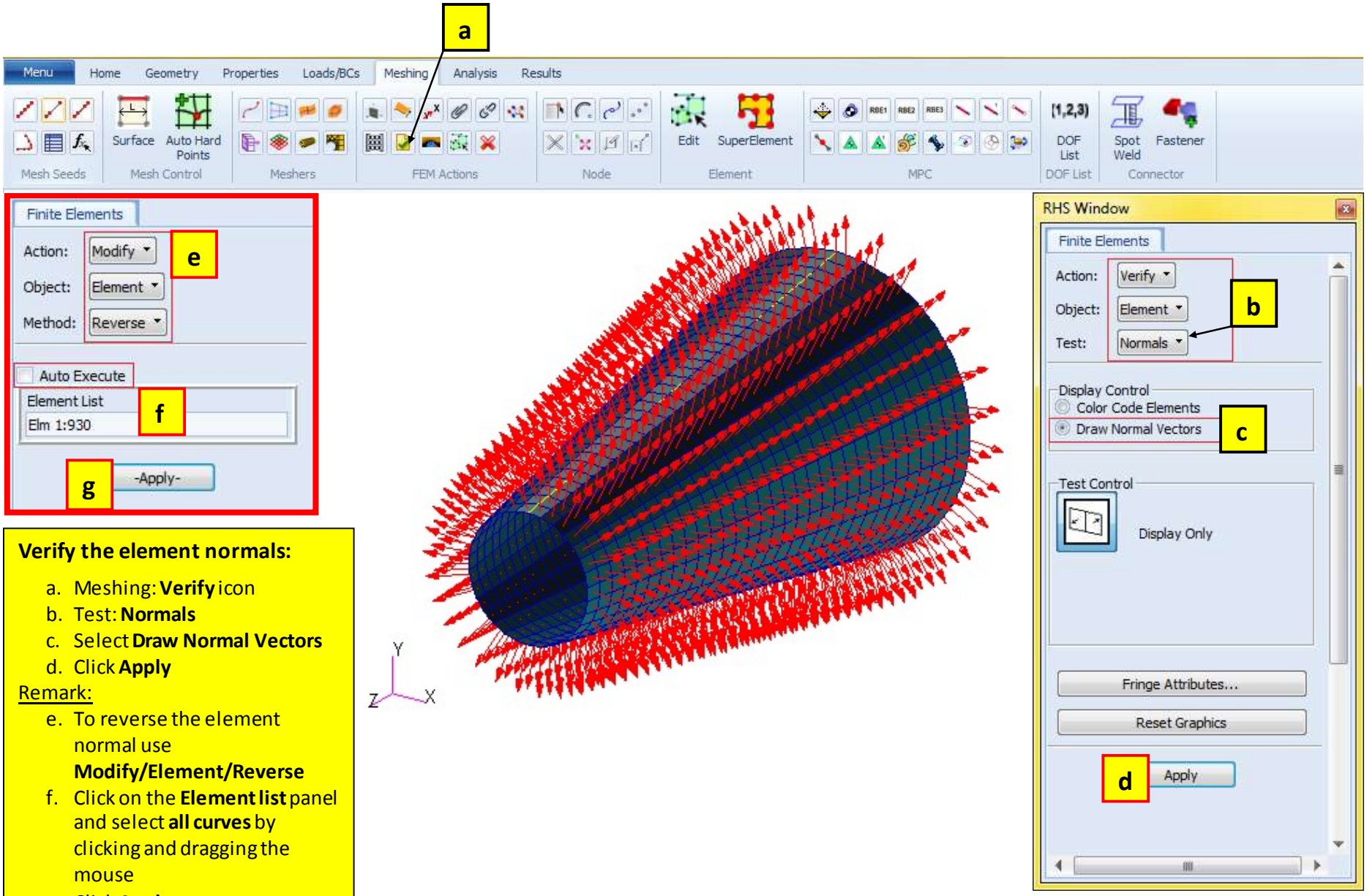
Sweep the elements to create the final mesh:

a. Meshing: **Sweep** icon *(FEM Actions tab)*
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 **Home/Smooth shaded** icon
b. Click on the **Iso 1 View** icon
c. Elements: **Equivalence/All/Tolerance Cube**
d. Click **Apply**

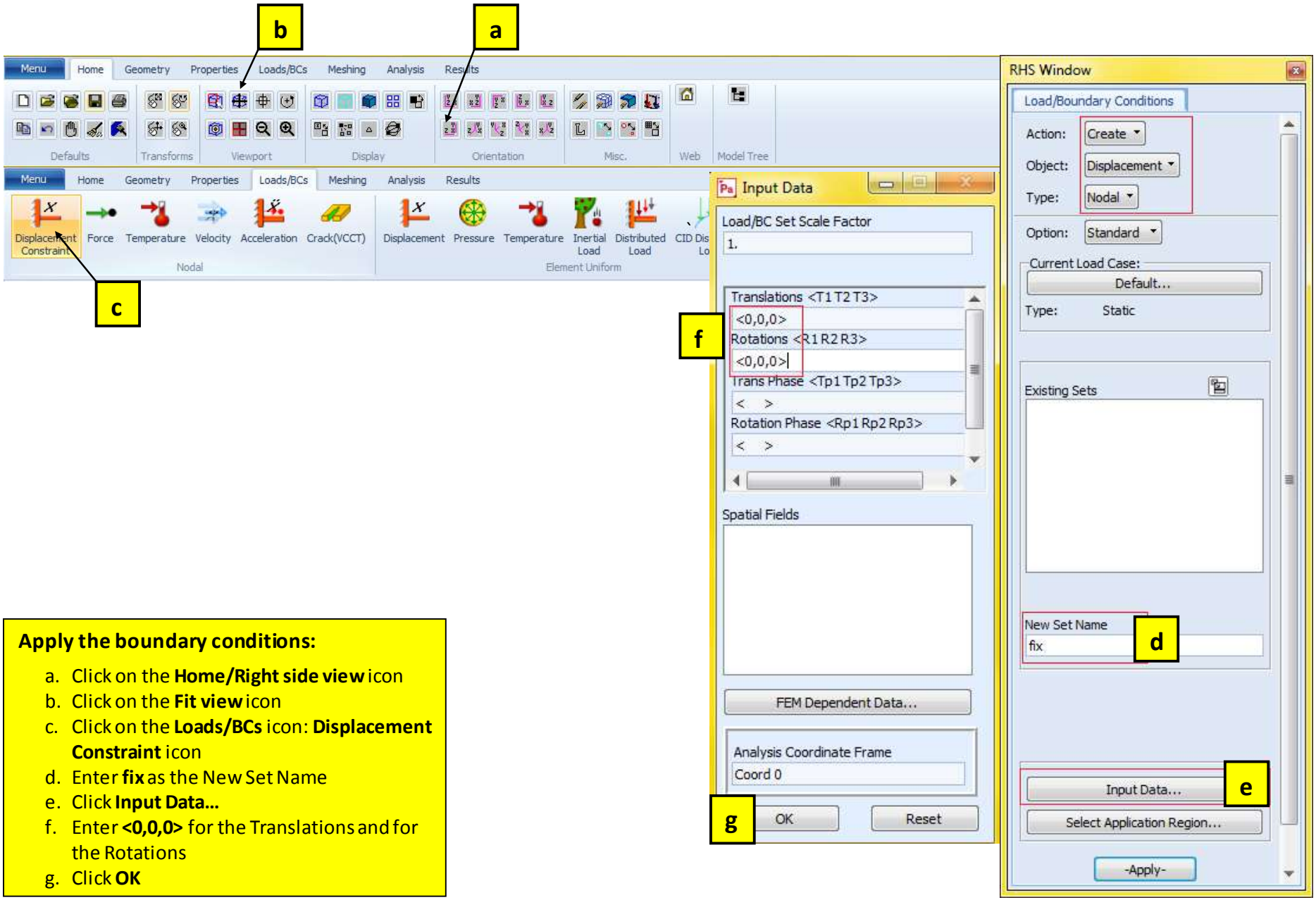


Verify the element normals:

a. Meshing: **Verify** icon
b. Test: **Normals**
c. Select **Draw Normal Vectors**
d. Click **Apply**

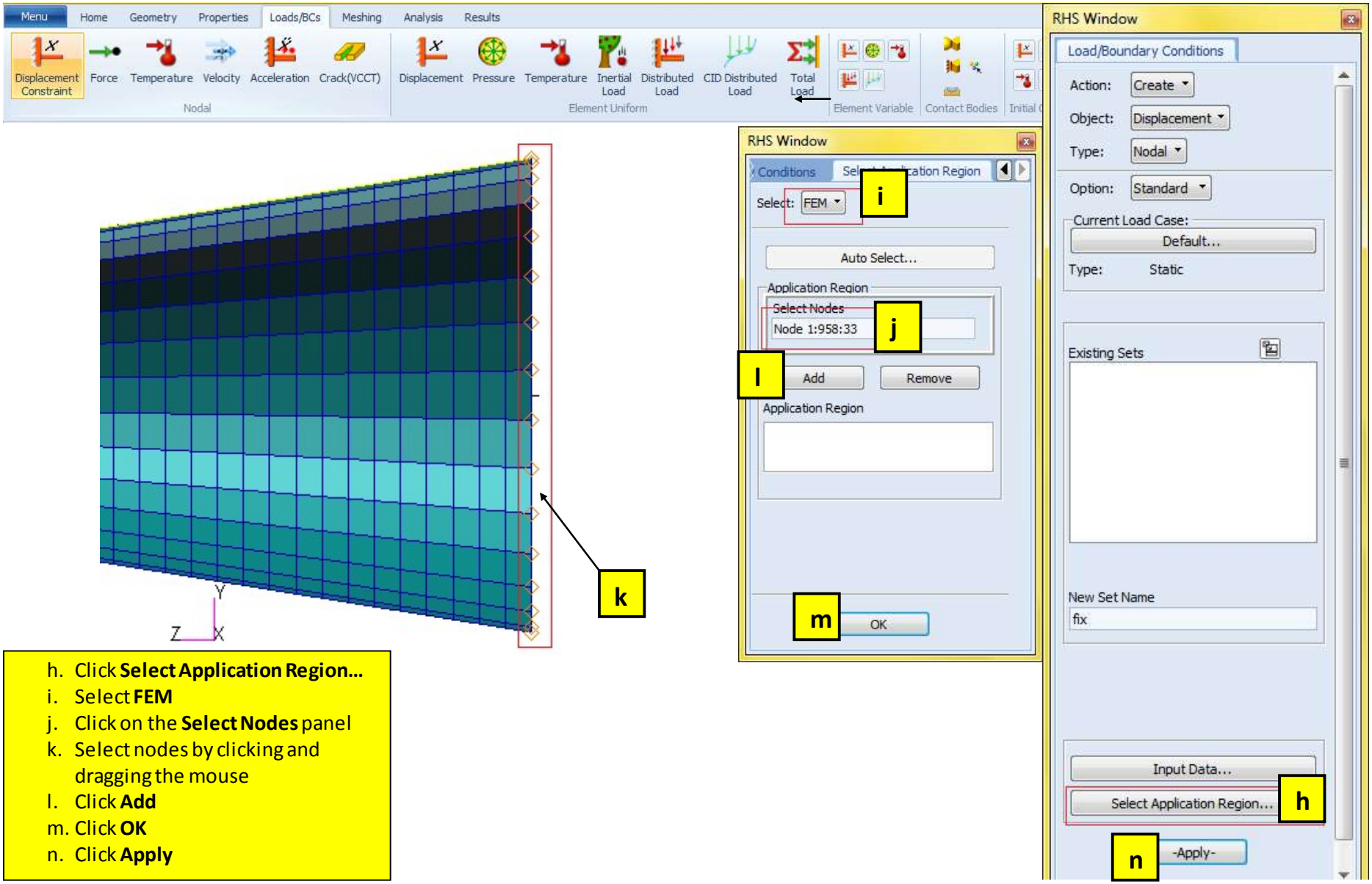
Remark:

e. To reverse the element normal use **Modify/Element/Reverse**
f. Click on the **Element list** panel and select **all curves** by clicking and dragging the mouse
g. Click **Apply**

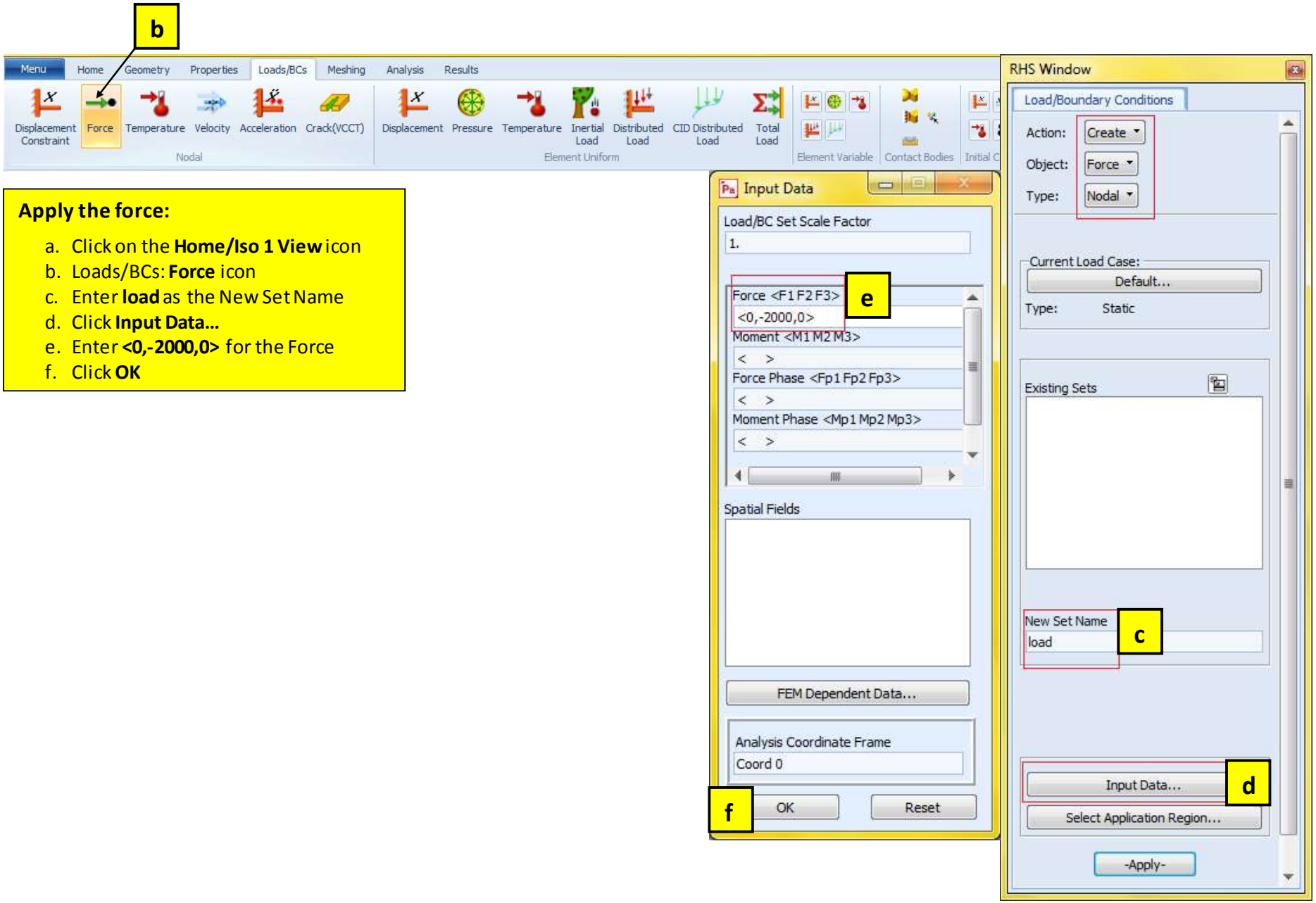


Apply the boundary conditions:

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

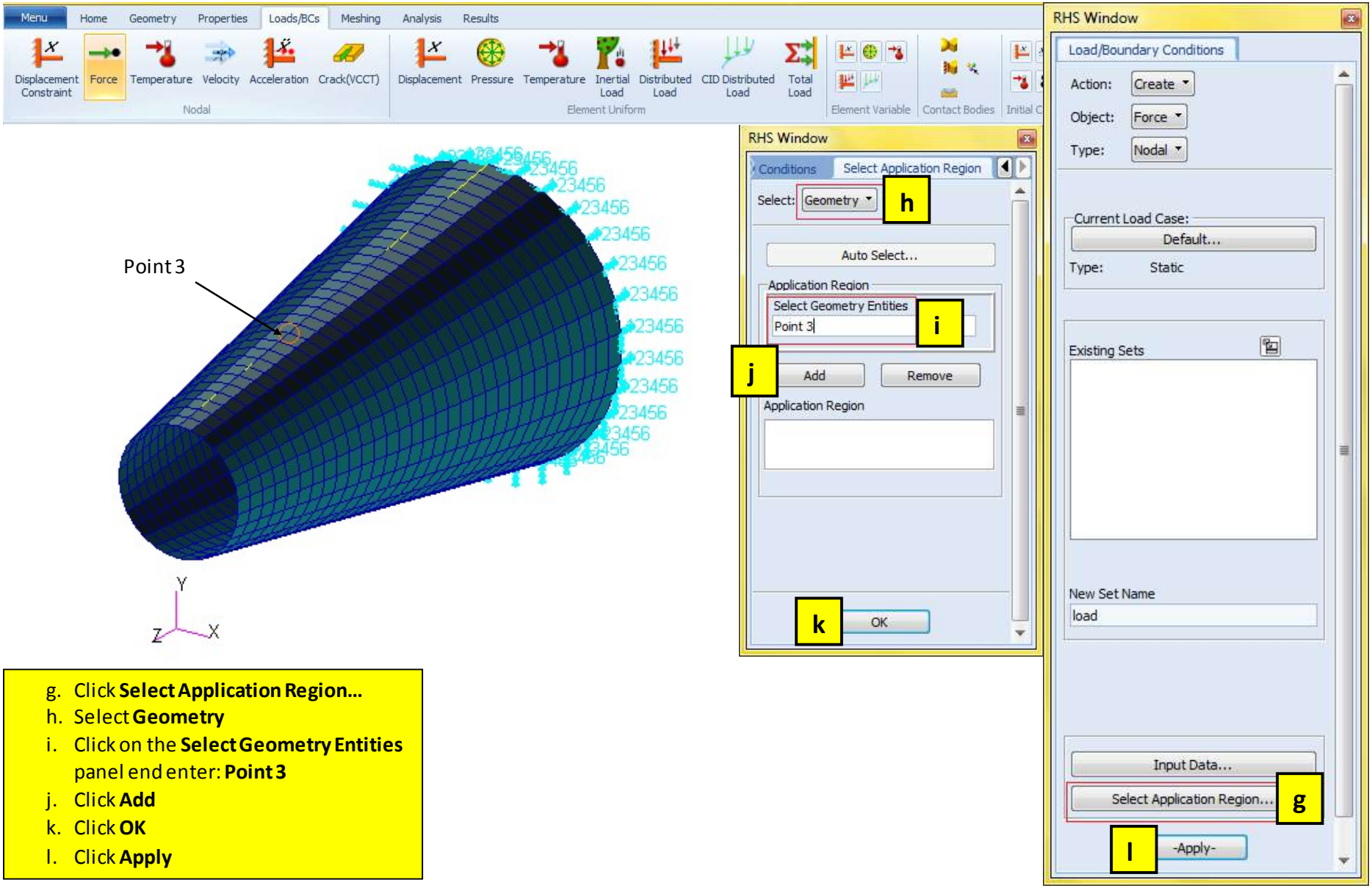


h. Click **Select Application Region...**
i. Select **FEM**
j. Click on the **Select Nodes** panel
k. Select nodes by clicking and dragging the mouse
l. Click **Add**
m. Click **OK**
n. Click **Apply**

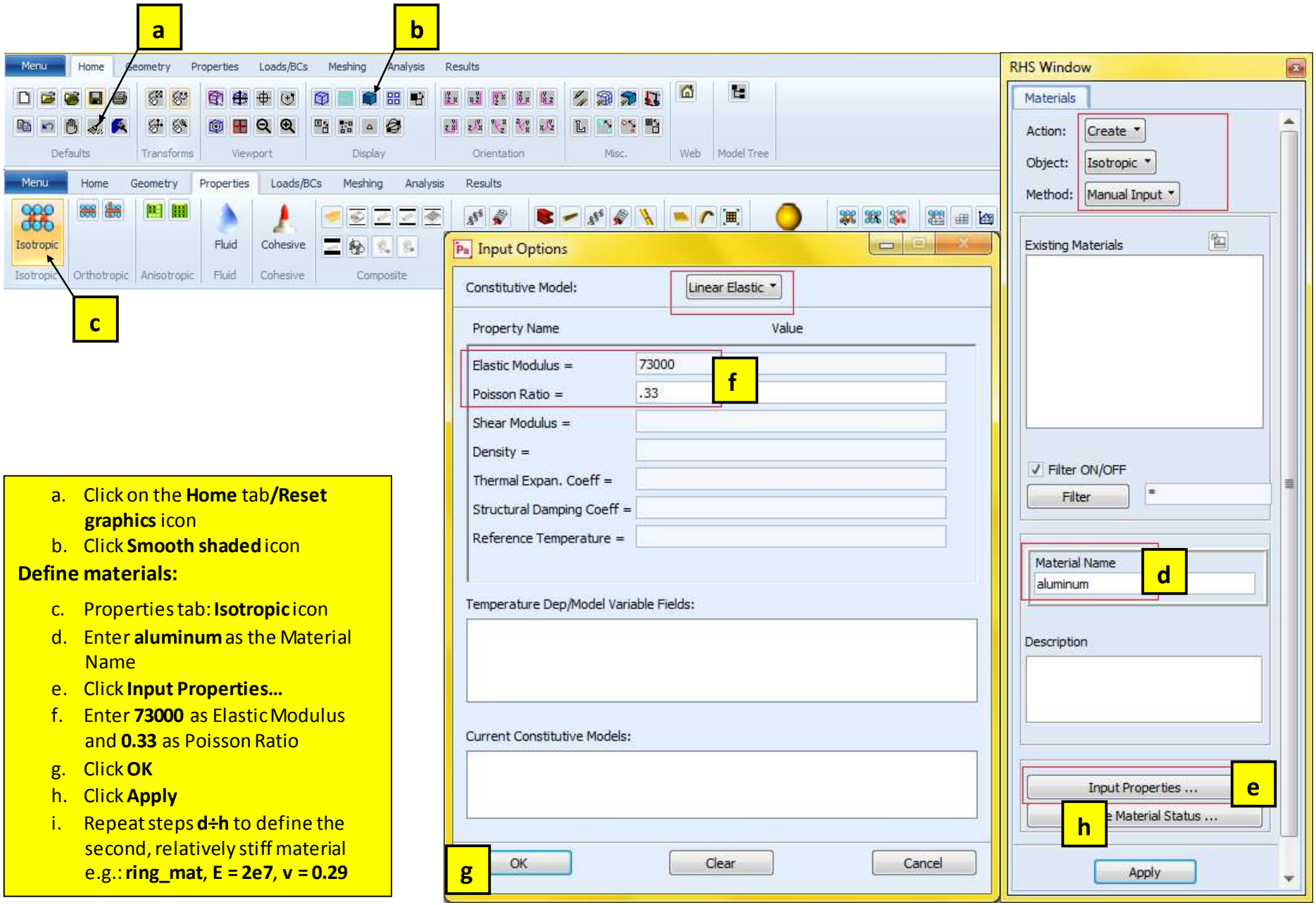


Apply the force:

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



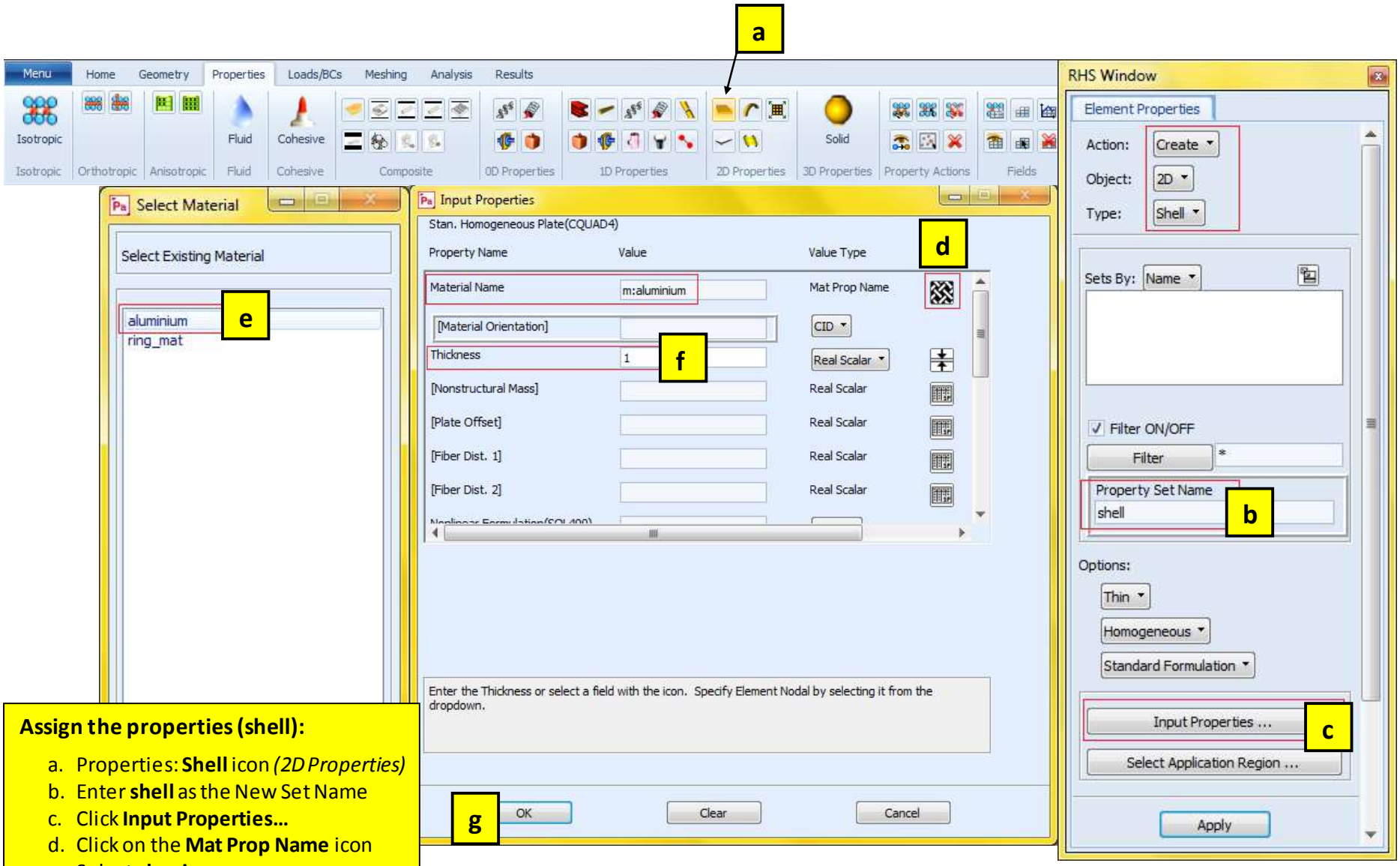
g. Click **Select Application Region…**
h. Select **Geometry**
i. Click on the **Select Geometry Entities** panel end enter: **Point 3**
j. Click **Add**
k. Click **OK**
l. Click **Apply**



a. Click on the **Home** tab**/Reset graphics** icon
b. Click **Smooth shaded** icon

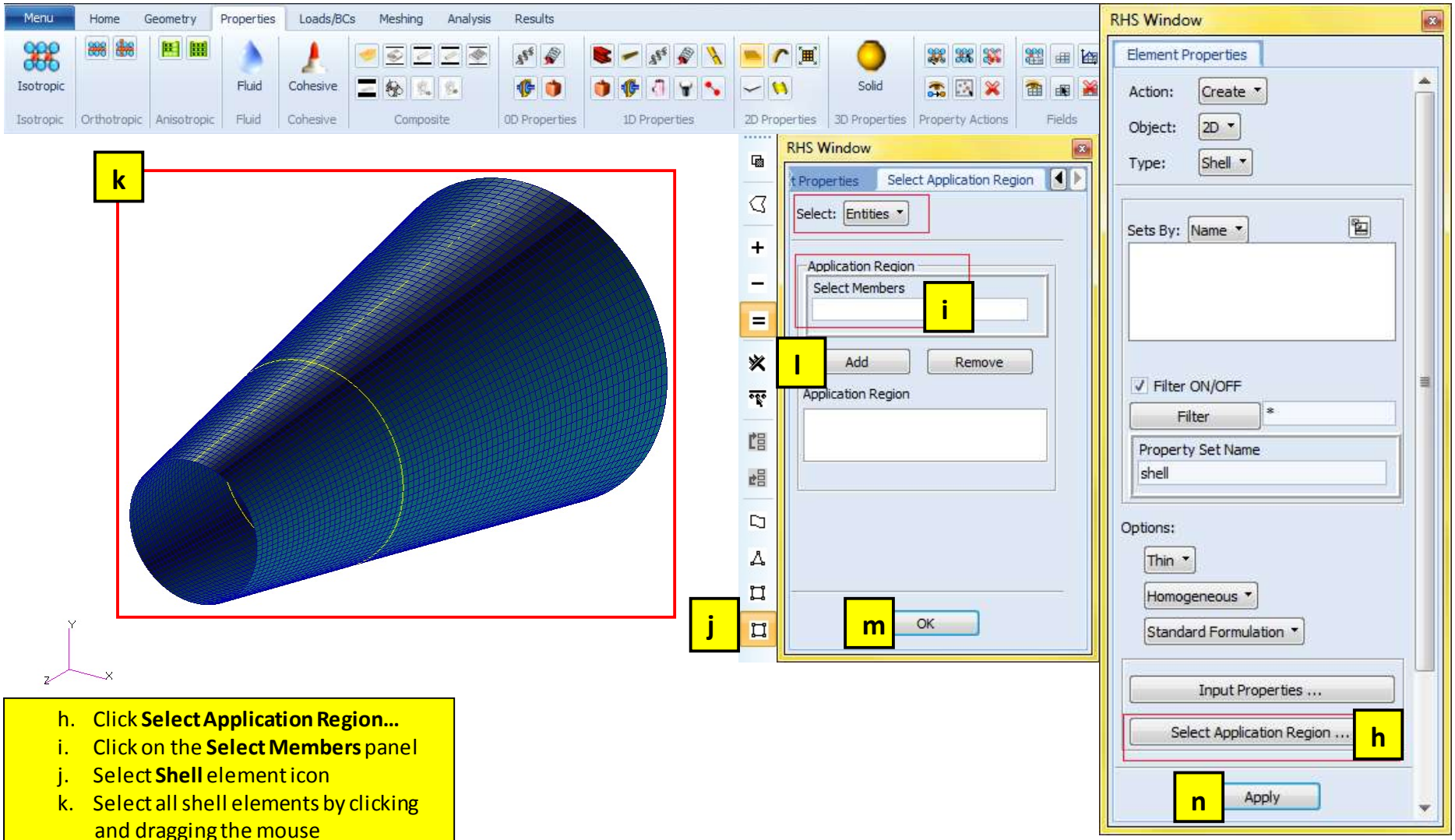
Define materials:

c. Properties tab: **Isotropic** icon
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 **d÷h** to define the second, relatively stiff material e.g.: **ring_mat, E = 2e7, v = 0.29**

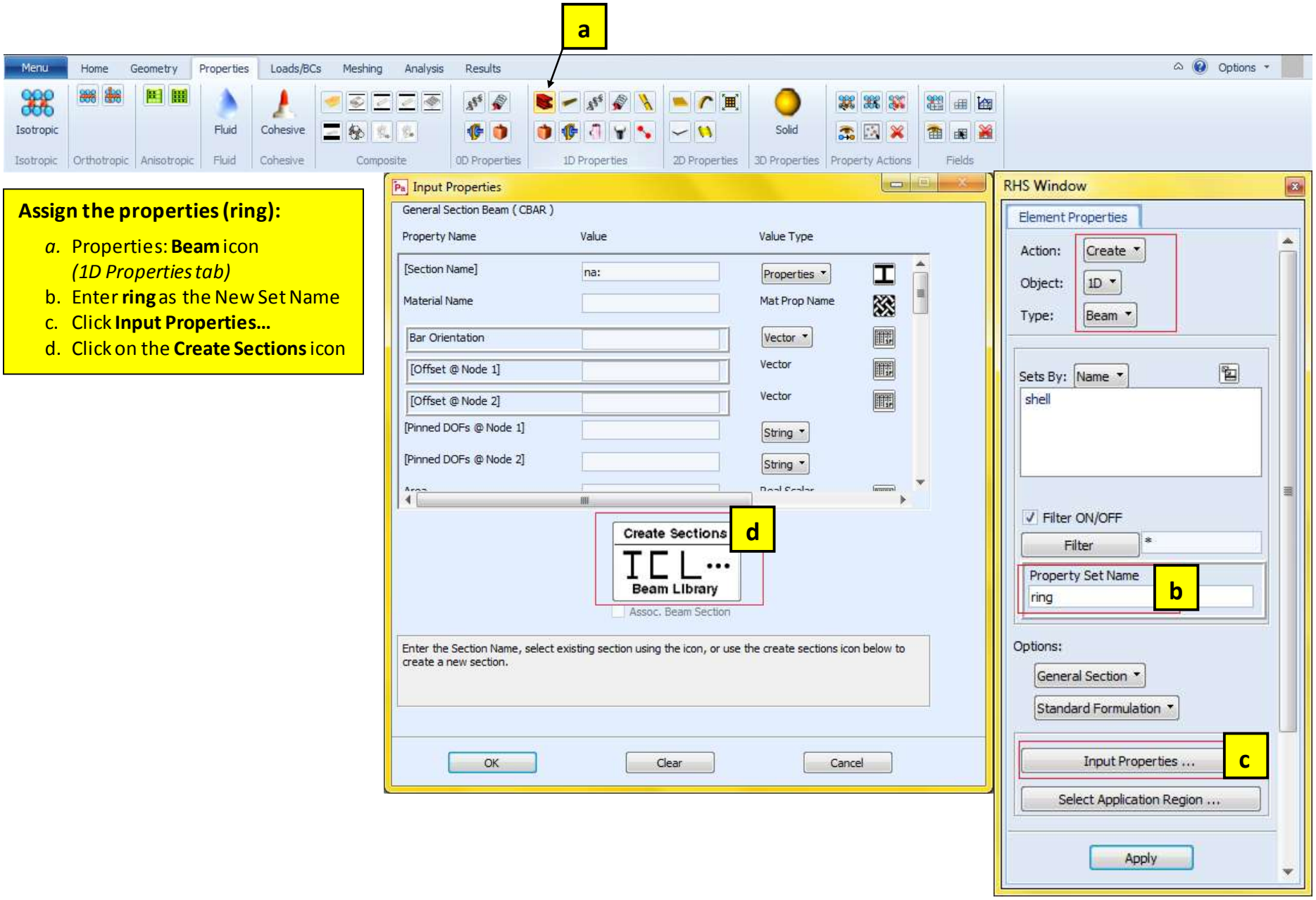


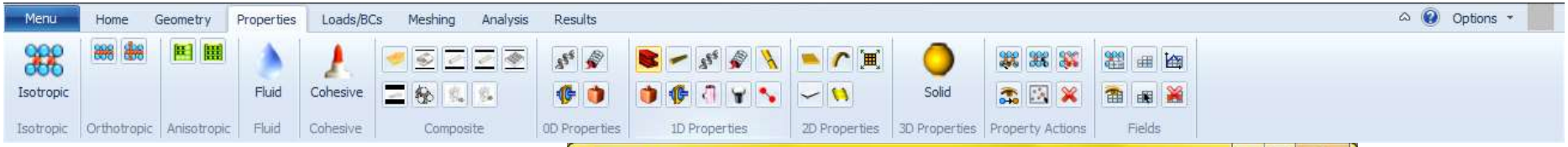
Assign the properties (shell):

a. Properties: **Shell** icon *(2D Properties)*
b. Enter **shell** as the New Set Name
c. Click **Input Properties…**
d. Click on the **Mat Prop Name** icon
e. Select **aluminum**
f. Enter **1** as the Thickness
g. Click **OK**

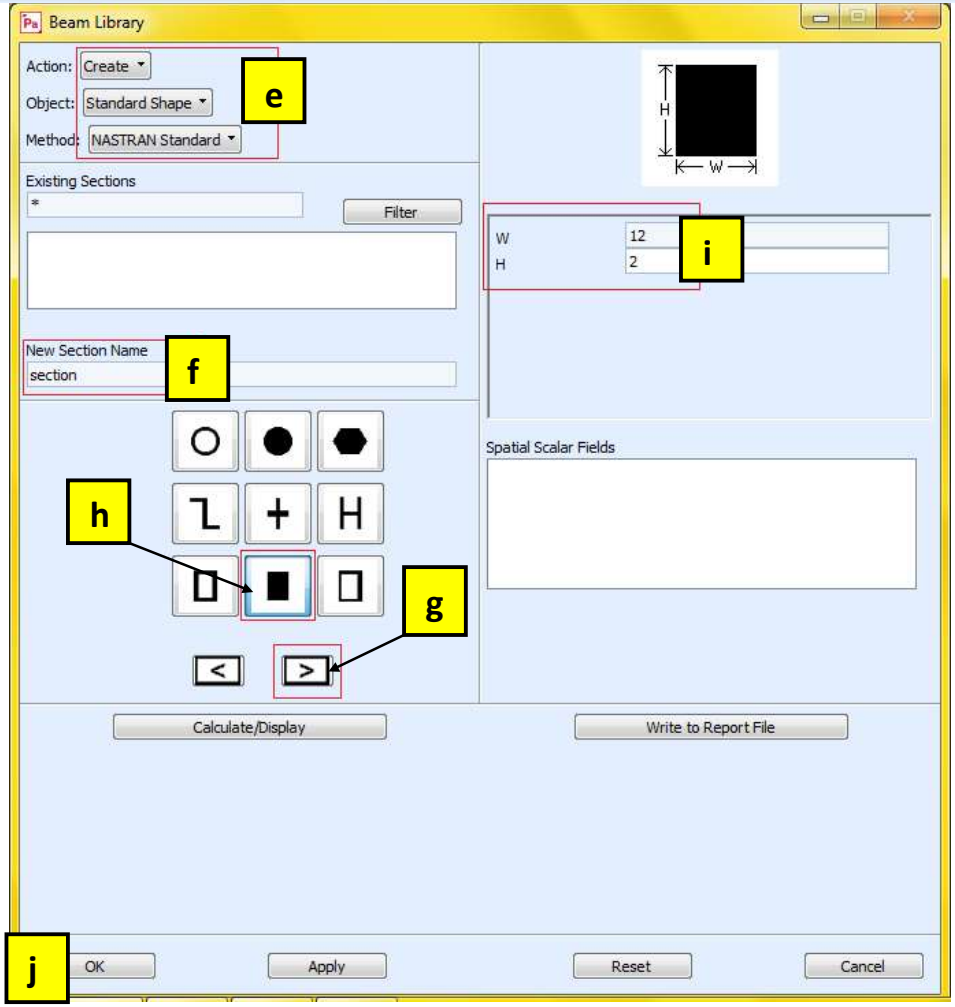


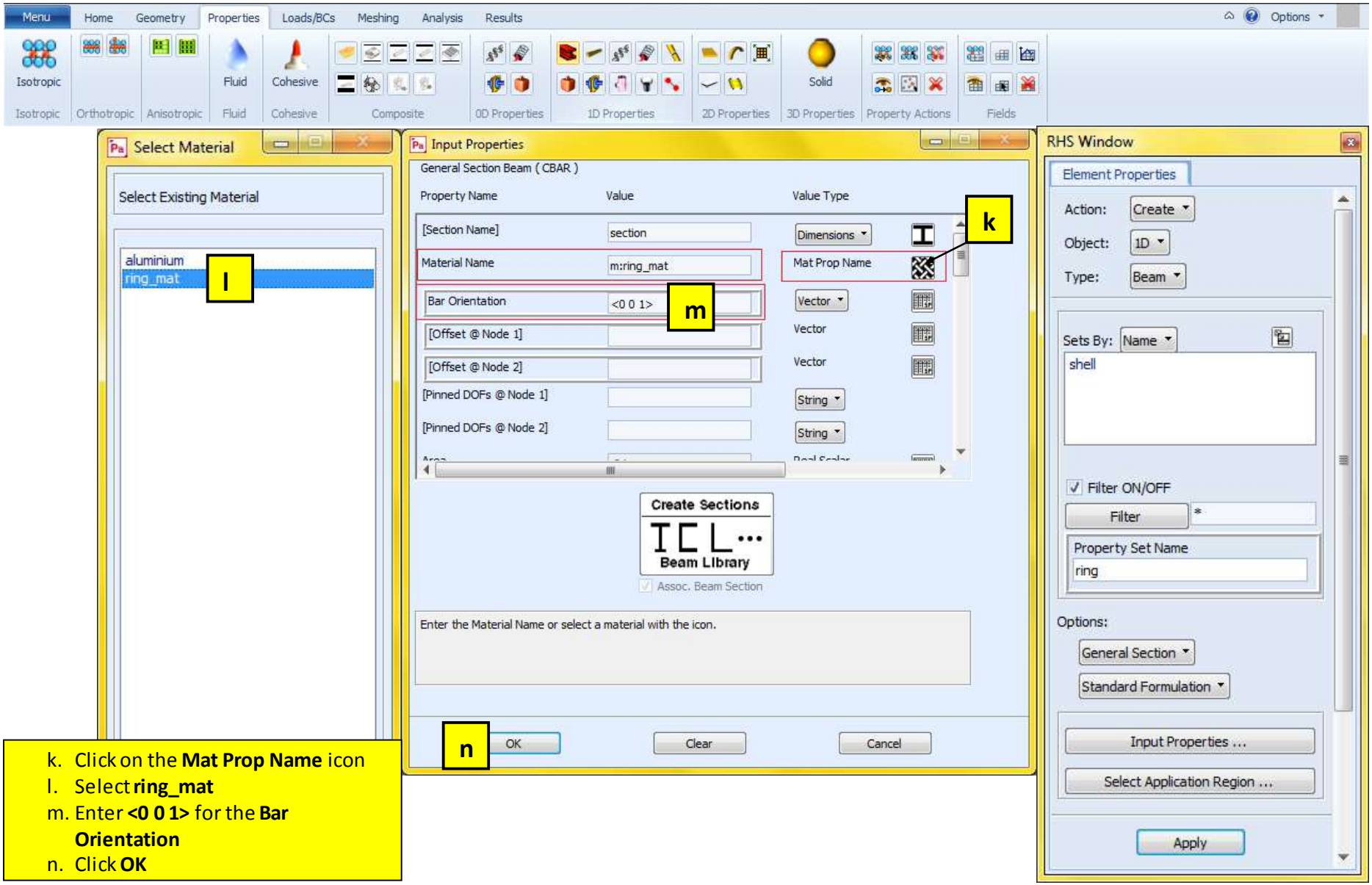
h. Click **Select Application Region...**
i. Click on the **Select Members** panel
j. Select **Shell** element icon
k. Select all shell elements by clicking and dragging the mouse
l. Click **Add**
m. Click **OK**
n. Click **Apply**


a
Menu
Home
Geometry
Properties
Loads/BCs
Meshing
Analysis
Results
Options
Isotropic
Orthotropic
Anisotropic
Fluid
Cohesive
Composite
0D Properties
1D Properties
2D Properties
Solid
3D Properties
Property Actions
Fields
Assign the properties (ring):
a. Properties: Beam icon (1D Properties tab)
b. Enter ring as the New Set Name
c. Click Input Properties...
d. Click on the Create Sections icon
Input Properties
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
Create Sections
d
Beam Library
Assoc. Beam Section
Enter the Section Name, select existing section using the icon, or use the create sections icon below to create a new section.
OK
Clear
Cancel
RHS Window
Element Properties
Action:
Create
Object:
1D
Type:
Beam
Sets By:
Name
shell
Filter ON/OFF
Filter
*
Property Set Name
ring
b
Options:
General Section
Standard Formulation
Input Properties ...
c
Select Application Region ...
Apply

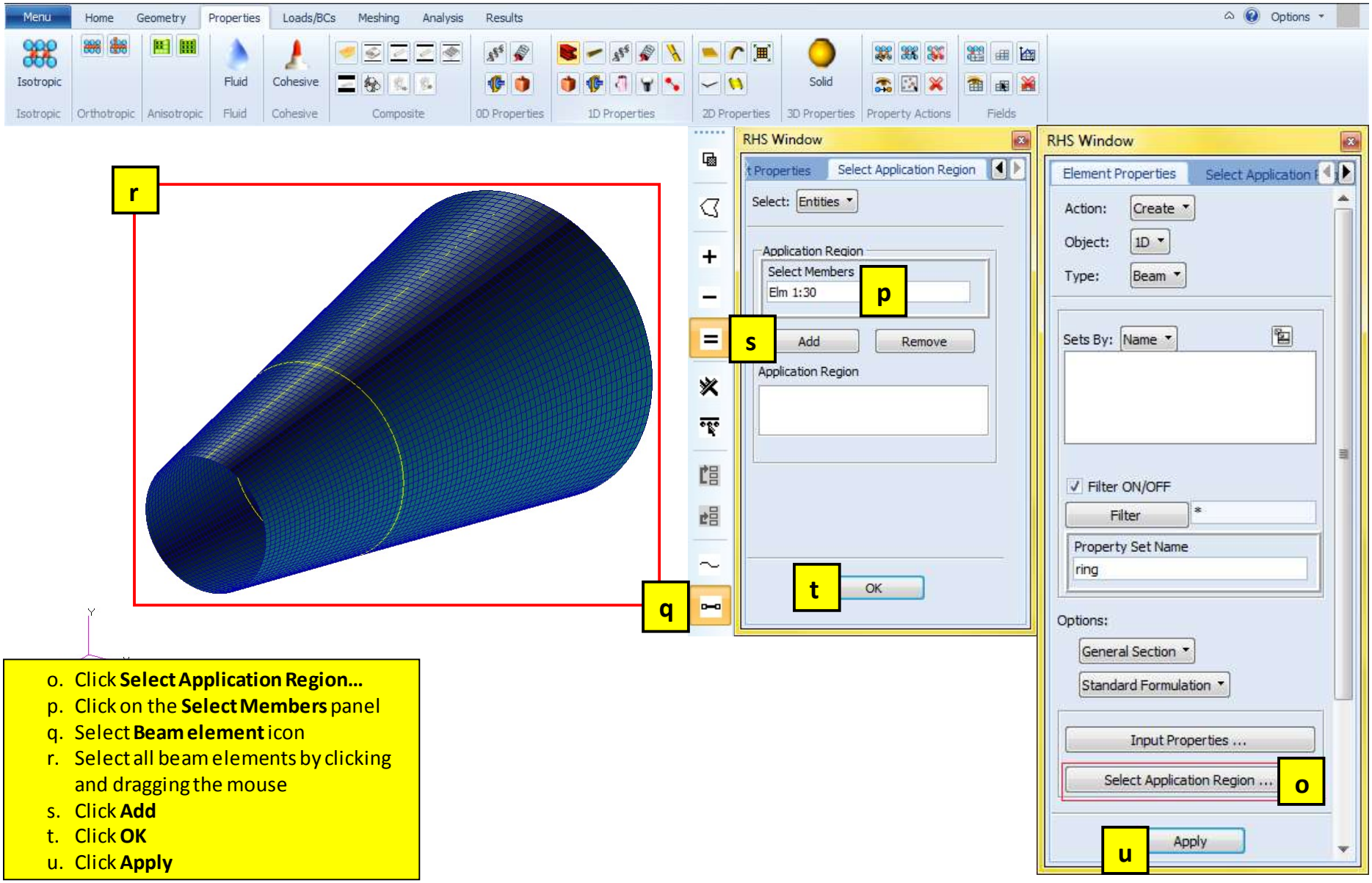


e. Create Sections: **Create/Standard Shape/NASTRAN Standard**
f. Enter **section** as the New Section Name
g. Click 
h. Click 
i. Enter **12** as the value of **W** and **2** as the value of **H**
j. Click **OK**

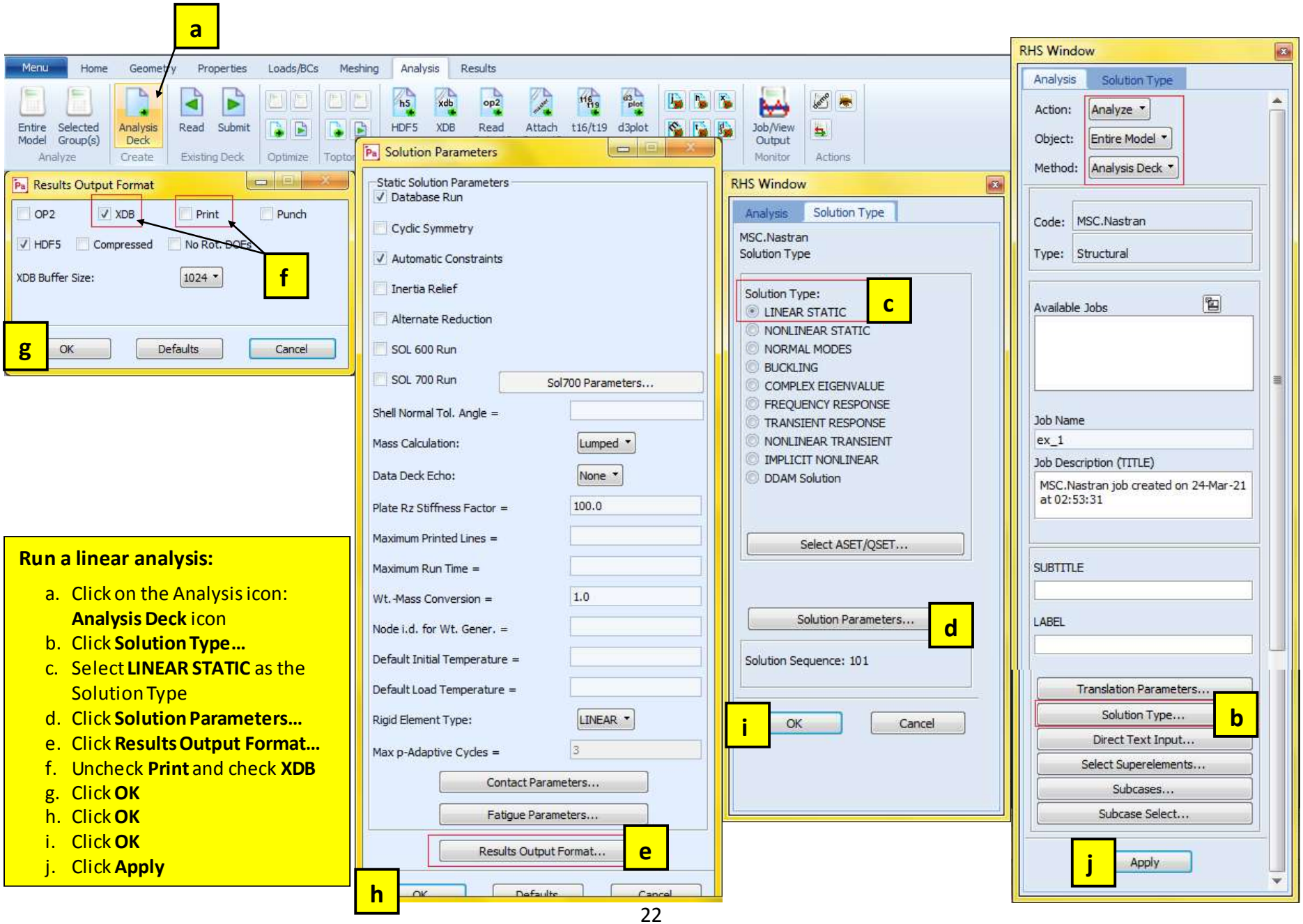




k. Click on the **Mat Prop Name** icon
l. Select **ring_mat**
m. Enter **<0 0 1>** for the **Bar Orientation**
n. Click **OK**



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**

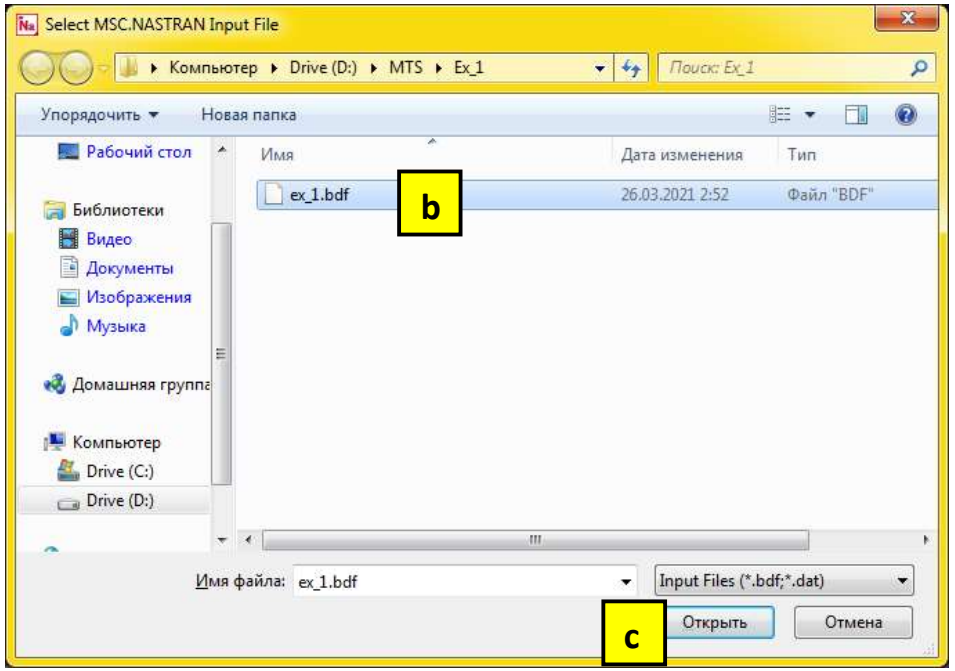


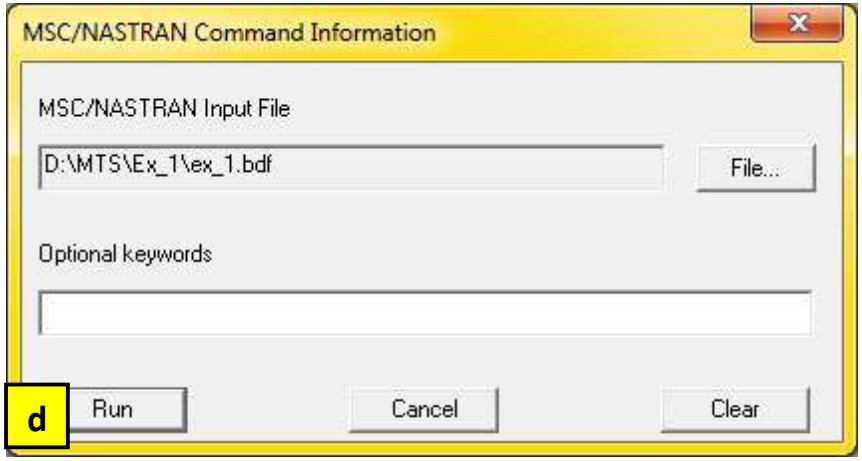
Run a linear analysis:

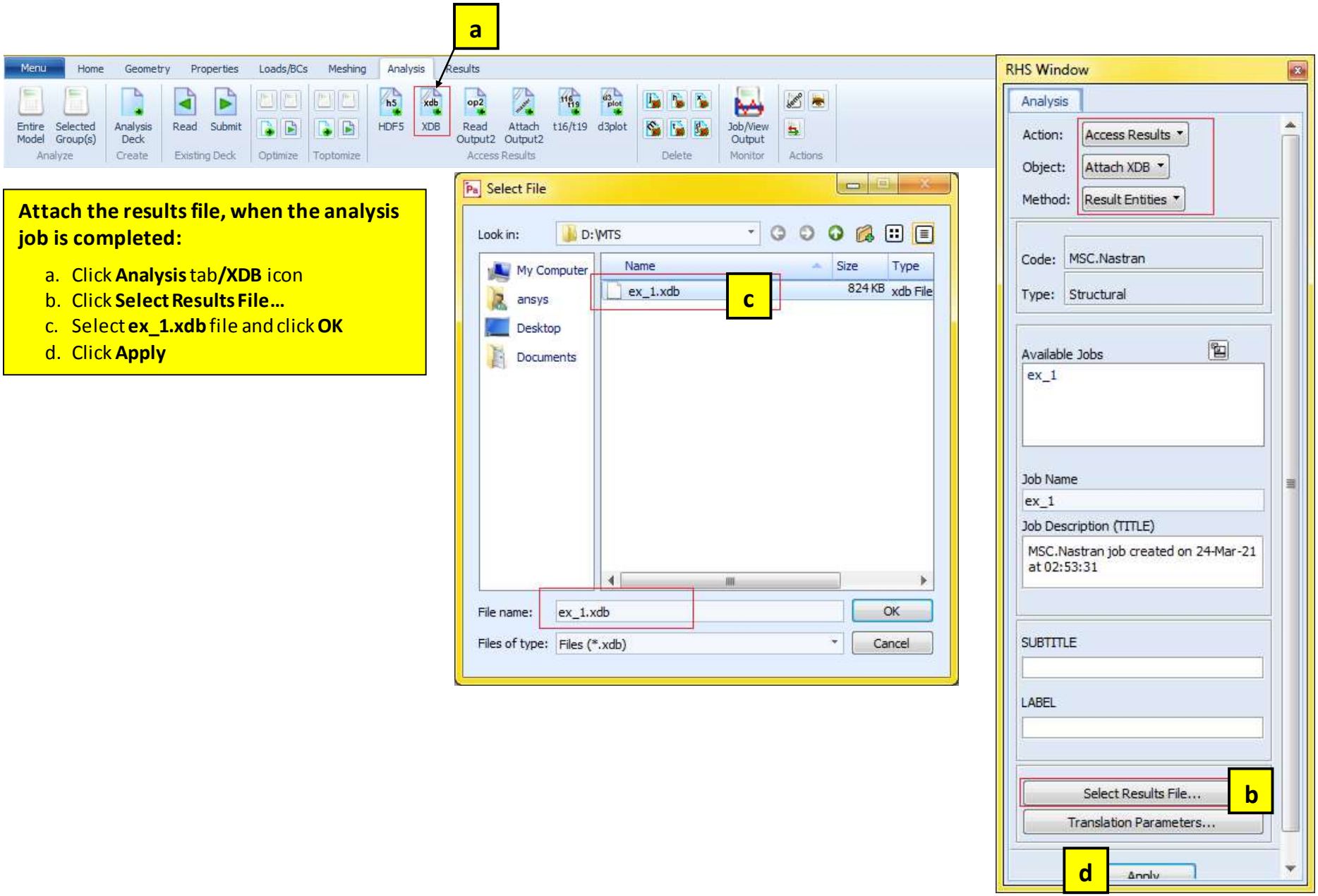
a. Click on the Analysis icon: **Analysis Deck** icon
b. Click **Solution Type…**
c. Select **LINEAR STATIC** as the Solution Type
d. Click **Solution Parameters…**
e. Click **Results Output Format…**
f. Uncheck **Print** and check **XDB**
g. Click **OK**
h. Click **OK**
i. Click **OK**
j. Click **Apply**

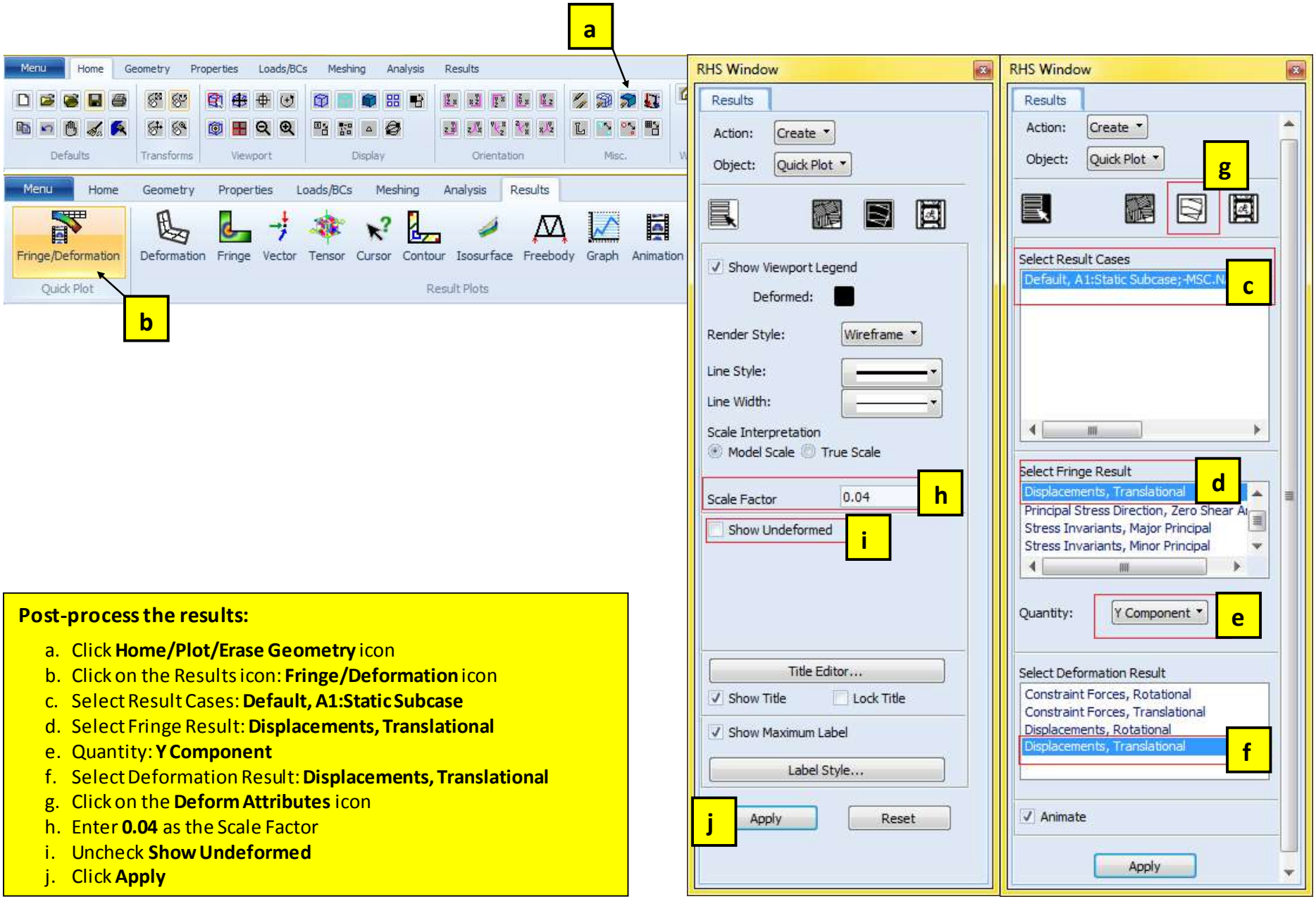
Run **Nastran** analysis:

a. Open the **NASTRAN**
b. Select **ex_1.bdf** file
c. Click **Open**
d. Click **Run**



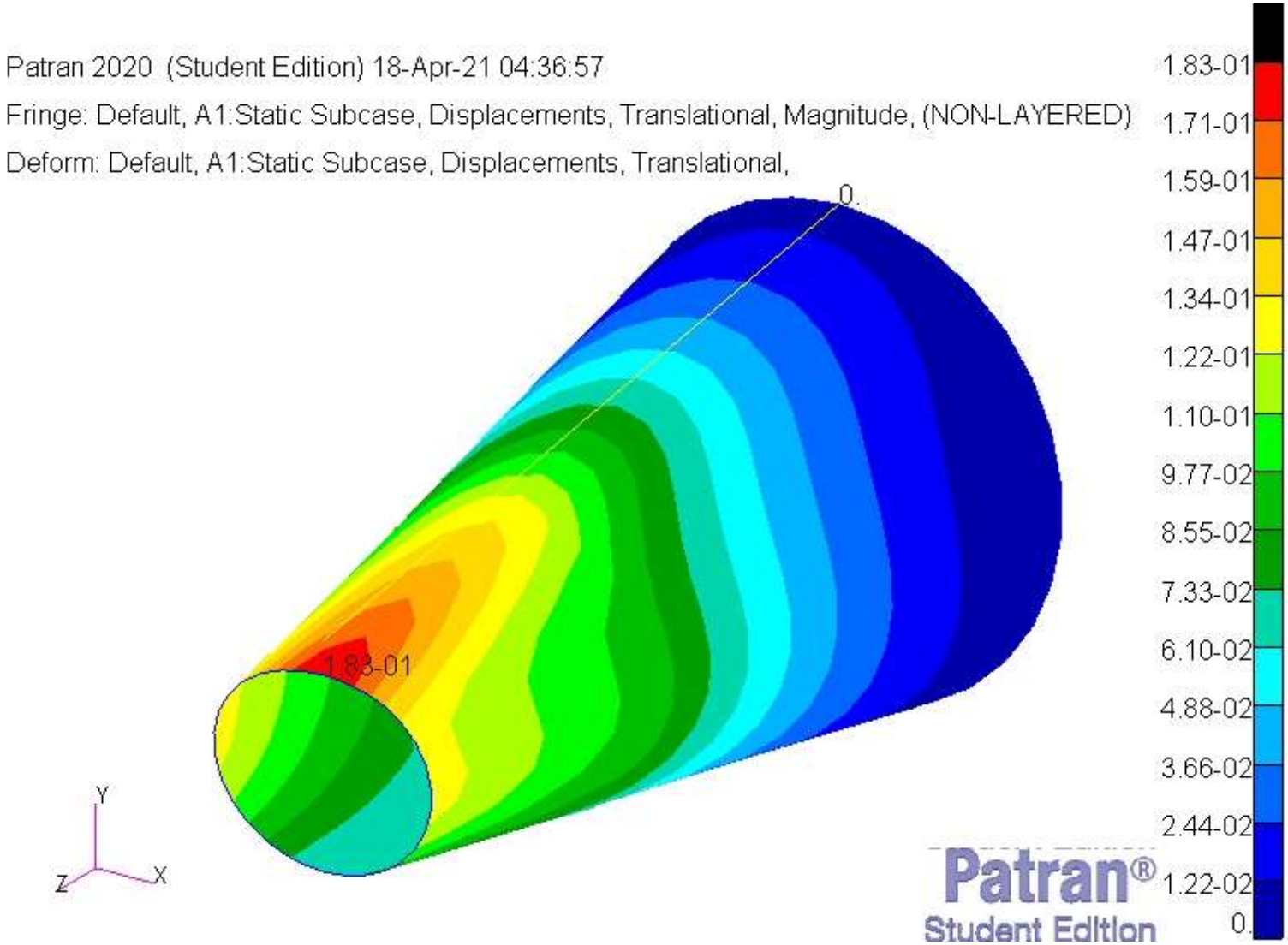


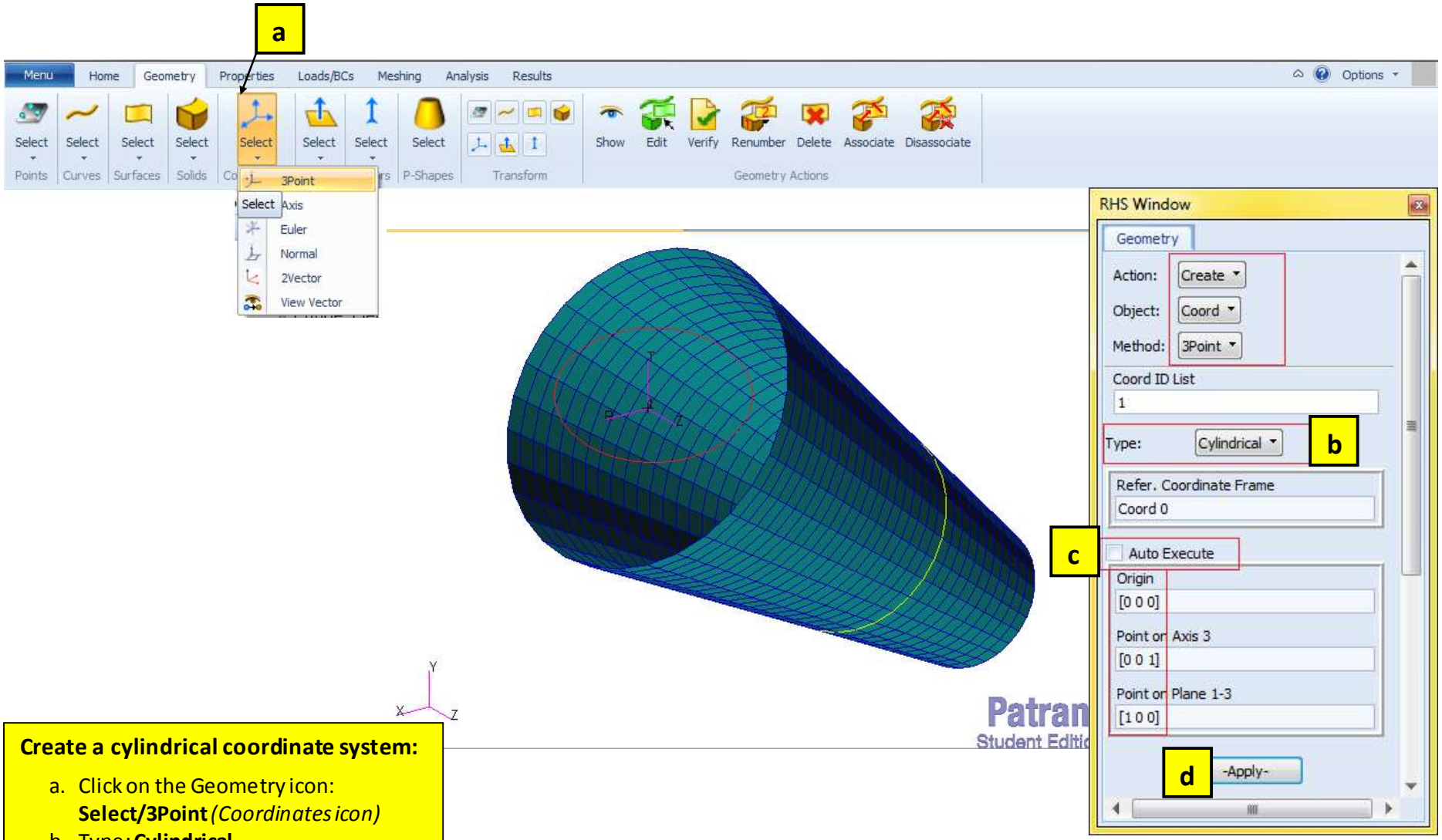

a
Menu
Home
Geometry
Properties
Loads/BCs
Meshing
Analysis
Results
Entire Model
Selected Group(s)
Analysis Deck
Read
Submit
HDF5
XDB
Read Output2
Attach Output2
t16/t19
d3plot
Job/View Output
Analyze
Create
Existing Deck
Optimize
Toptomize
Access Results
Delete
Monitor
Actions
Attach the results file, when the analysis job is completed:
a. Click Analysis tab/XDB icon
b. Click Select Results File…
c. Select ex_1.xdb file and click OK
d. Click Apply
Select File
Look in:
D:\MTS
My Computer
ansys
Desktop
Documents
Name
Size
Type
ex_1.xdb
824 KB
xdb File
c
File name:
ex_1.xdb
OK
Files of type:
Files (*.xdb)
Cancel
RHS Window
Analysis
Action:
Access Results
Object:
Attach XDB
Method:
Result Entities
Code:
MSC.Nastran
Type:
Structural
Available Jobs
ex_1
Job Name
ex_1
Job Description (TITLE)
MSC.Nastran job created on 24-Mar-21 at 02:53:31
SUBTITLE
LABEL
Select Results File...
b
Translation Parameters...
d
Apply



Post-process the results:

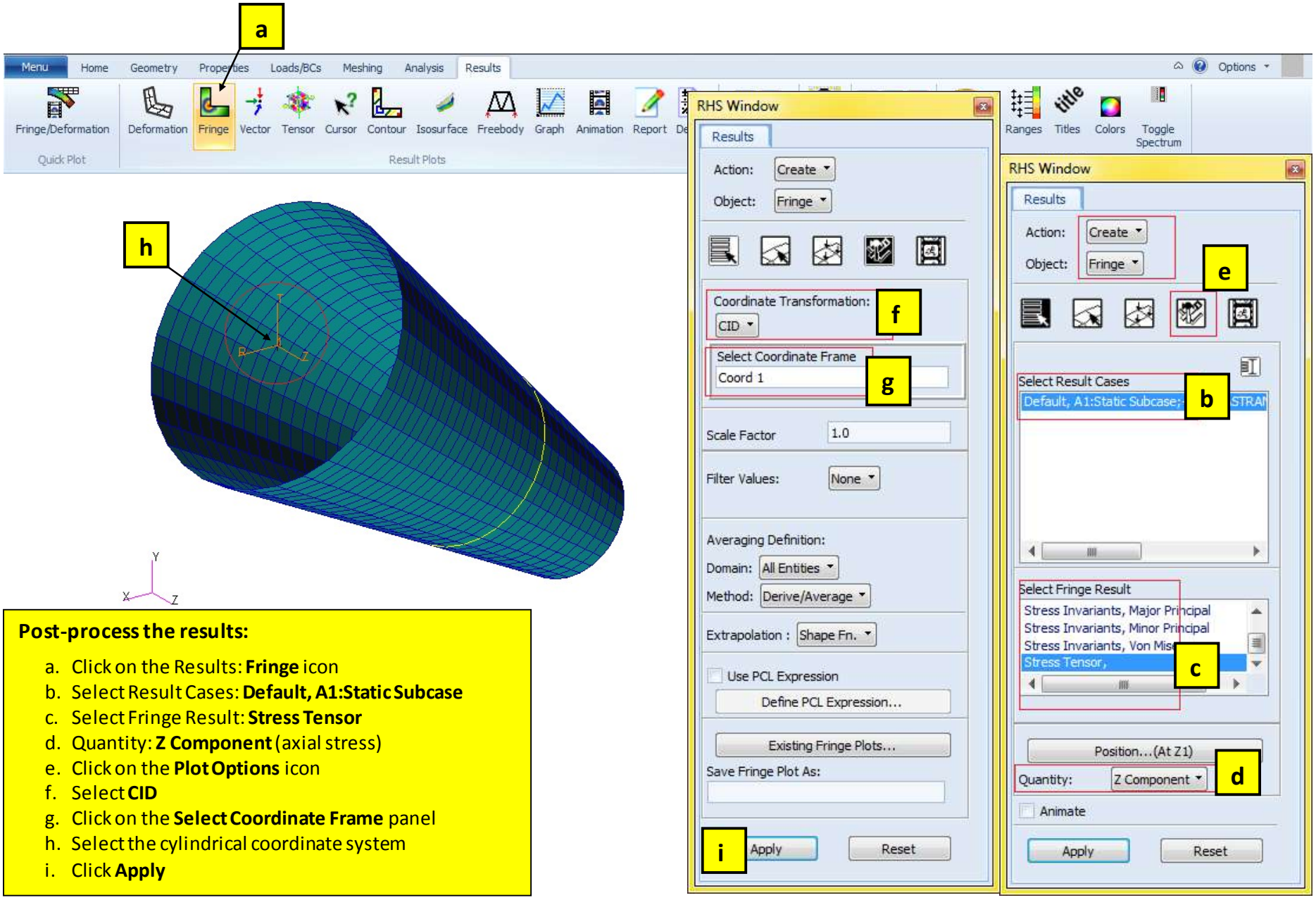
a. Click **Home/Plot/Erase Geometry** icon
b. Click on the Results icon: **Fringe/Deformation** icon
c. Select Result Cases: **Default, A1:Static Subcase**
d. Select Fringe Result: **Displacements, Translational**
e. Quantity: **Y Component**
f. Select Deformation Result: **Displacements, Translational**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


Patran 2020 (Student Edition) 18-Apr-21 04:36:57
Fringe: Default, A1:Static Subcase, Displacements, Translational, Magnitude, (NON-LAYERED)
Deform: Default, A1:Static Subcase, Displacements, Translational,
0.
1.83-01
Y
Z
X
1.83-01
1.71-01
1.59-01
1.47-01
1.34-01
1.22-01
1.10-01
9.77-02
8.55-02
7.33-02
6.10-02
4.88-02
3.66-02
2.44-02
1.22-02
0.
Patran®
Student Edition



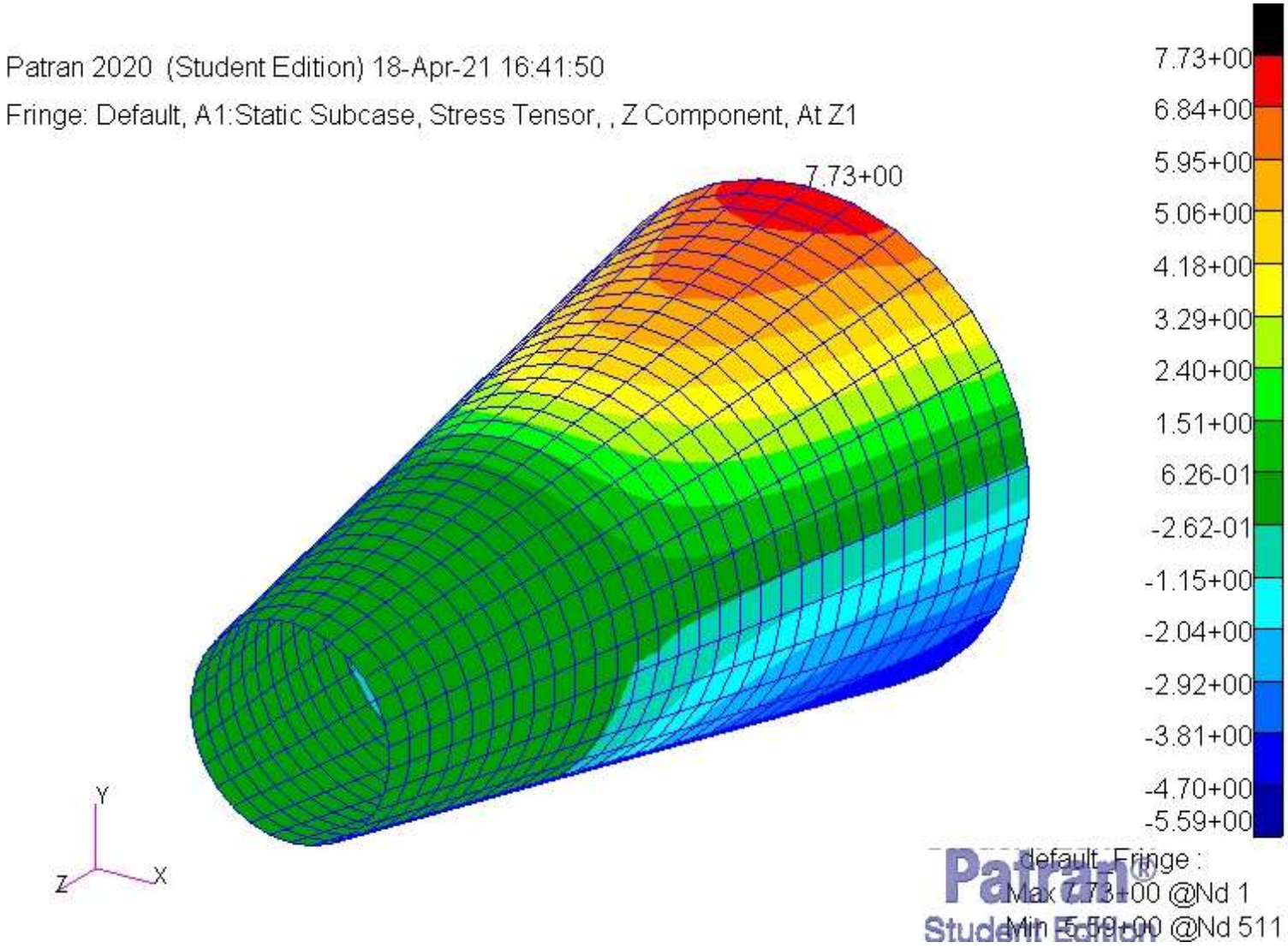
Create a cylindrical coordinate system:

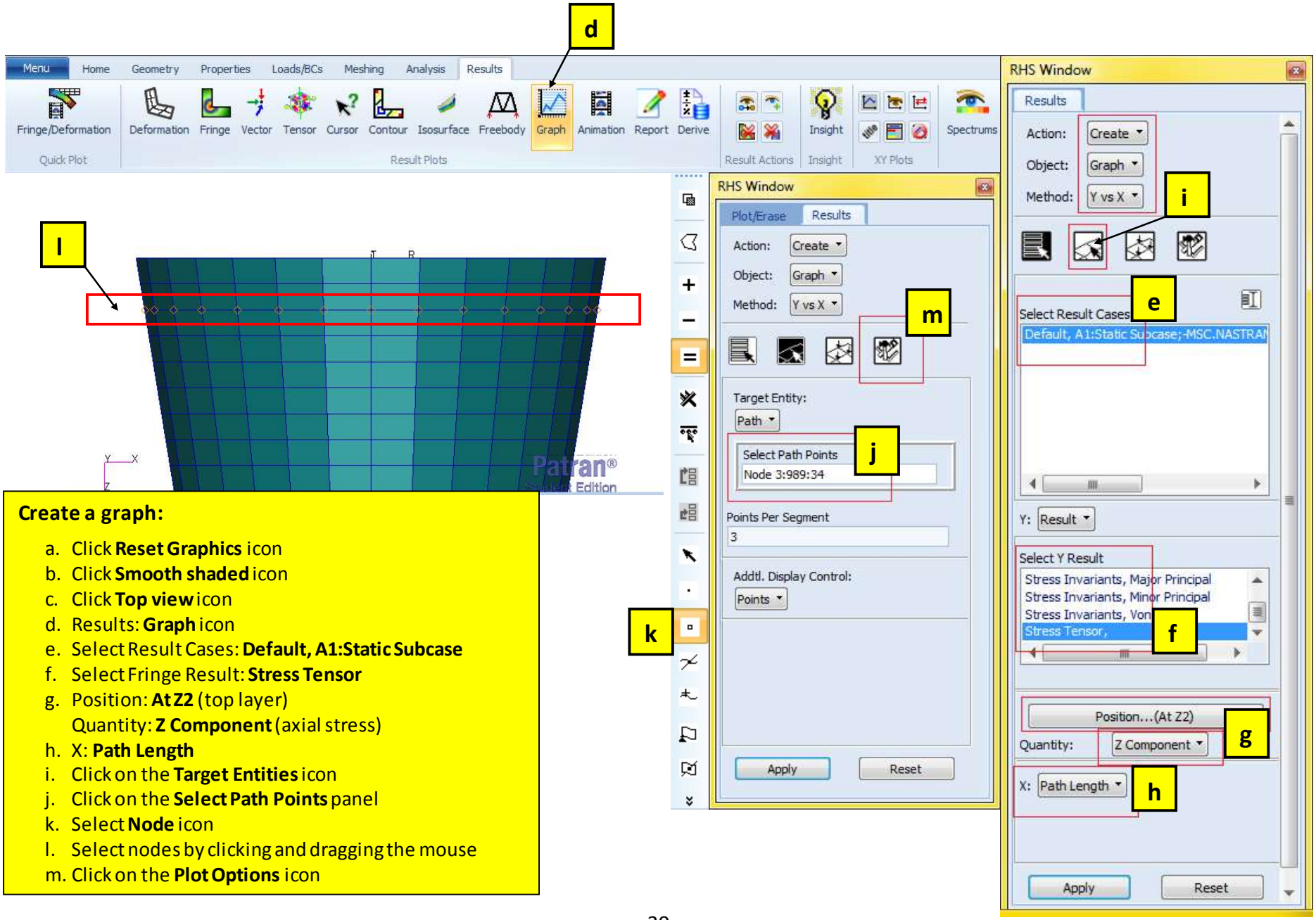
a. Click on the Geometry icon: **Select/3Point** *(Coordinates icon)*
b. Type: **Cylindrical**
c. Uncheck **Auto Execute**
d. Click **Apply**
e. Click **Home/Smooth shaded** icon
f. Click on the **Iso 4 View** icon



Post-process the results:

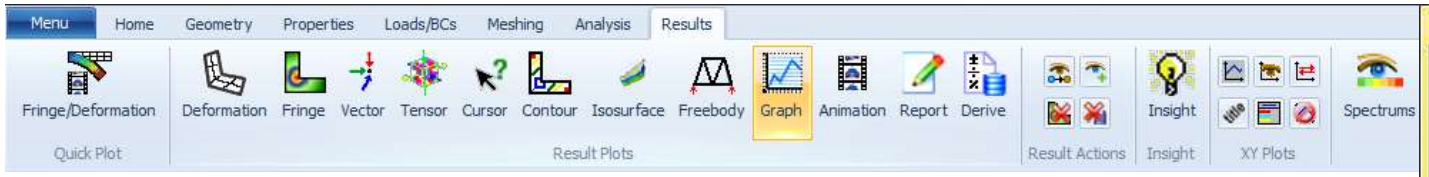
a. Click on the Results: **Fringe** icon
b. Select Result Cases: **Default, A1:Static Subcase**
c. Select Fringe Result: **Stress Tensor**
d. Quantity: **Z Component** (axial stress)
e. Click on the **Plot Options** icon
f. Select **CID**
g. Click on the **Select Coordinate Frame** panel
h. Select the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


Patran 2020 (Student Edition) 18-Apr-21 16:41:50
Fringe: Default, A1:Static Subcase, Stress Tensor, , Z Component, At Z1
7.73+00
7.73+00
6.84+00
5.95+00
5.06+00
4.18+00
3.29+00
2.40+00
1.51+00
6.26-01
-2.62-01
-1.15+00
-2.04+00
-2.92+00
-3.81+00
-4.70+00
-5.59+00
Y
Z
X
default Fringe :
Max 7.73+00 @Nd 1
Min -5.59+00 @Nd 511
Patran®
Student Edition

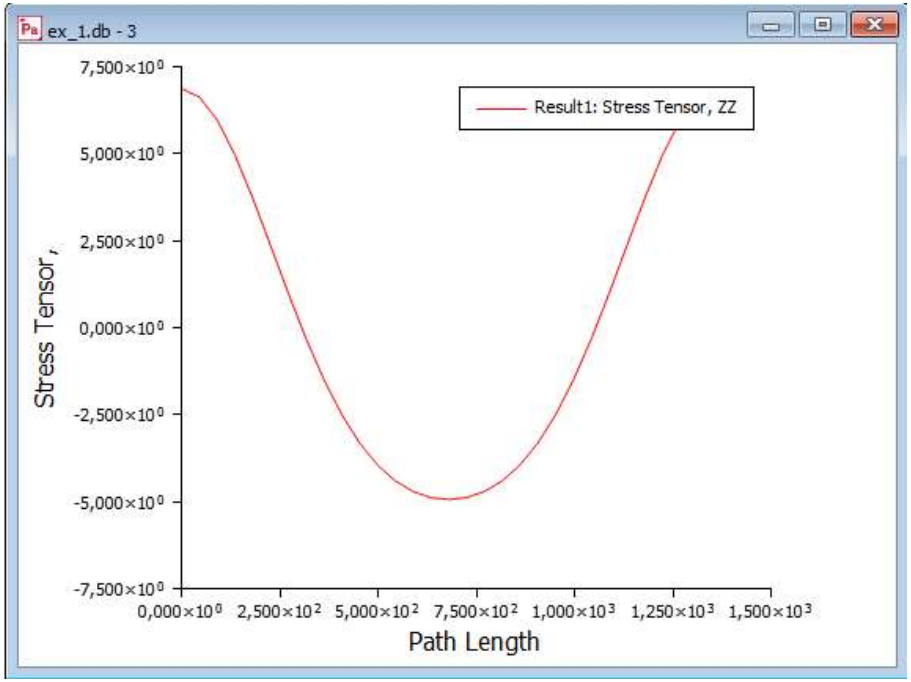


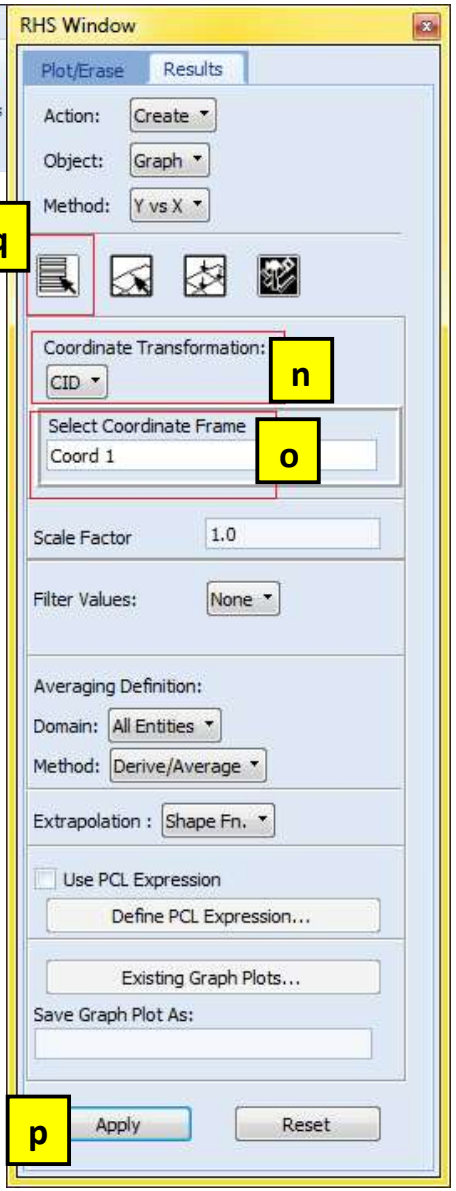
Create a graph:

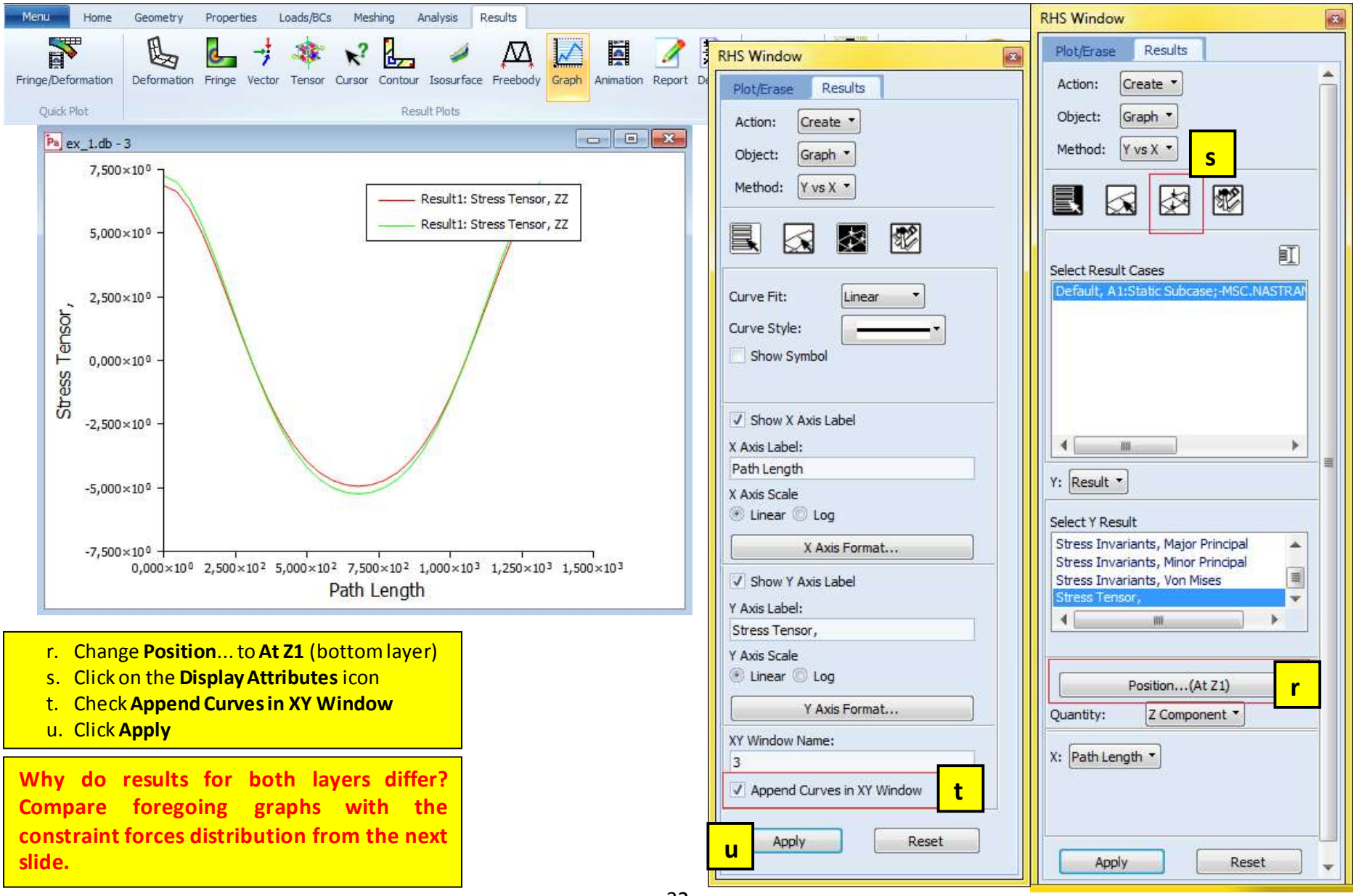
a. Click **Reset Graphics** icon
b. Click **Smooth shaded** icon
c. Click **Top view** icon
d. Results: **Graph** icon
e. Select Result Cases: **Default, A1:Static Subcase**
f. Select Fringe Result: **Stress Tensor**
g. Position: **At Z2** (top layer)
   Quantity: **Z Component** (axial stress)
h. X: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



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

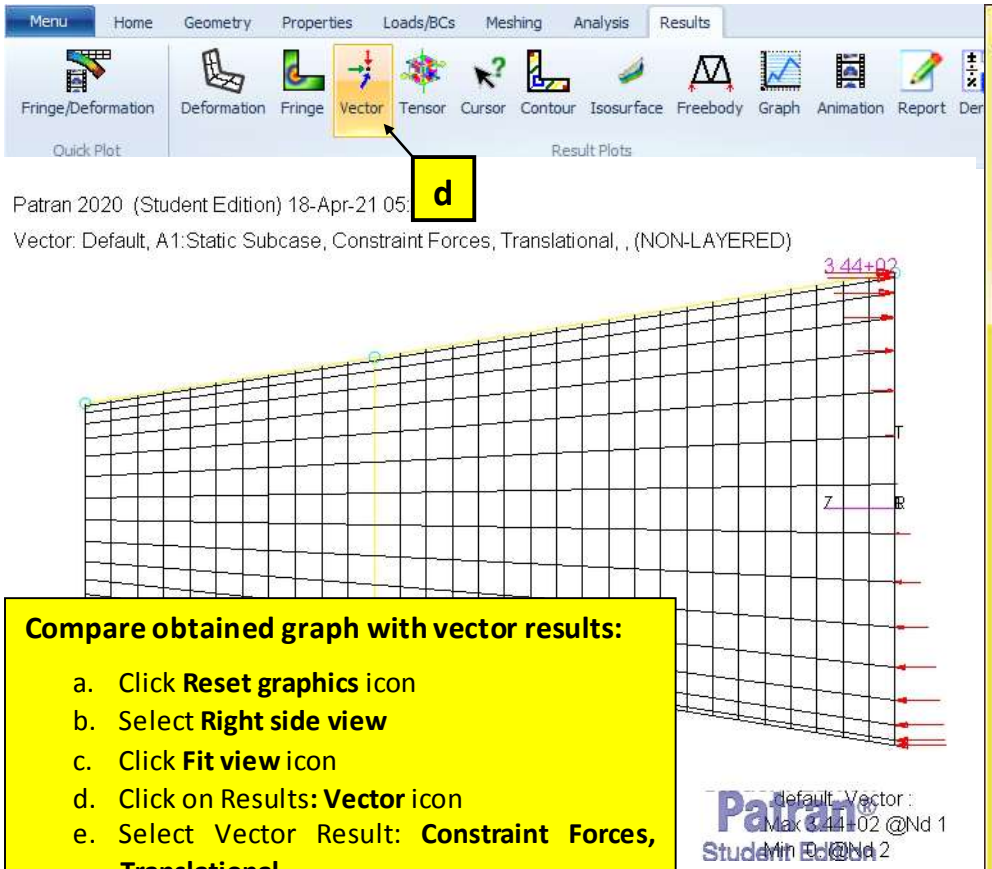






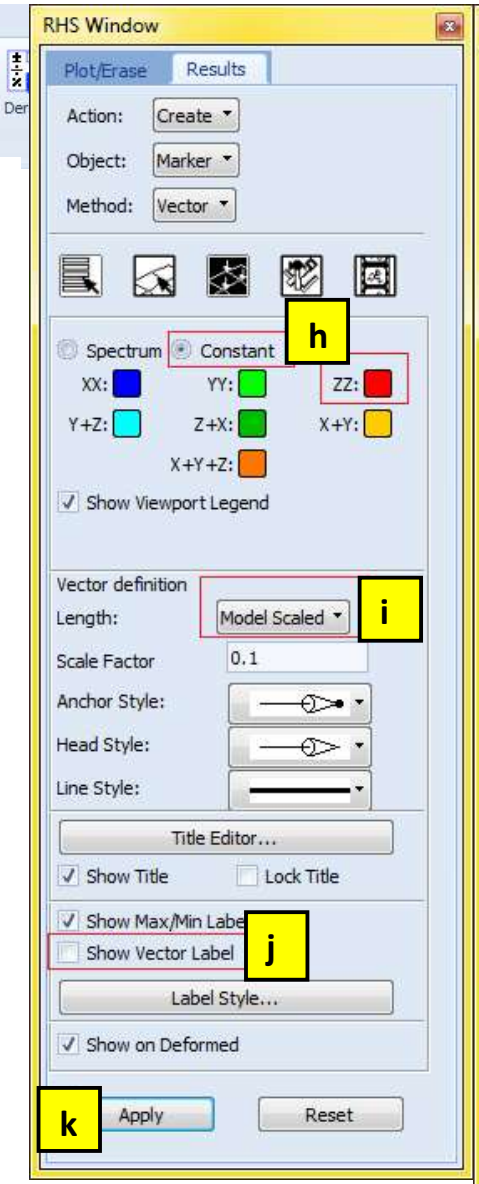
r. Change **Position**... to **At Z1** (bottom layer)
s. Click on the **Display Attributes** icon
t. Check **Append Curves in XY Window**
u. Click **Apply**

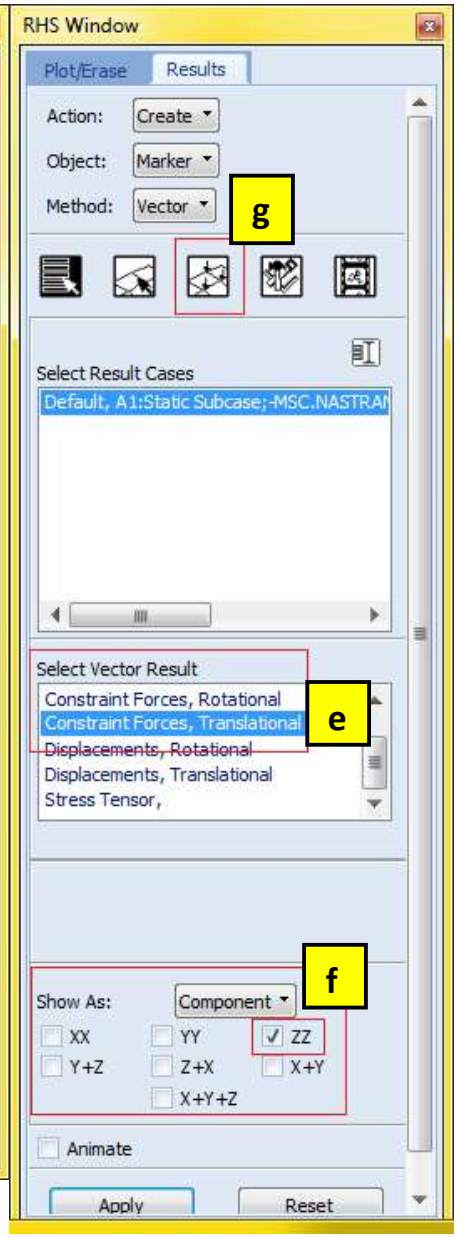
Why do results for both layers differ? Compare foregoing graphs with the constraint forces distribution from the next slide.

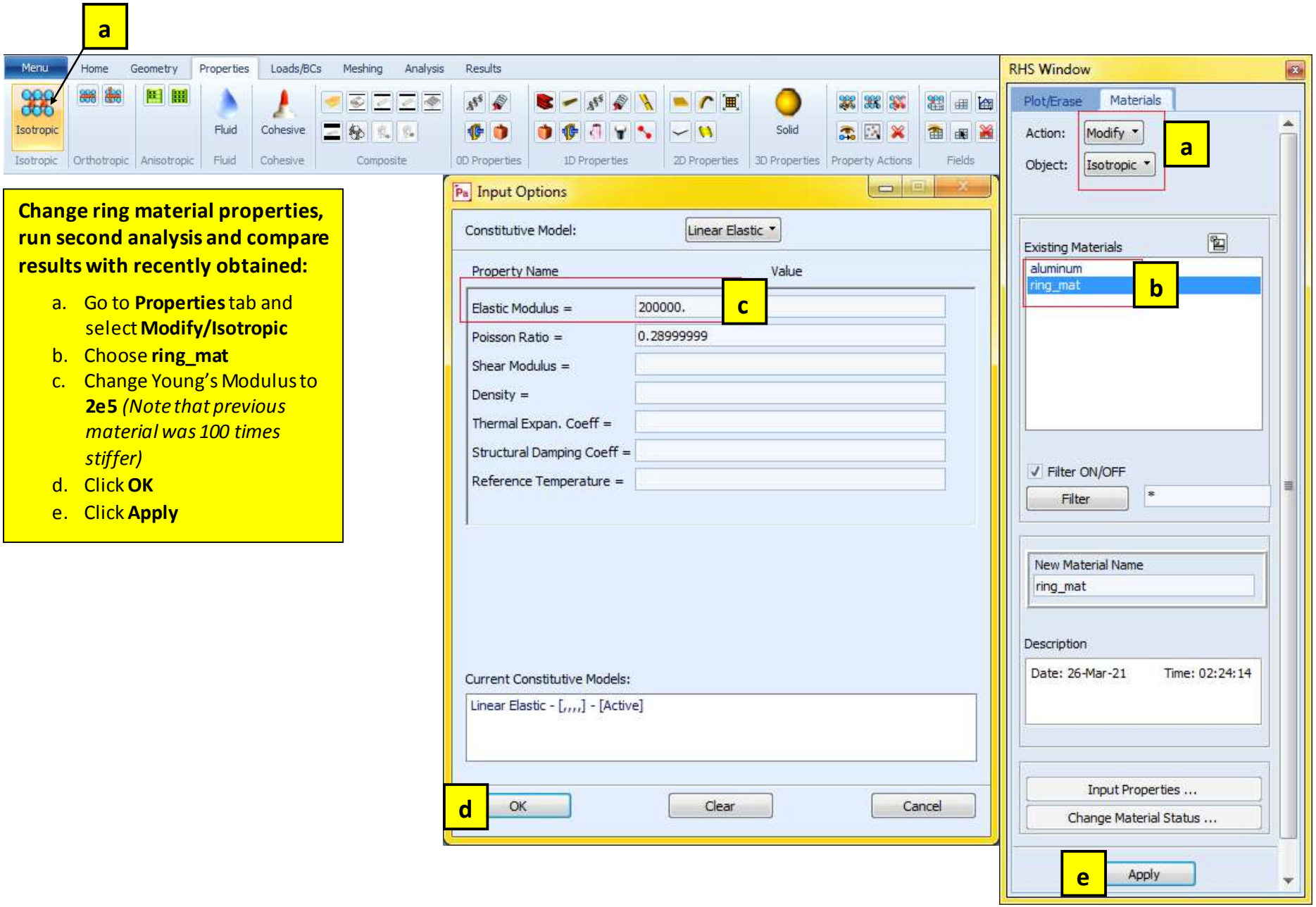


Compare obtained graph with vector results:

a. Click **Reset graphics** icon
b. Select **Right side view**
c. Click **Fit view** icon
d. Click on Results**: Vector** icon
e. Select Vector Result: **Constraint Forces, Translational**
f. Change option to **Component** and check only **ZZ** option
g. Click on **Display Attributes** icon
h. Check **Constant** vector color option and choose red color for **ZZ:** component
i. Change **Length** to **Model Scaled**
j. Uncheck **Show Vector Label**
k. Click **Apply**

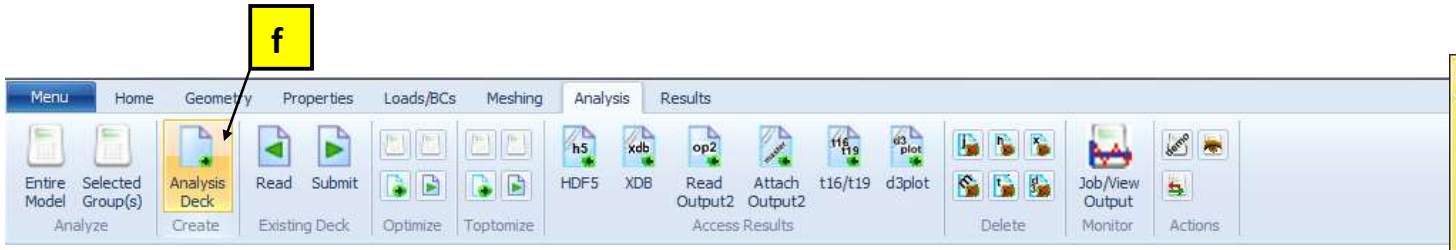




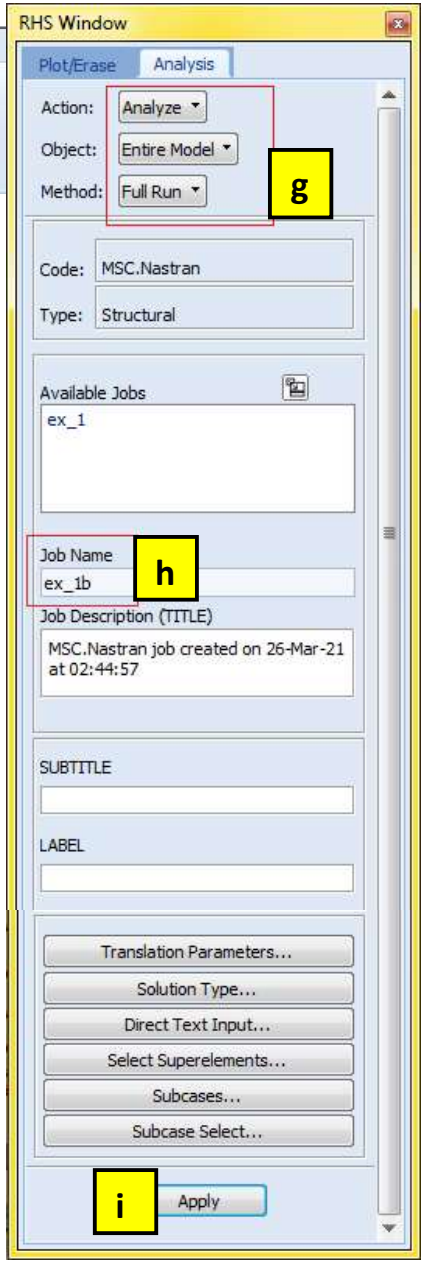


Change ring material properties, run second analysis and compare results with recently obtained:

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**

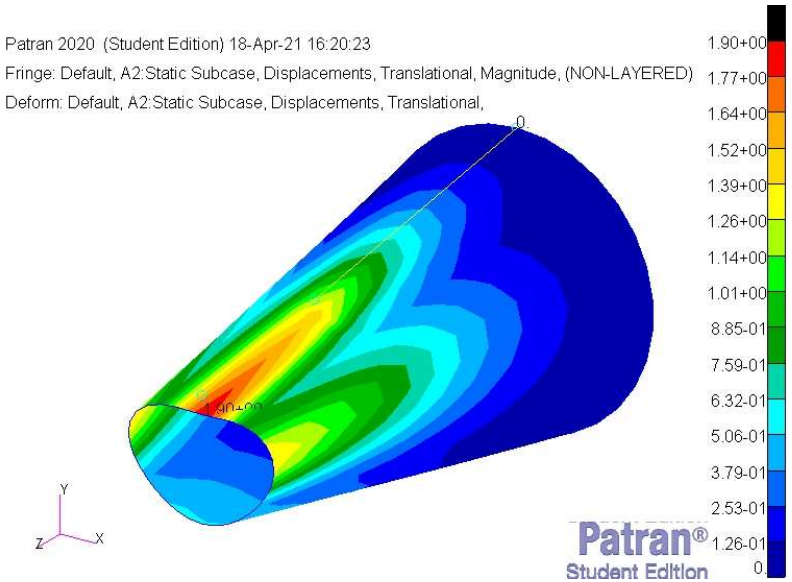


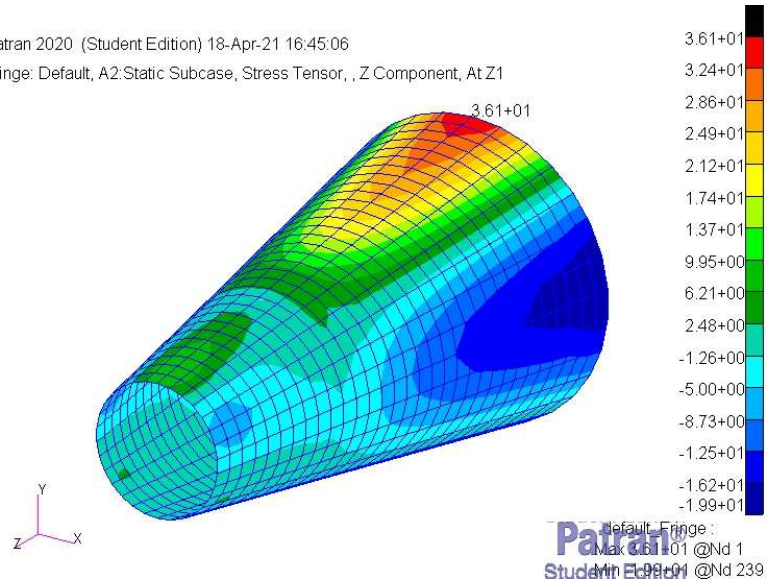
f. Click Analysis tab: **Analysis Deck** icon
g. Method: **Full Run**
h. Change **Job Name** to **ex_1b**
i. Select **Apply**
j. After analysis is done, ***attach*** **ex_1b.xdb** file and **Plot results** in the same manner as previously. Results referring to the second analysis will be tagged as **A2: Static subcase**

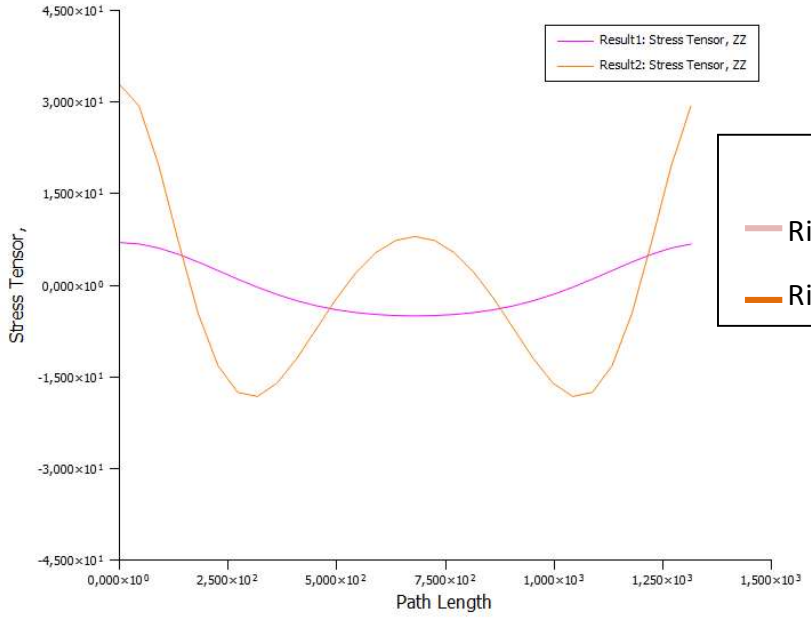


Use following reference to verify the results of the second analysis

- What qualitative changes can be noticed?
- Which calculation yield results more similar to the theoretical solution? Why?







Legend:

Ring stiffness***: E=2e5***, at **Z2**

Ring stiffness***: E=2e7***, at **Z2**