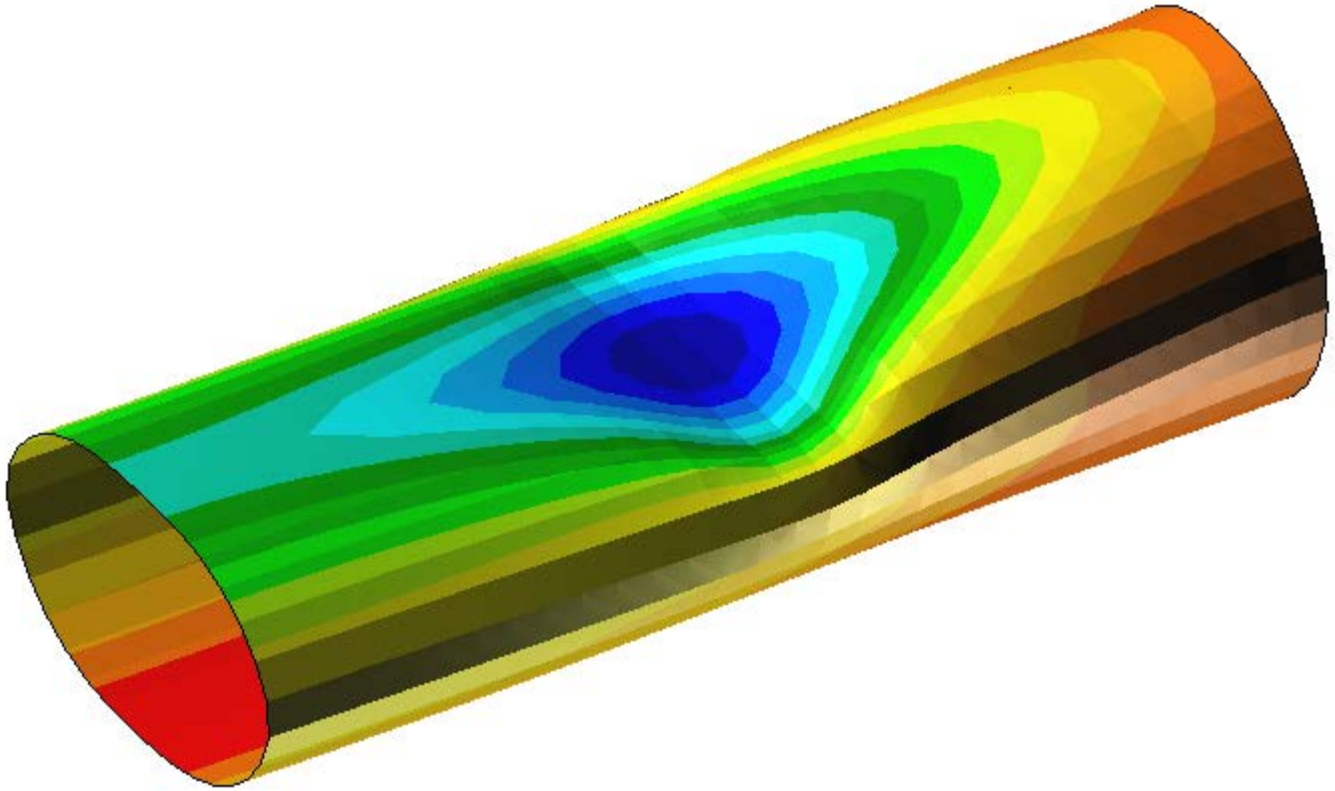
NONLINEAR MECHANICS OF STRUCTURES

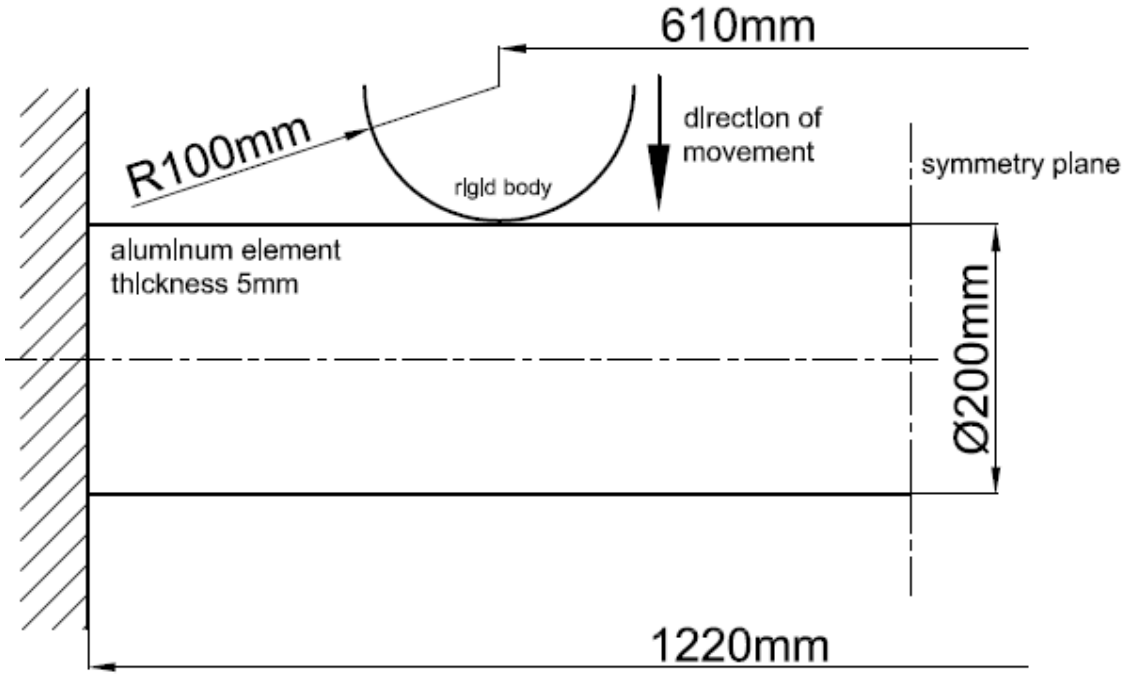
EXERCISE 2



HUMAN CAPITAL
NATIONAL COHESION STRATEGY

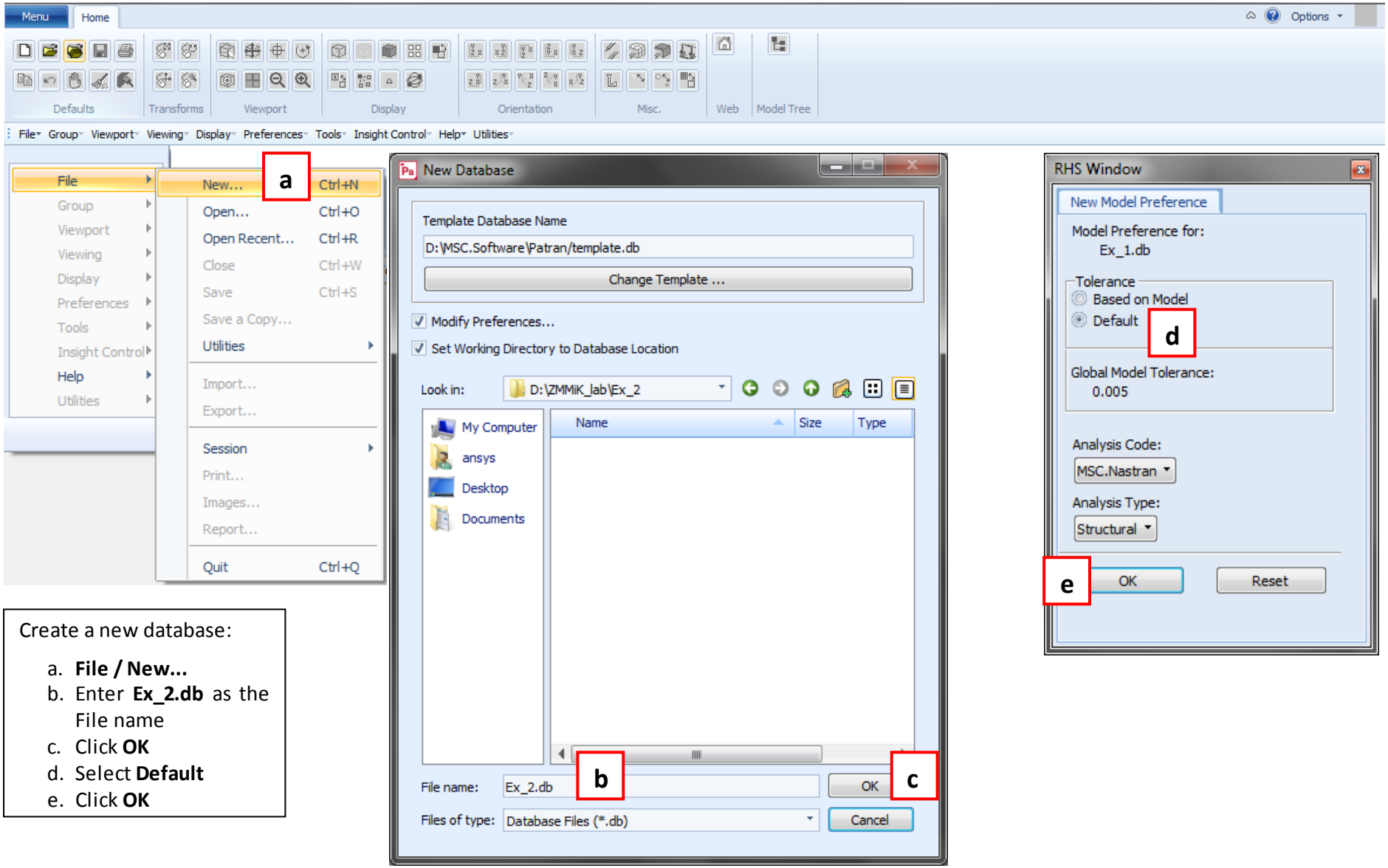
EUROPEAN UNION
EUROPEAN
SOCIAL FUND

PROBLEM DESCRIPTION



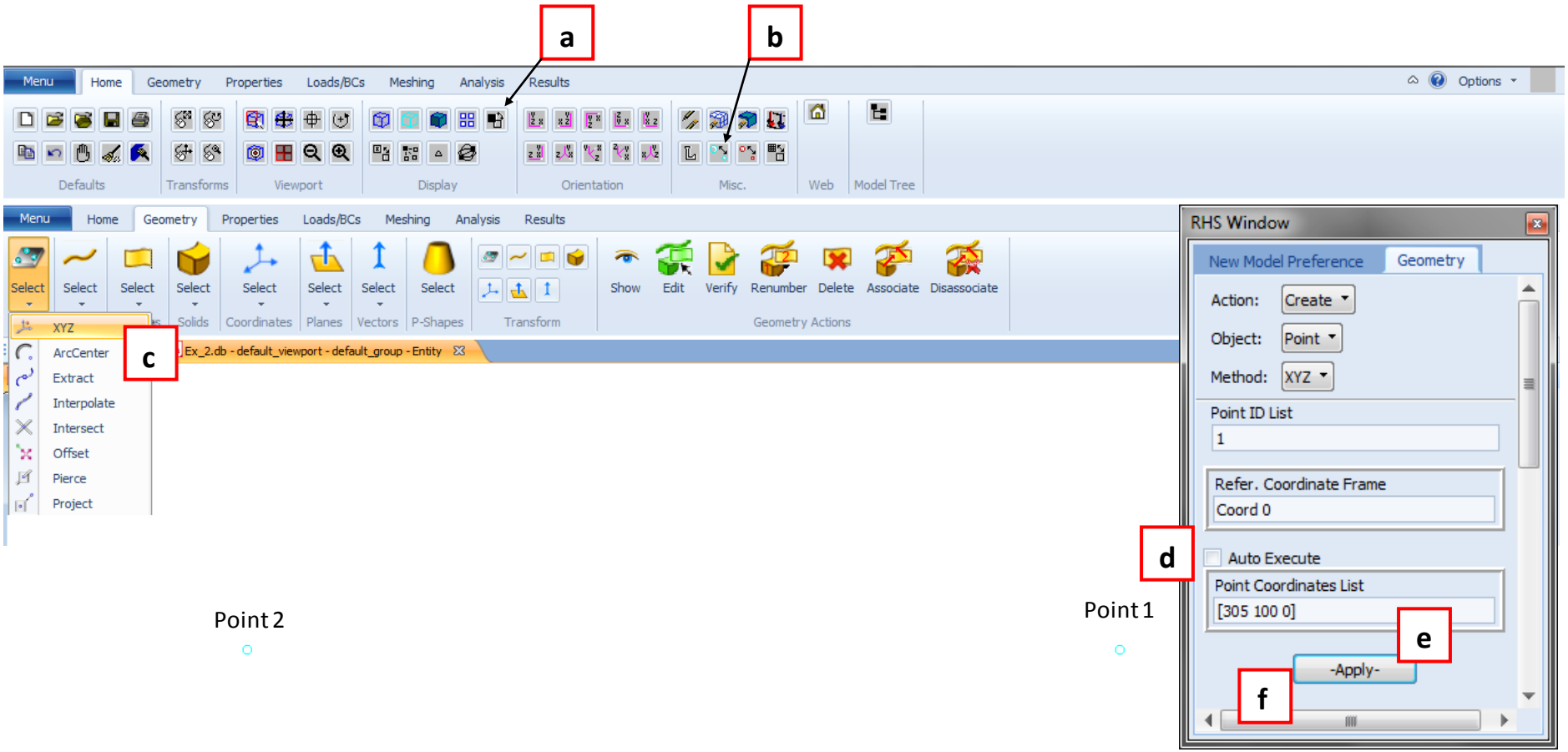
A thin-walled element (a pipe) is crushed by two cylindrical rigid bodies. Due to the symmetry of the task only half of the pipe (with appropriate boundary conditions) is modeled. The goal of this exercise is to perform a nonlinear analysis, which includes high plasticity of the used material, and assess the obtained results.

Units: mm, N, MPa



Create a new database:

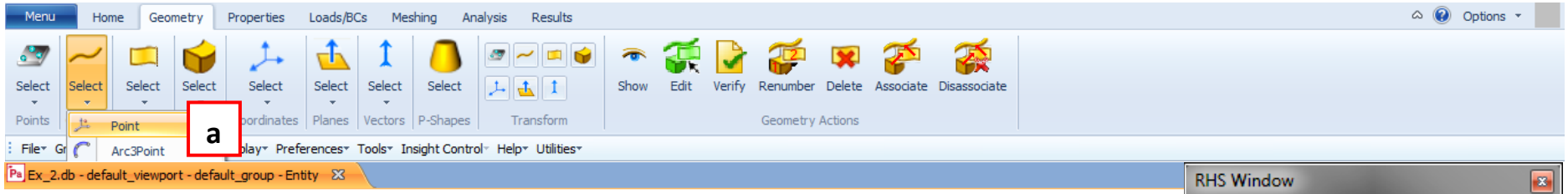
a. **File / New...**
b. Enter **Ex_2.db** as the File name
c. Click **OK**
d. Select **Default**
e. Click **OK**

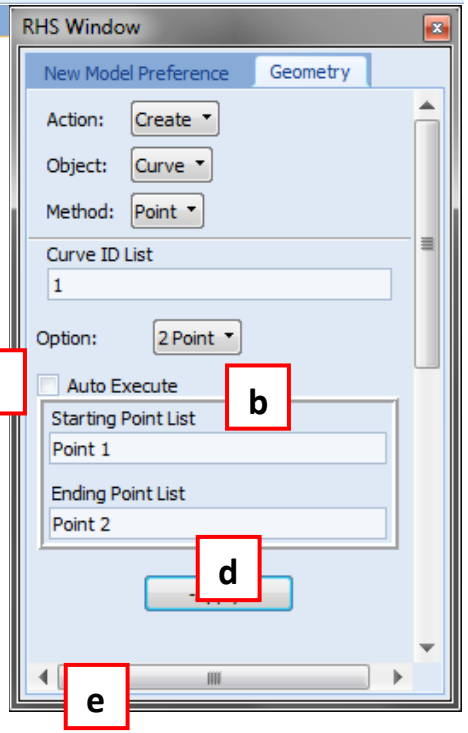


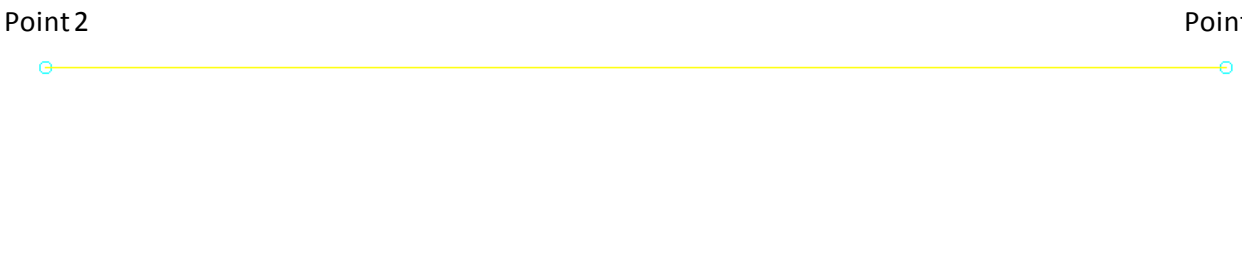
a. Change *Background Color* to **Black** (click on the **Cycle Background** icon)
b. Click on the **Point size** icon

Create geometry points:

c. Click on the **Geometry** icon (Points tab): **Select/XYZ**
d. Uncheck **Auto Execute**
e. Enter **[305 100 0]** as the Point Coordinates List
f. Click **Apply**
g. Create one more point using coordinates: **[-305 100 0]**





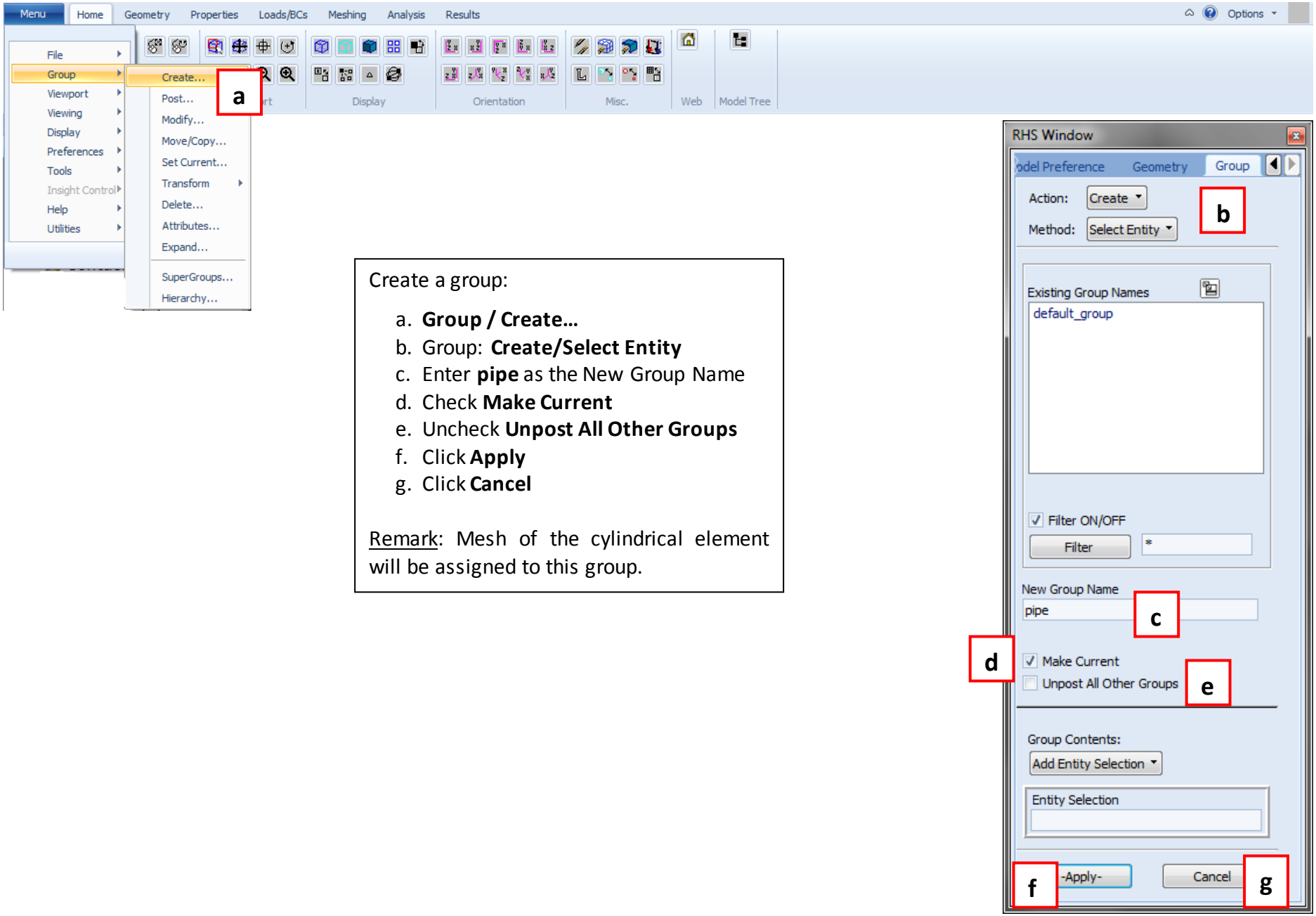


Create a curve:

a. Click on the **Geometry** icon (Curves tab): **Select/Point**
b. Option: **2 Point**
c. Uncheck **Auto Execute**
d. Select **Point 1** as the starting point and **Point 2** as the ending point
e. Click **Apply**

Patran®
Student Edition

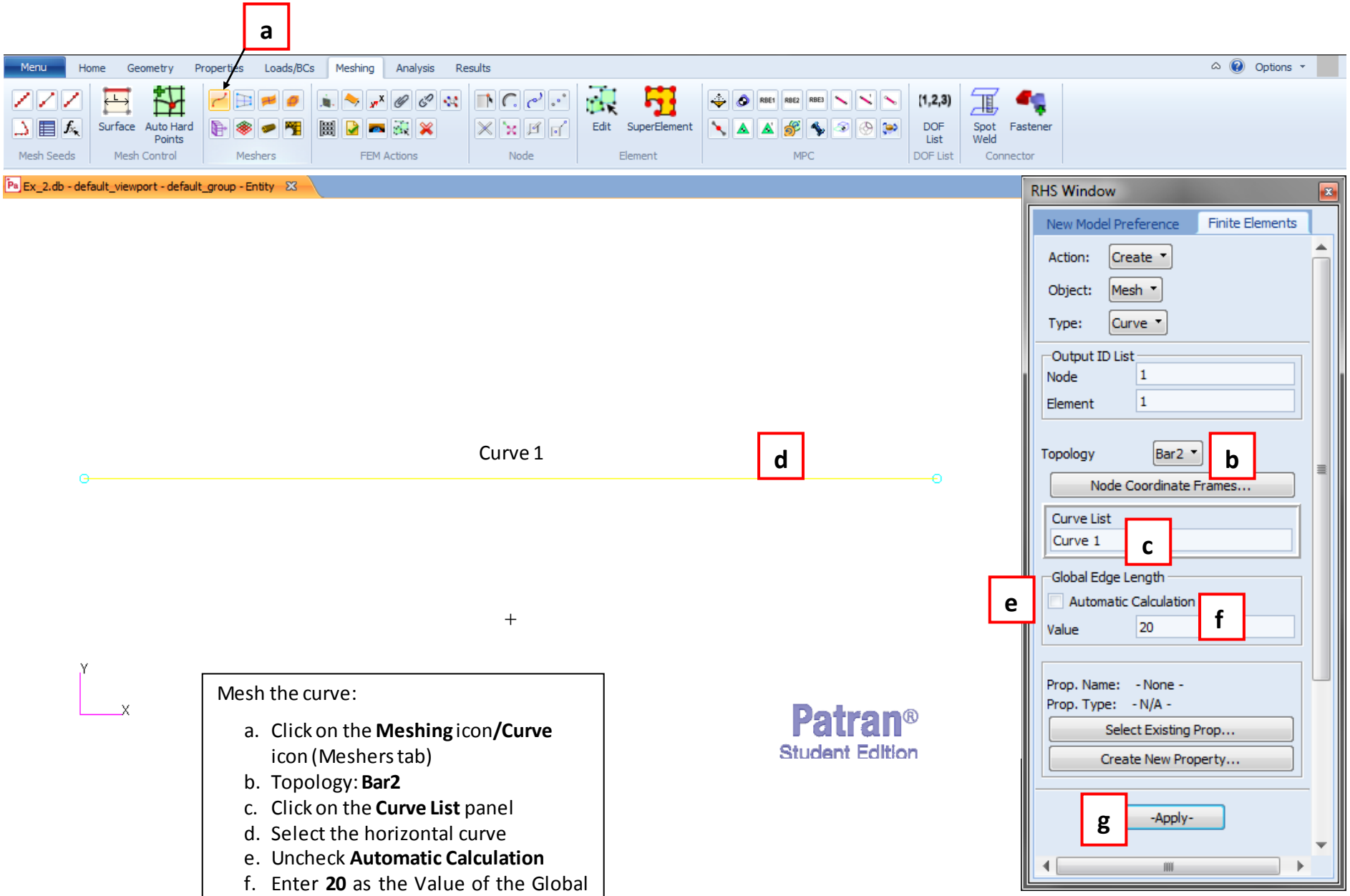
HUMAN CAPITAL
NATIONAL COHESION STRATEGY
EUROPEAN UNION
EUROPEAN
SOCIAL FUND



Create a group:

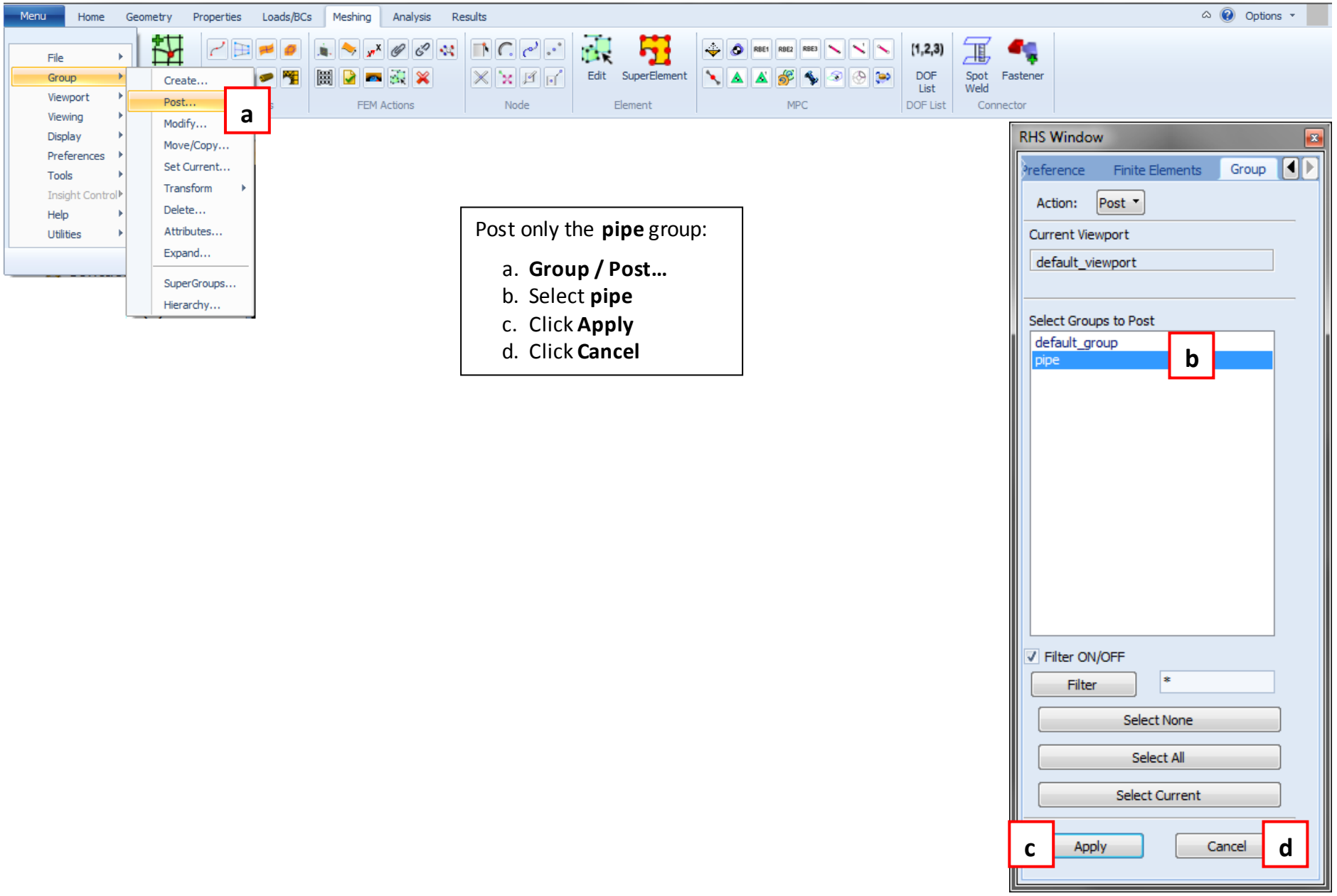
a. **Group / Create…**
b. Group: **Create/Select Entity**
c. Enter **pipe** as the New Group Name
d. Check **Make Current**
e. Uncheck **Unpost All Other Groups**
f. Click **Apply**
g. Click **Cancel**

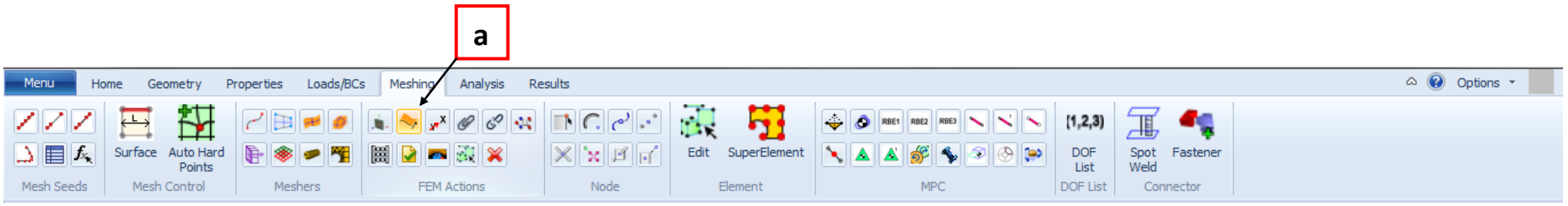
Remark: Mesh of the cylindrical element will be assigned to this group.



Mesh the curve:

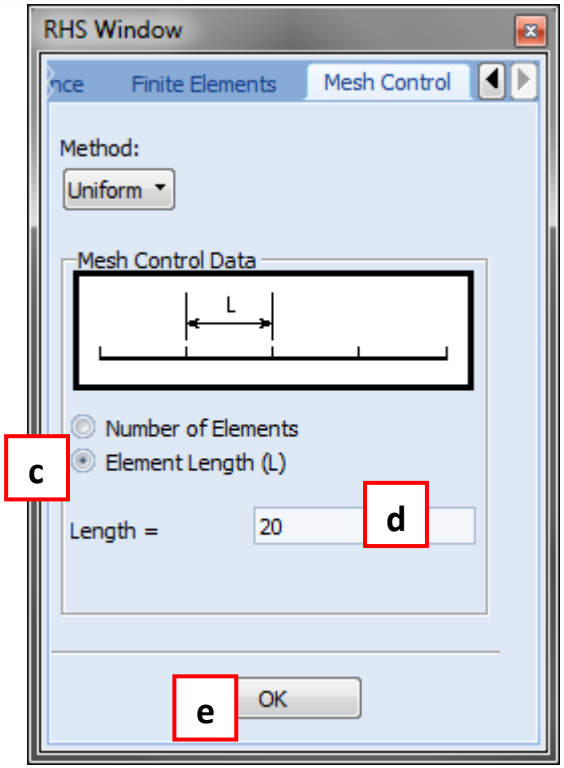
a. Click on the **Meshing** icon/**Curve** icon (Meshers tab)
b. Topology: **Bar2**
c. Click on the **Curve List** panel
d. Select the horizontal curve
e. Uncheck **Automatic Calculation**
f. Enter **20** as the Value of the Global Edge Length
g. Click **Apply**

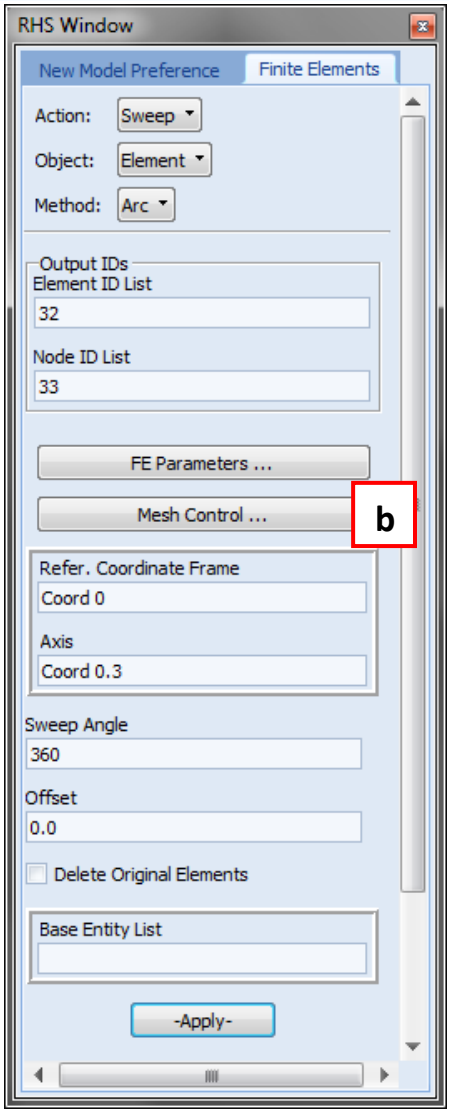


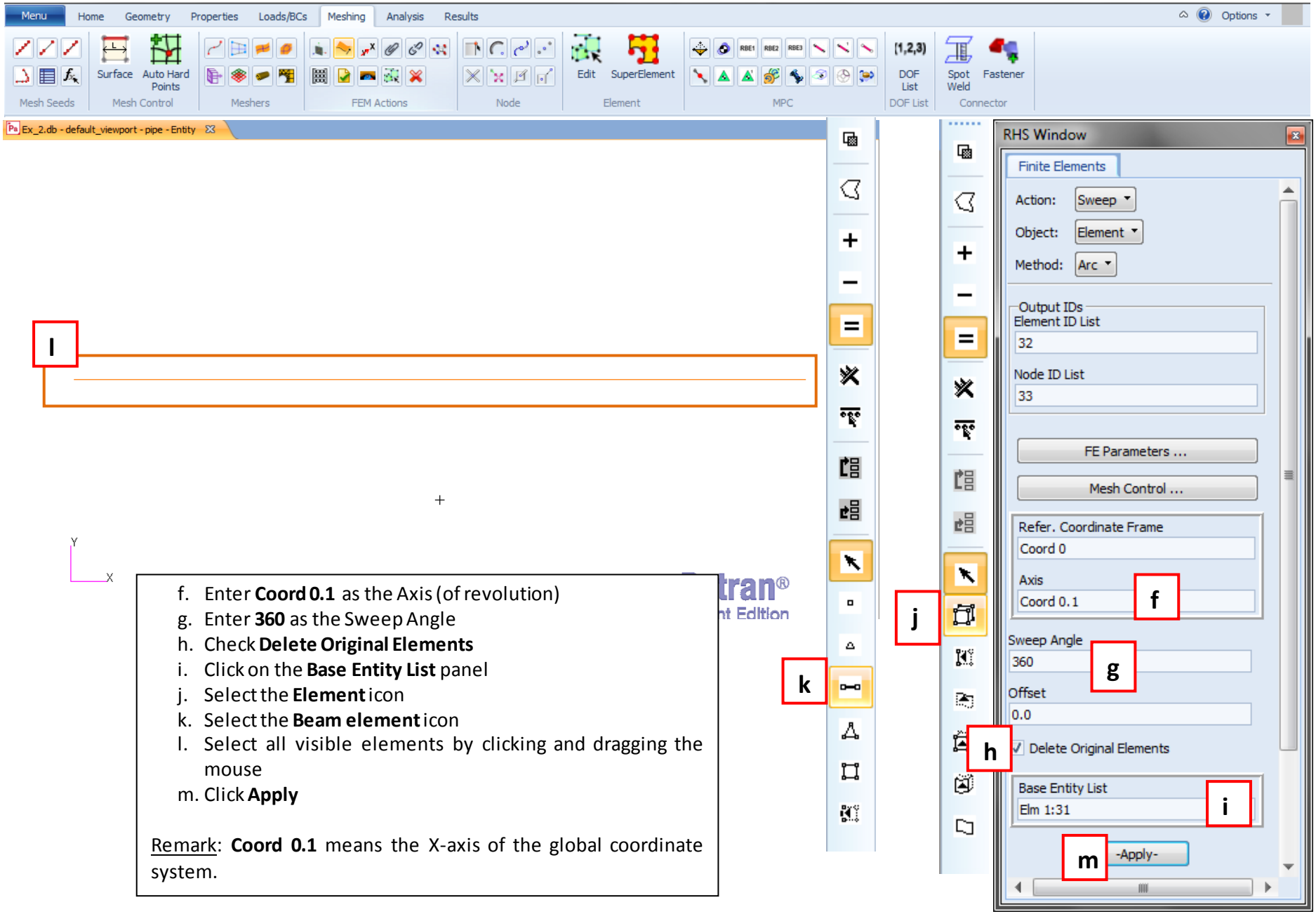


Sweep the elements:

a. Meshing: **Sweep** icon (FEM Actions tab)
b. Click **Mesh Control…**
c. Select **Element Length (L)**
d. Enter **20** as the Length
e. Click **OK**

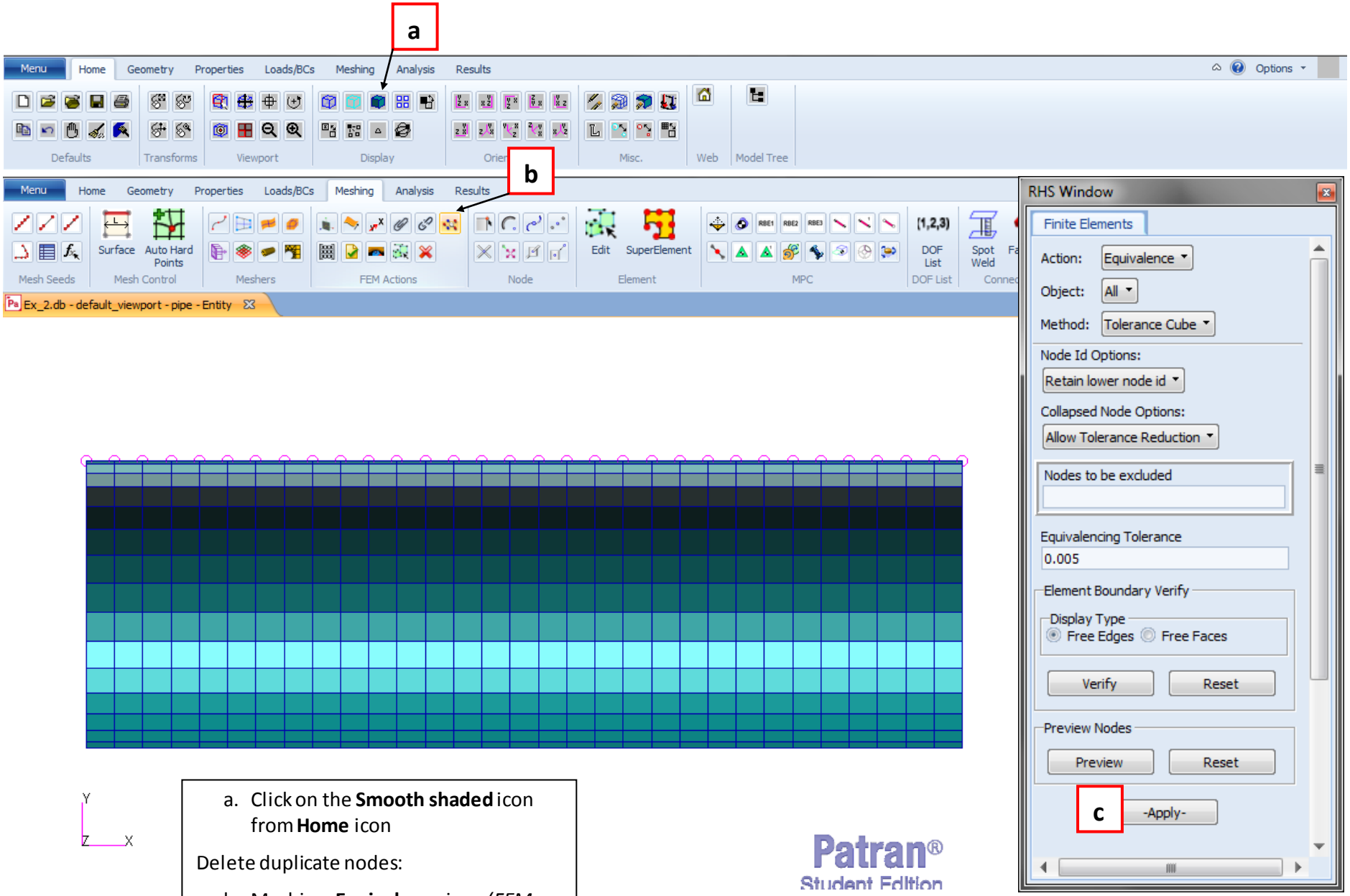






f. Enter **Coord 0.1** as the Axis (of revolution)
g. Enter **360** as the Sweep Angle
h. Check **Delete Original Elements**
i. Click on the **Base Entity List** panel
j. Select the **Element** icon
k. Select the **Beam element** icon
l. Select all visible elements by clicking and dragging the mouse
m. Click **Apply**

Remark: **Coord 0.1** means the X-axis of the global coordinate system.



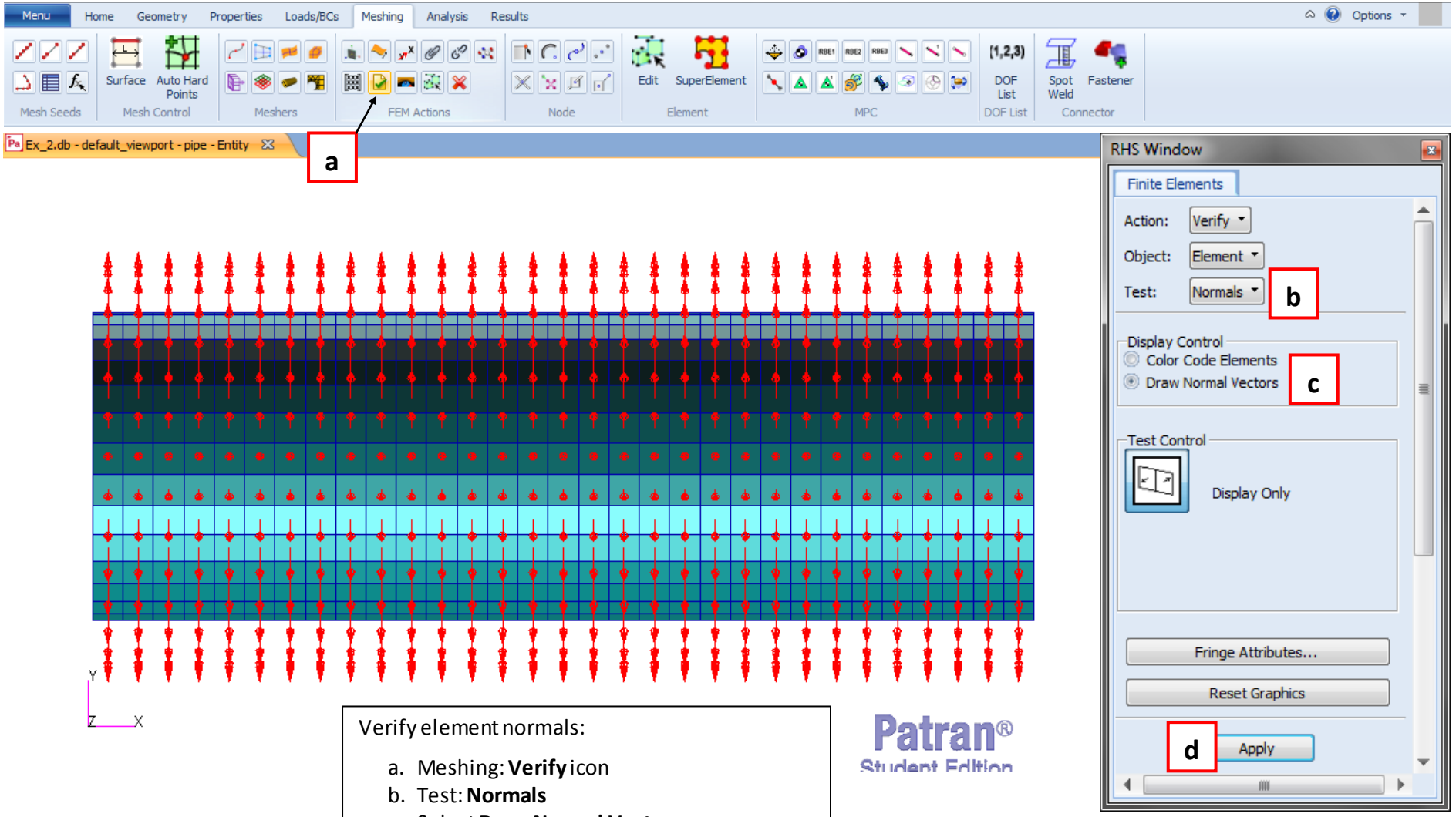
a. Click on the **Smooth shaded** icon from **Home** icon

Delete duplicate nodes:

b. Meshing: **Equivalence** icon *(FEM Actions tab)*

c. Click **Apply**

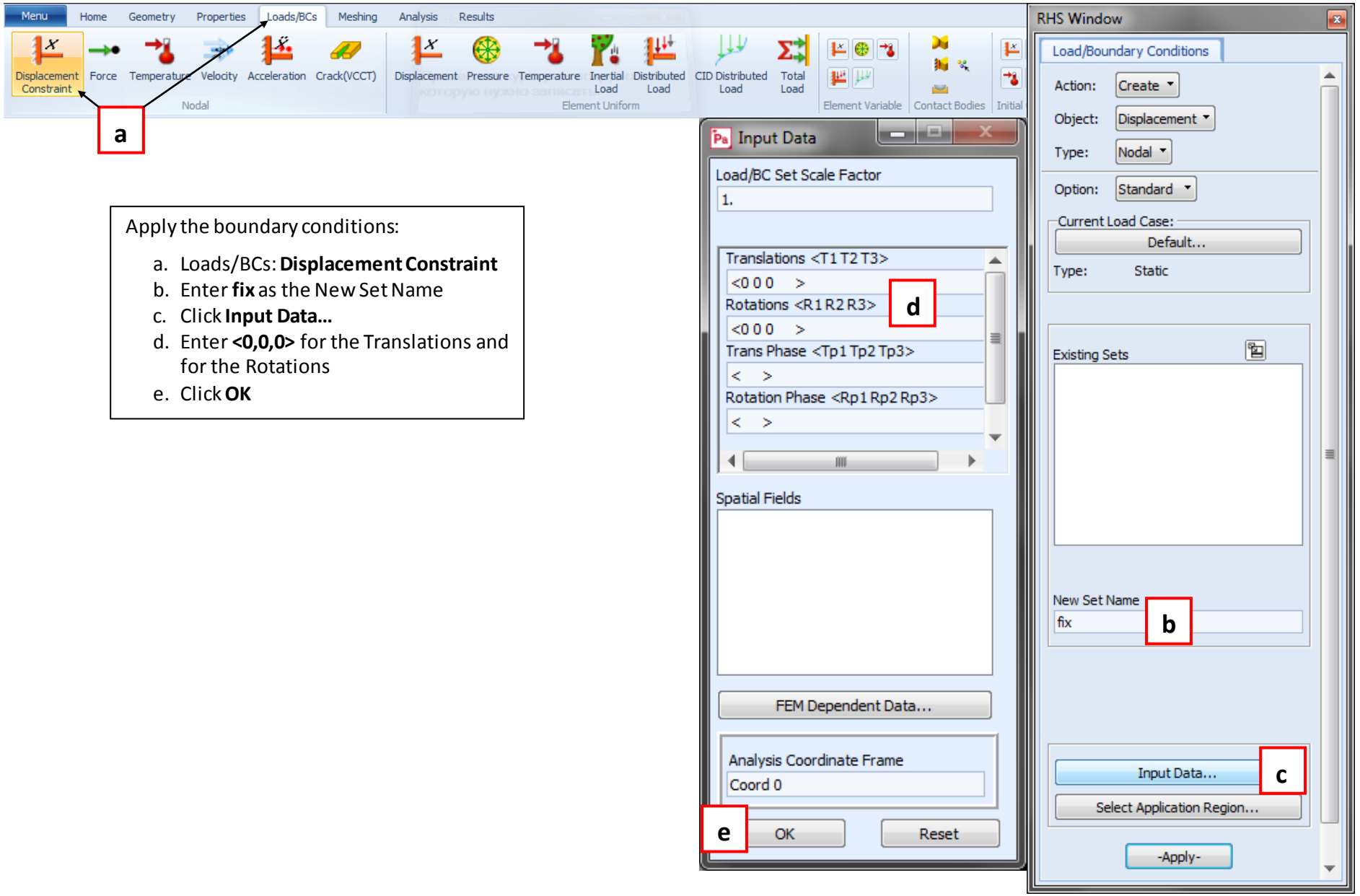
Patran® Student Edition



Verify element normals:

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

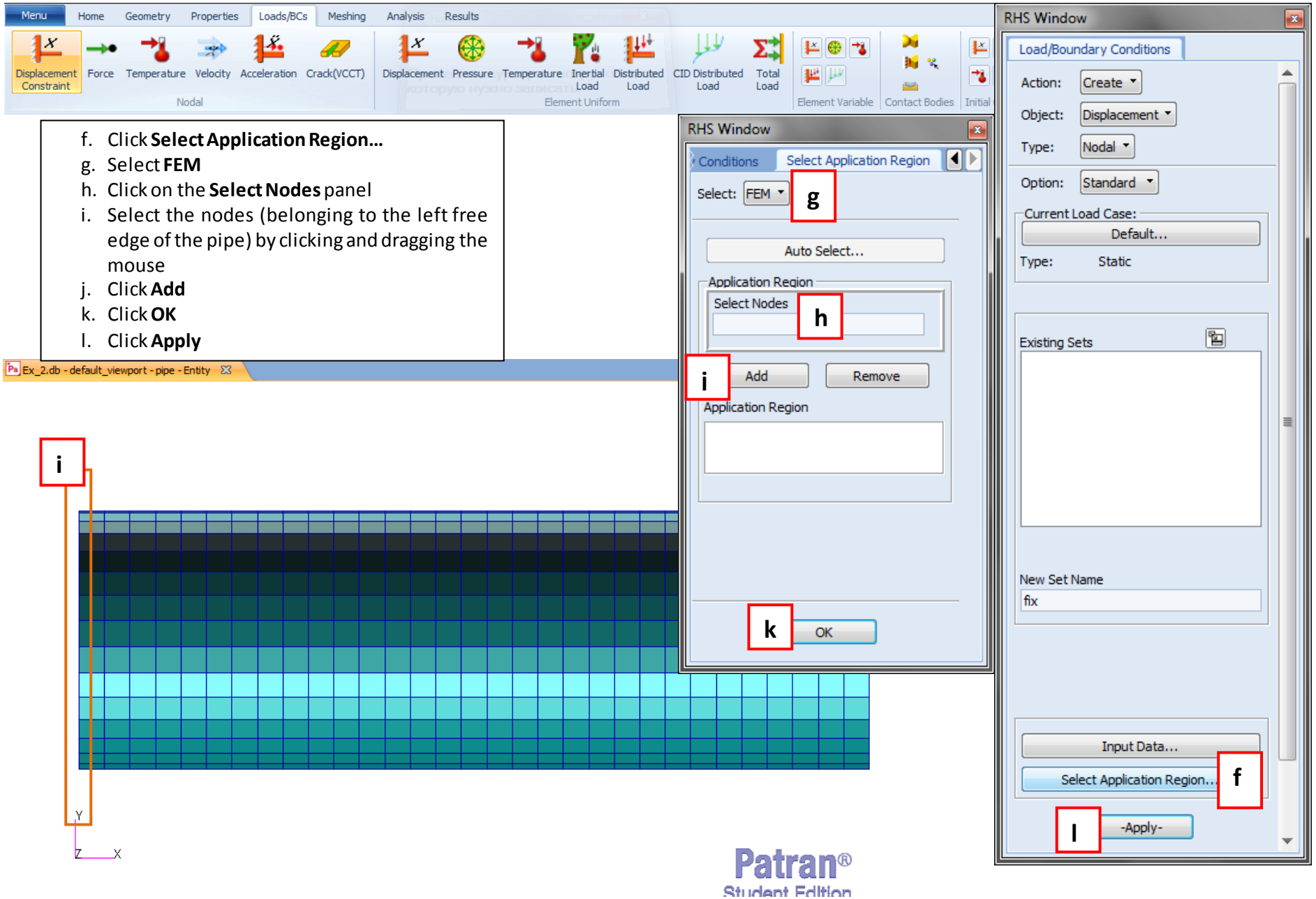
Remark: To reverse element normals use **Elements: Modify/Element/Reverse**

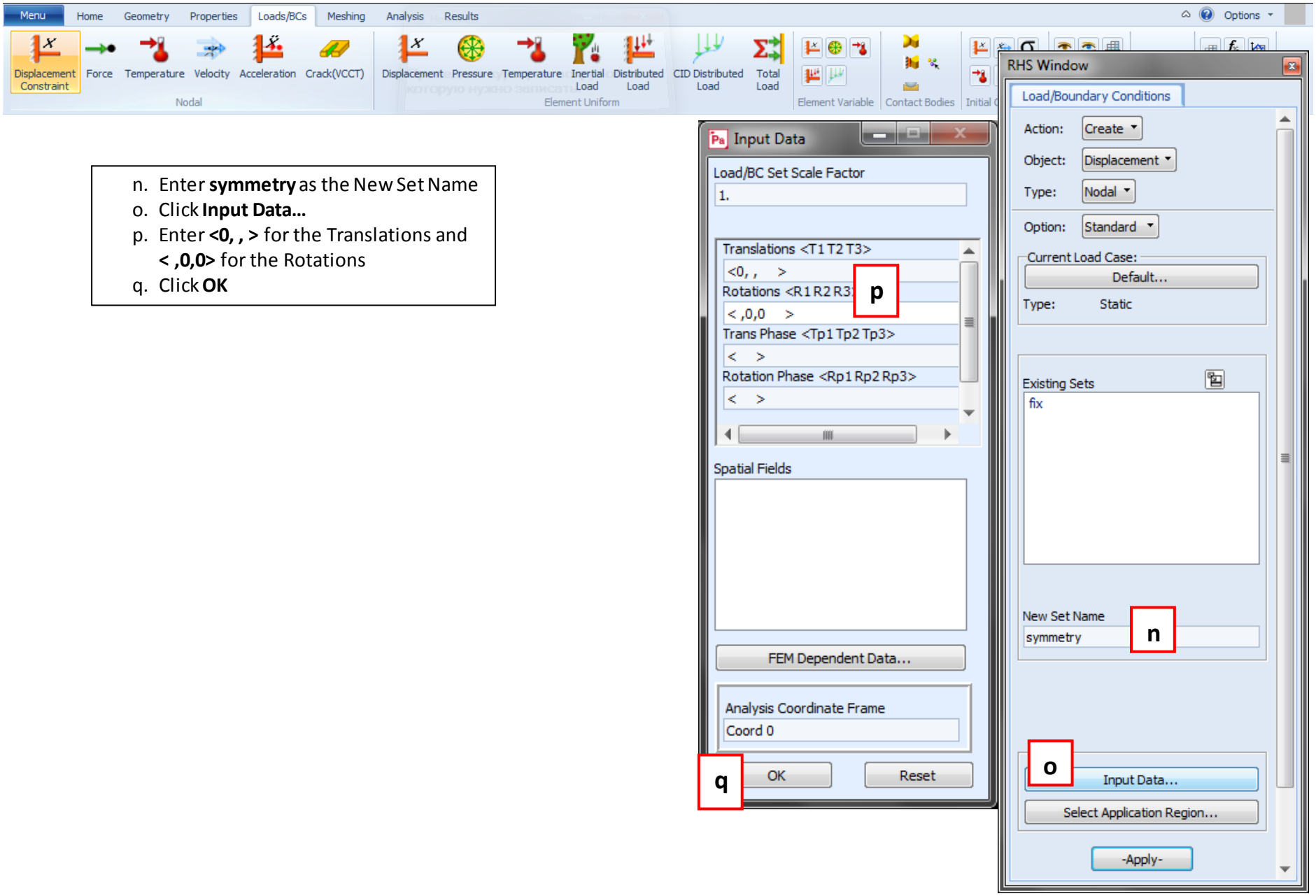


Apply the boundary conditions:

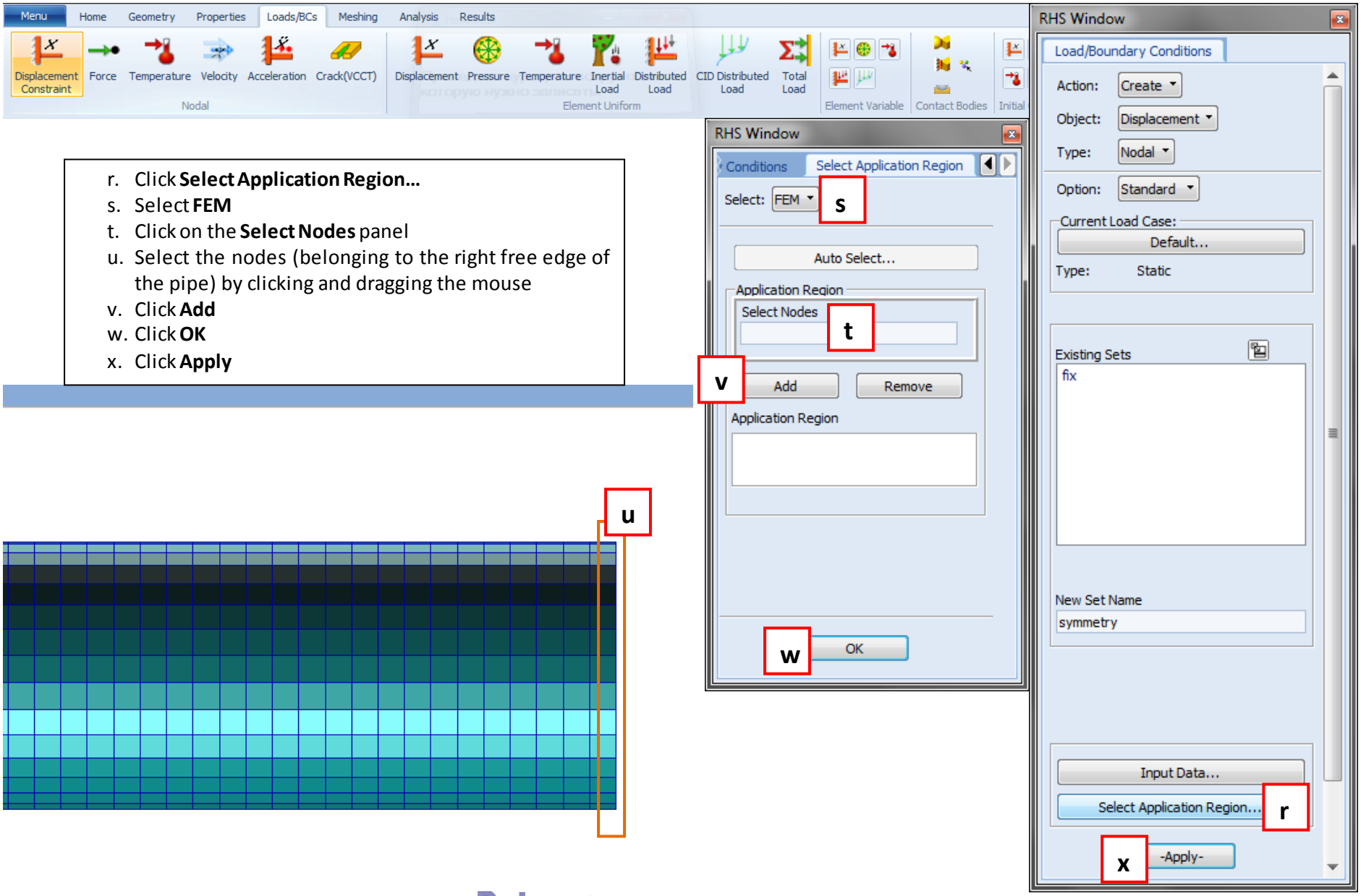
a. Loads/BCs: **Displacement Constraint**
b. Enter **fix** as the New Set Name
c. Click **Input Data...**
d. Enter **<0,0,0>** for the Translations and for the Rotations
e. Click **OK**

f. Click **Select Application Region…**
g. Select **FEM**
h. Click on the **Select Nodes** panel
i. Select the nodes (belonging to the left free edge of the pipe) by clicking and dragging the mouse
j. Click **Add**
k. Click **OK**
l. Click **Apply**



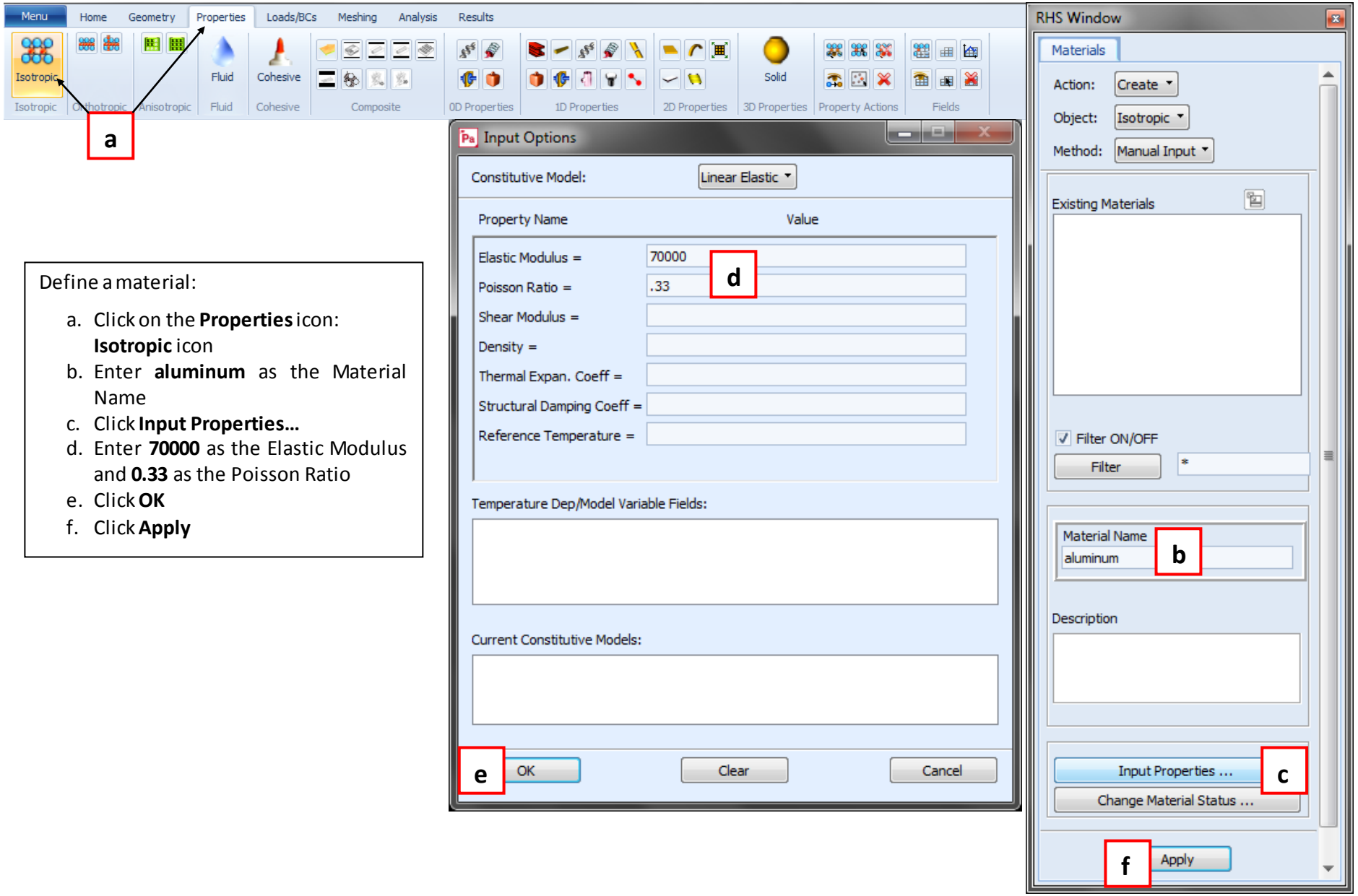


n. Enter **symmetry** as the New Set Name
o. Click **Input Data…**
p. Enter **<0, , >** for the Translations and **< ,0,0>** for the Rotations
q. Click **OK**



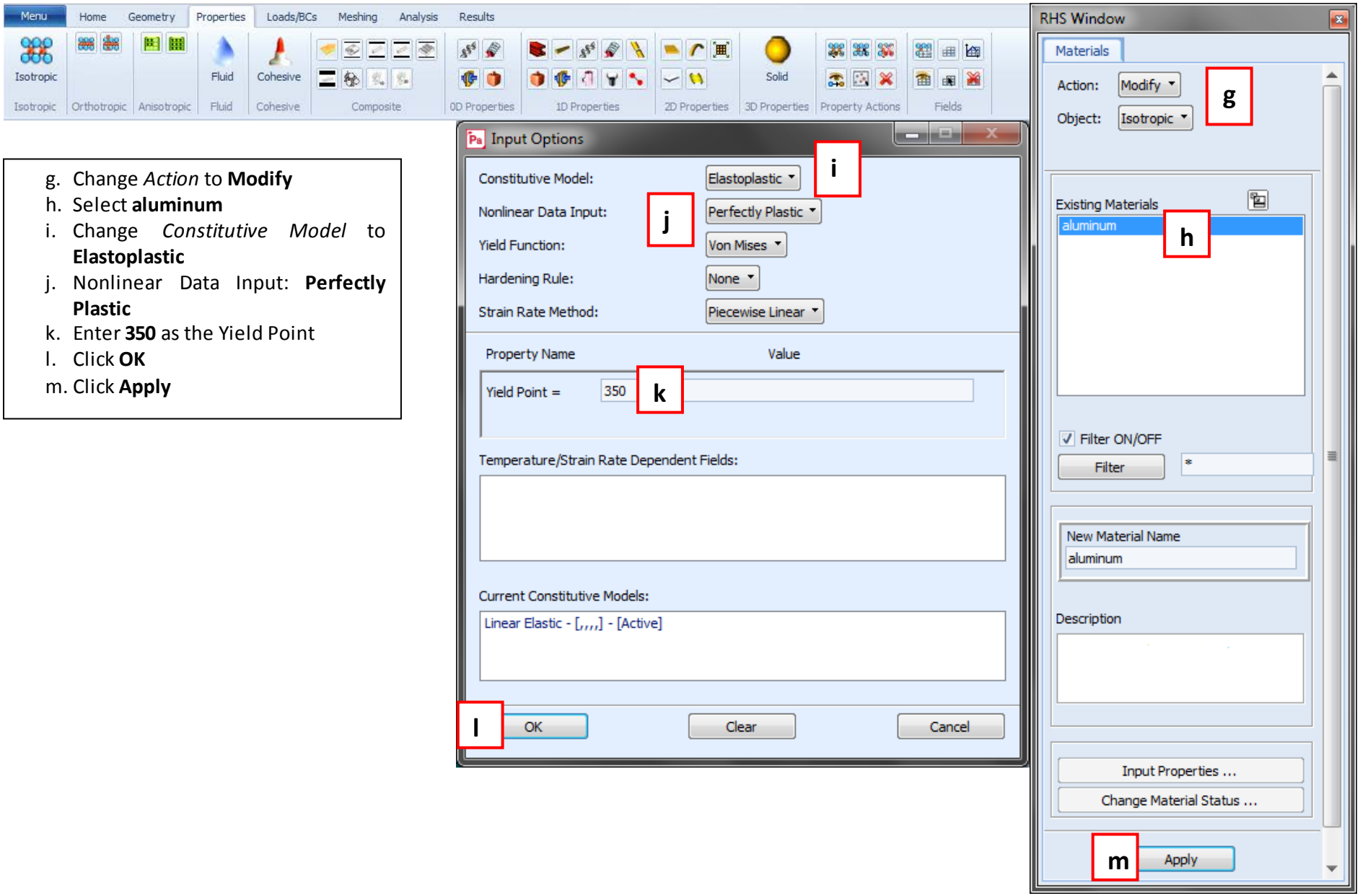
r. Click **Select Application Region…**
s. Select **FEM**
t. Click on the **Select Nodes** panel
u. Select the nodes (belonging to the right free edge of the pipe) by clicking and dragging the mouse
v. Click **Add**
w. Click **OK**
x. Click **Apply**

Patran® Student Edition

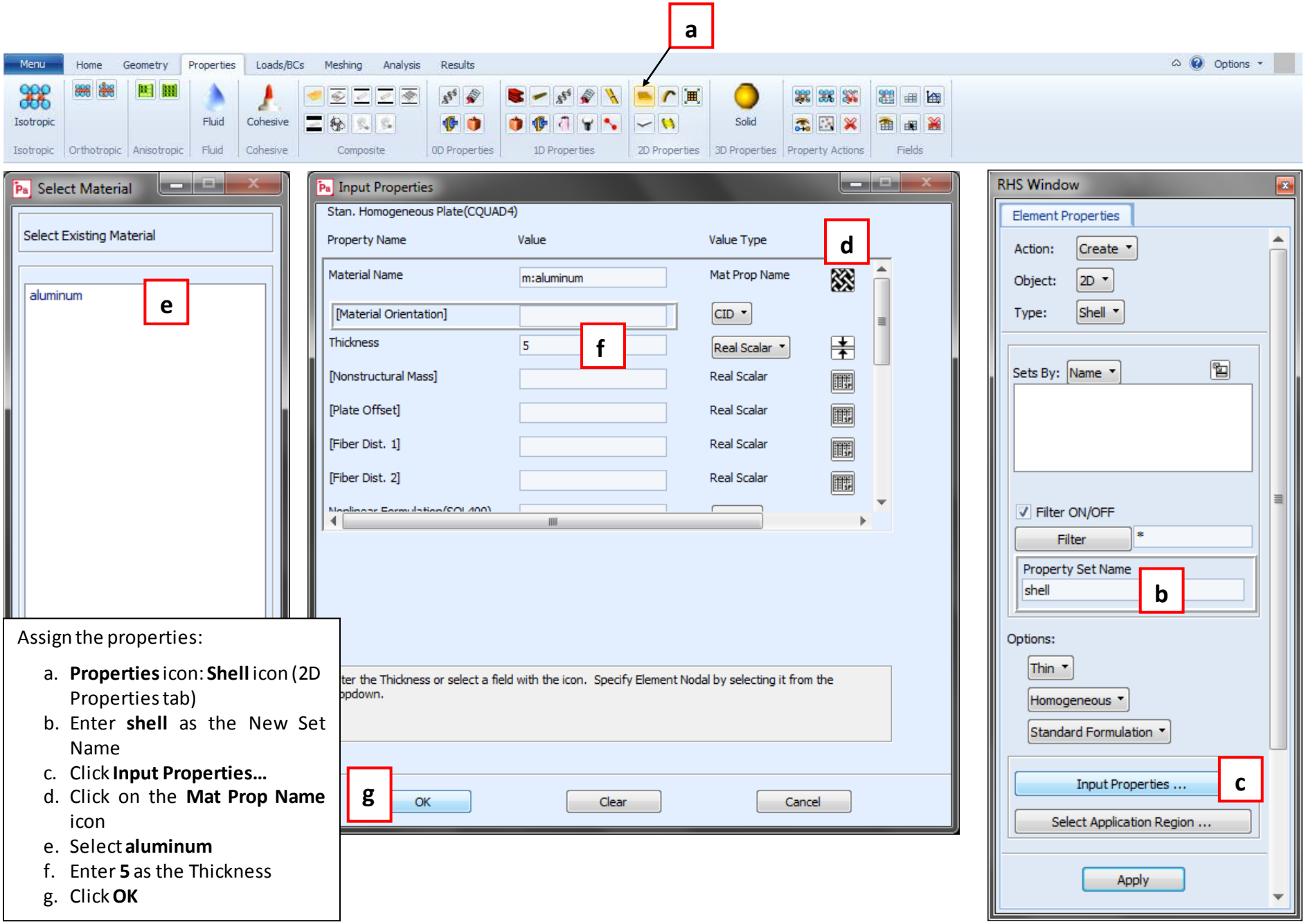


Define a material:

a. Click on the **Properties** icon: **Isotropic** icon
b. Enter **aluminum** as the Material Name
c. Click **Input Properties…**
d. Enter **70000** as the Elastic Modulus and **0.33** as the Poisson Ratio
e. Click **OK**
f. Click **Apply**

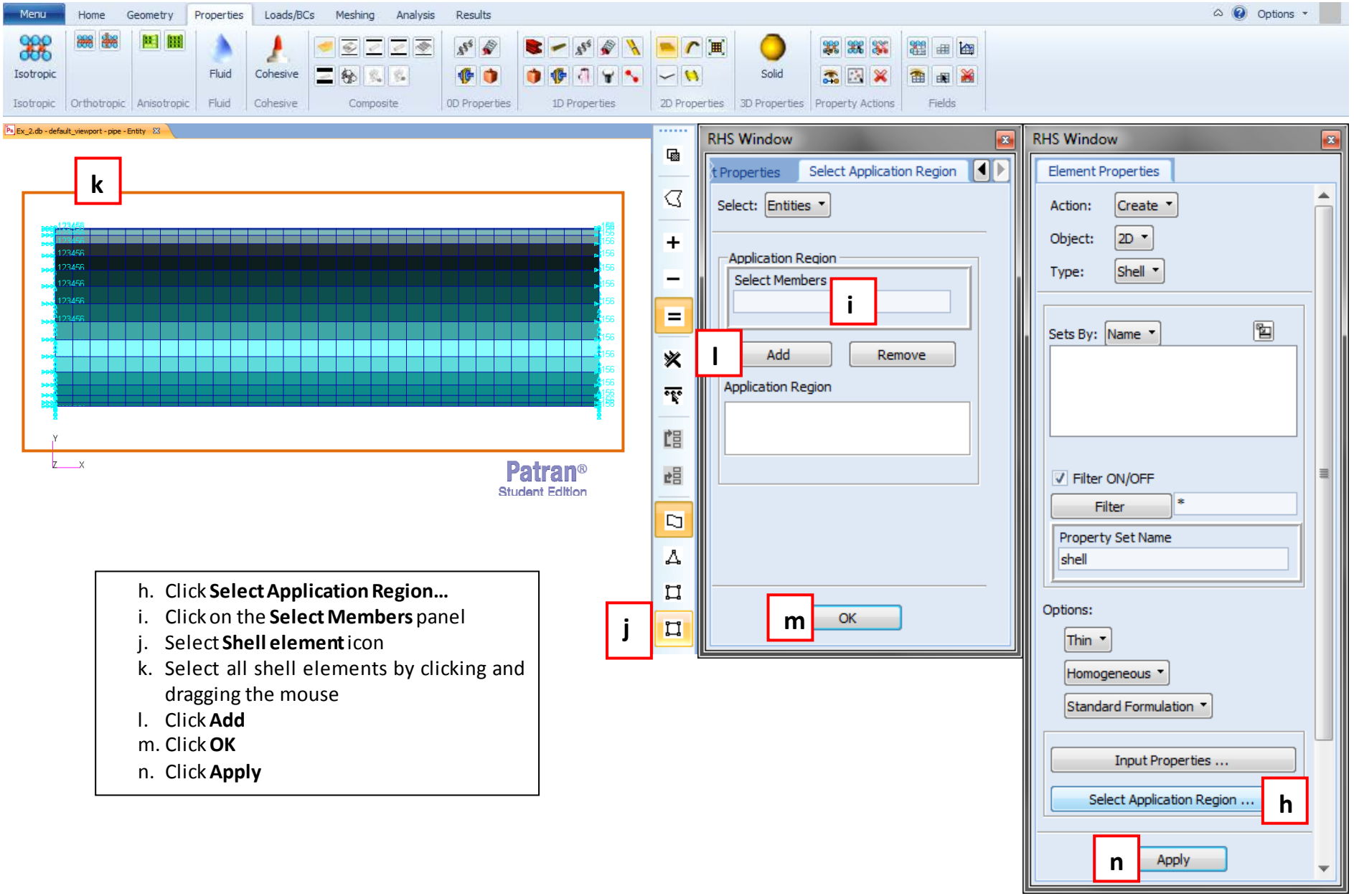


g. Change *Action* to **Modify**
h. Select **aluminum**
i. Change *Constitutive Model* to **Elastoplastic**
j. Nonlinear Data Input: **Perfectly Plastic**
k. Enter **350** as the Yield Point
l. Click **OK**
m. Click **Apply**

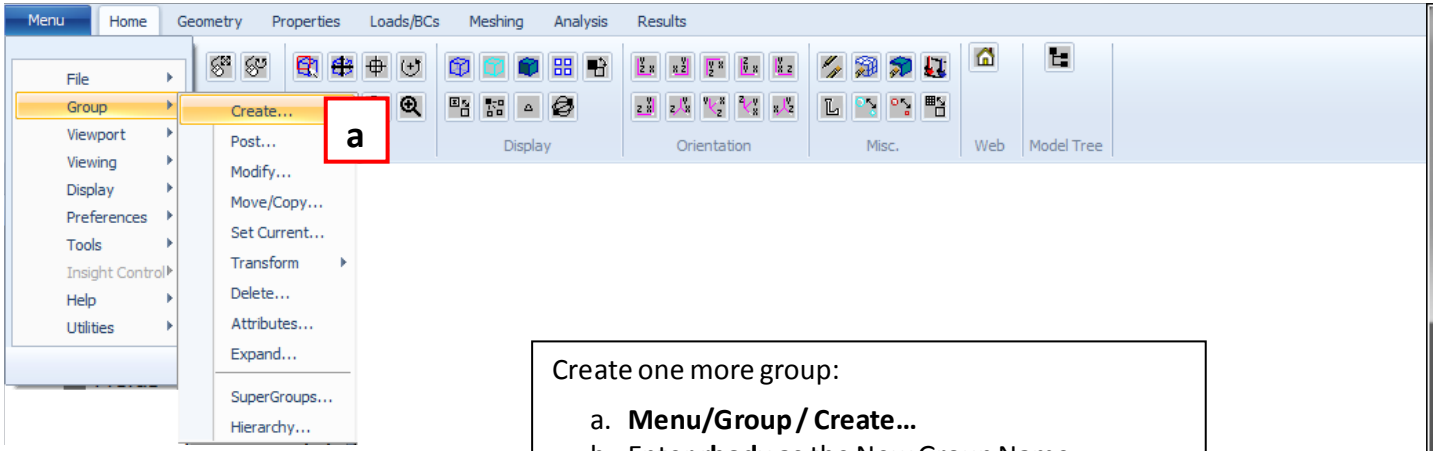


Assign the properties:

a. **Properties** icon: **Shell** icon (2D Properties tab)
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 **5** as the Thickness
g. Click **OK**



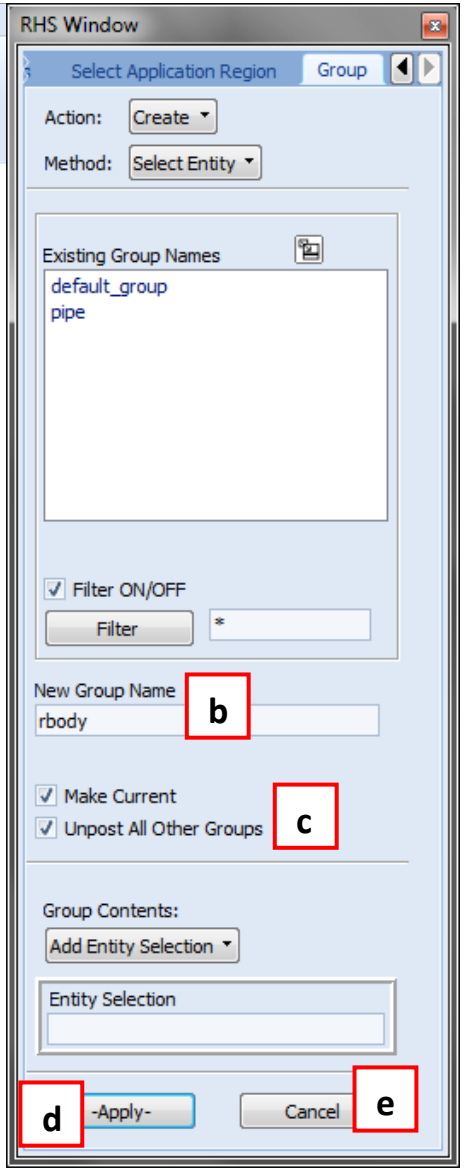
h. Click **Select Application Region…**
i. Click on the **Select Members** panel
j. Select **Shell element** icon
k. Select all shell elements by clicking and dragging the mouse
l. Click **Add**
m. Click **OK**
n. Click **Apply**

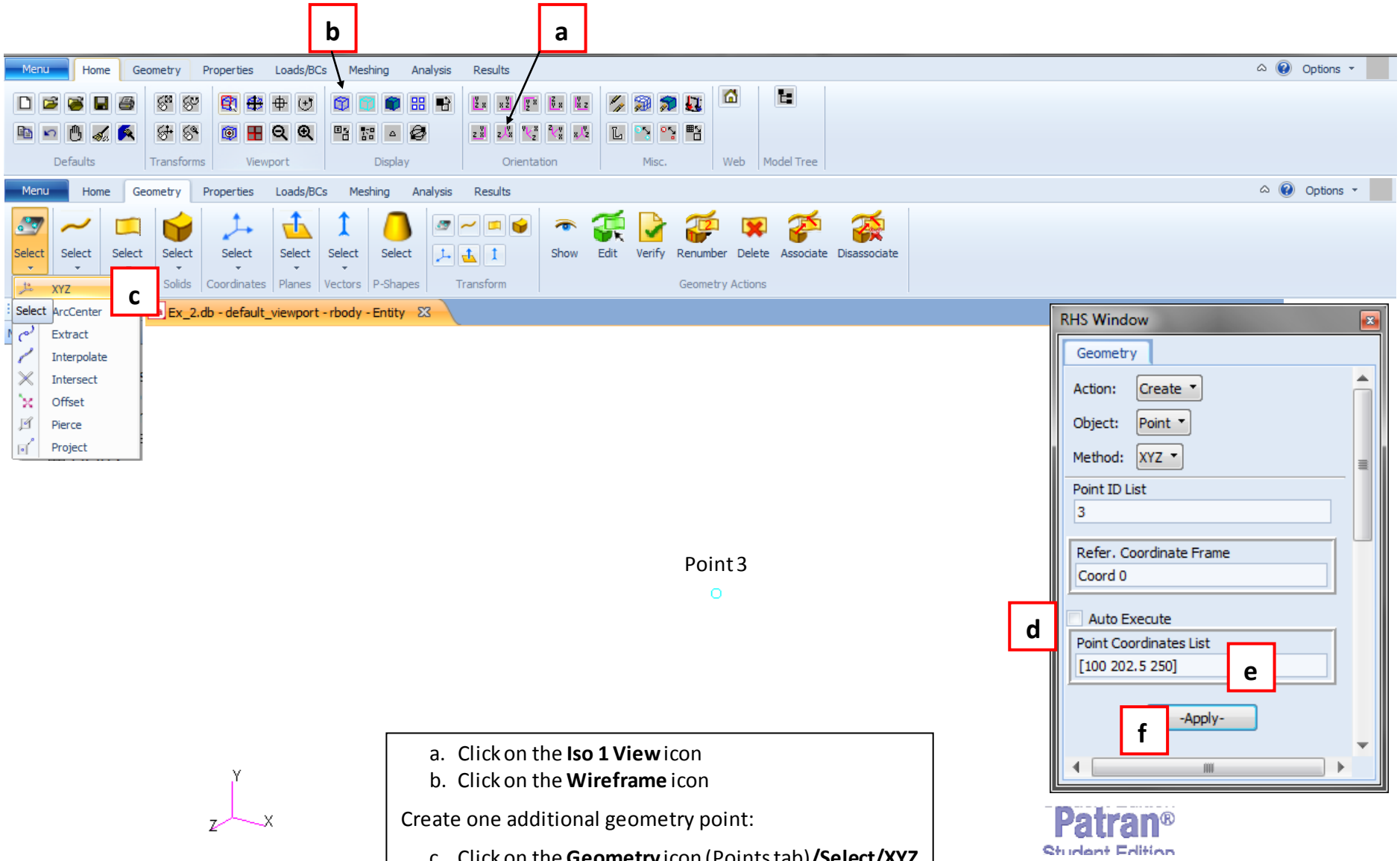


Create one more group:

a. **Menu/Group / Create…**
b. Enter **rbody** as the New Group Name
c. Check **Make Current** and check **Unpost All Other Groups**
d. Click **Apply**
e. Click **Cancel**

Remark: A moving rigid body (half of a cylinder) will be assigned to this group.

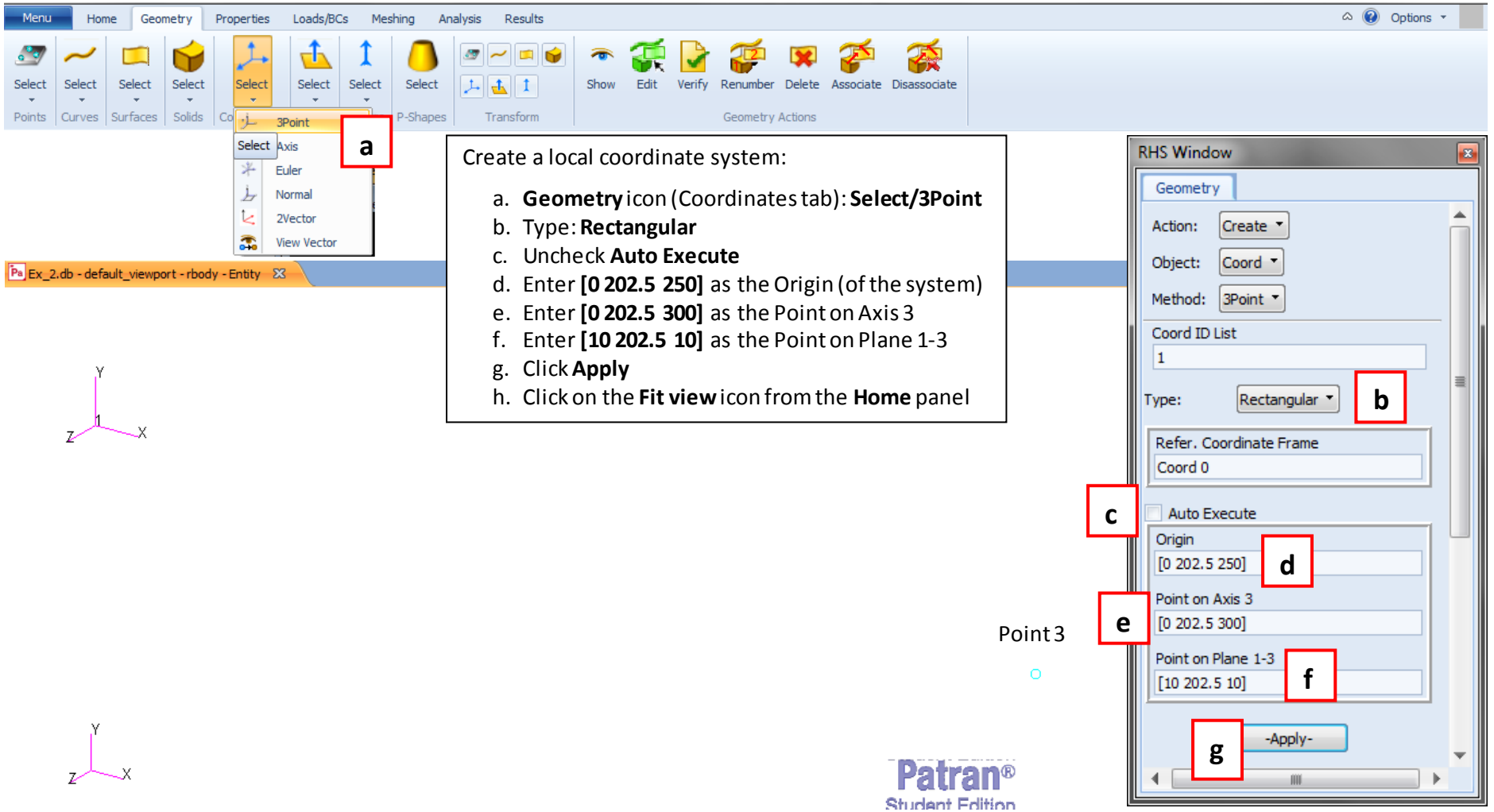


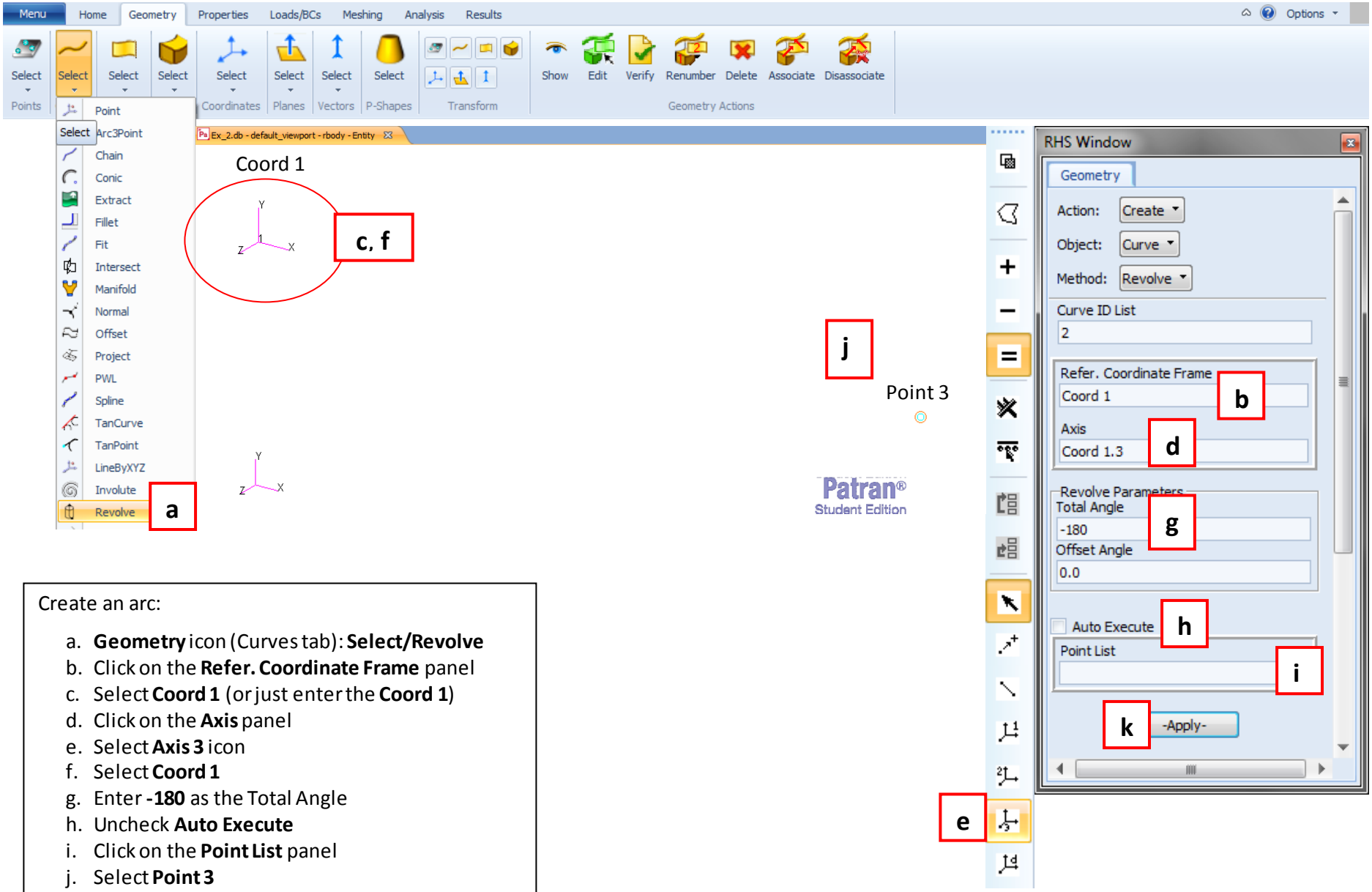


a. Click on the **Iso 1 View** icon
b. Click on the **Wireframe** icon

Create one additional geometry point:

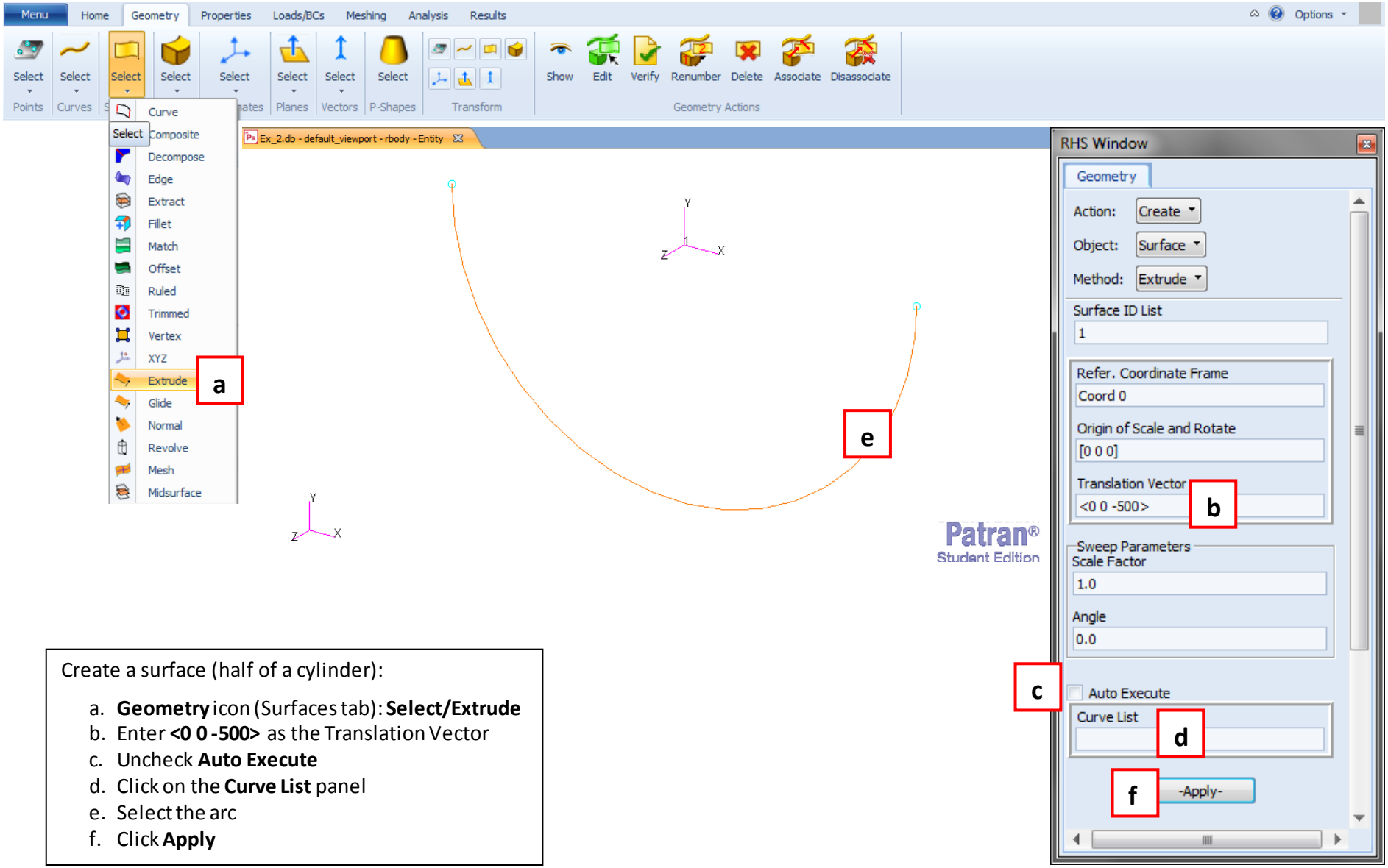
c. Click on the **Geometry** icon (Points tab)**/Select/XYZ**
d. Uncheck **Auto Execute**
e. Enter **[100 202.5 250]** as the Point Coordinates List
f. Click **Apply**





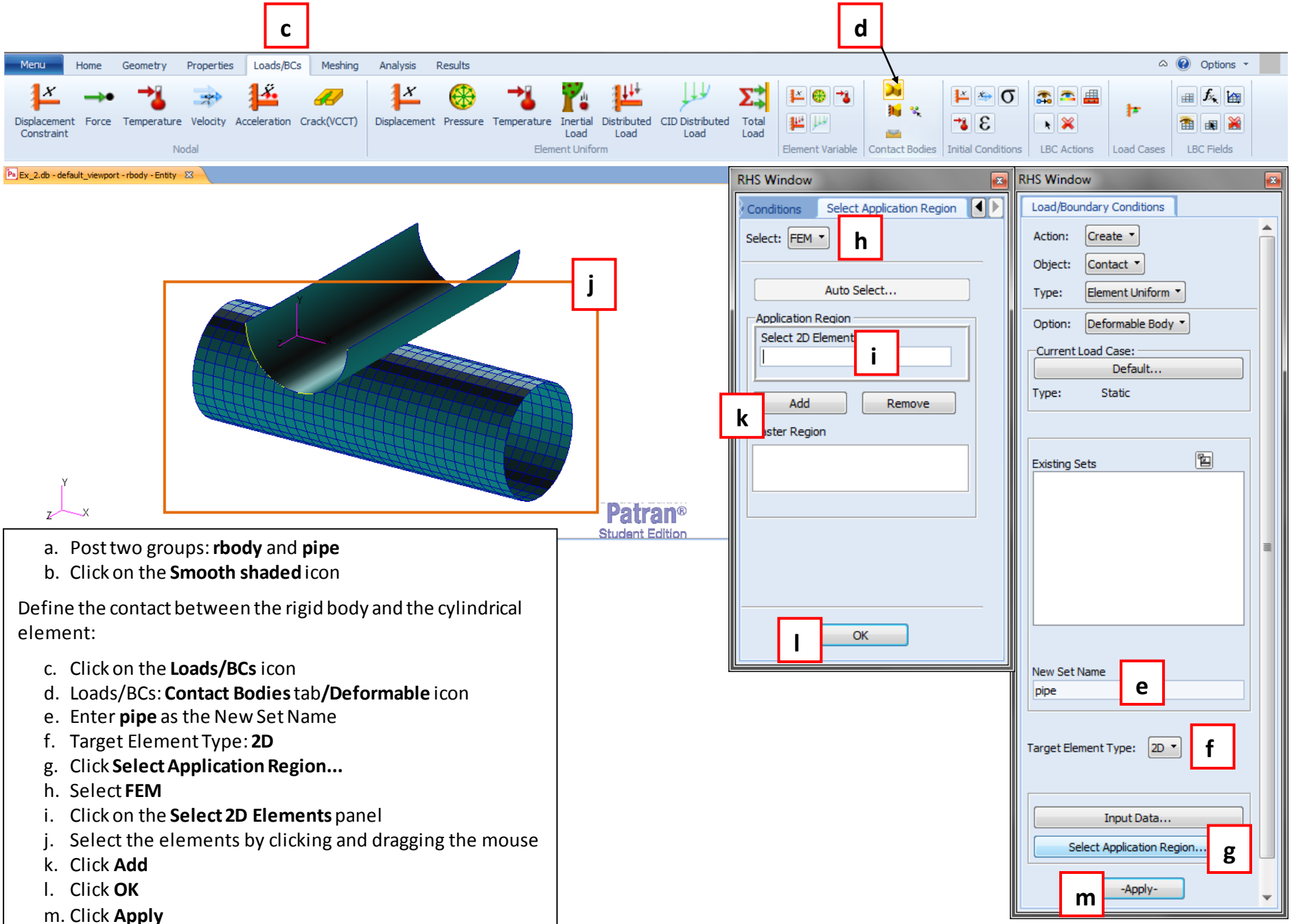
Create an arc:

a. **Geometry** icon (Curves tab): **Select/Revolve**
b. Click on the **Refer. Coordinate Frame** panel
c. Select **Coord 1** (or just enter the **Coord 1**)
d. Click on the **Axis** panel
e. Select **Axis 3** icon
f. Select **Coord 1**
g. Enter **-180** as the Total Angle
h. Uncheck **Auto Execute**
i. Click on the **Point List** panel
j. Select **Point 3**
k. Click **Apply**



Create a surface (half of a cylinder):

a. **Geometry** icon (Surfaces tab): **Select/Extrude**
b. Enter **<0 0 -500>** as the Translation Vector
c. Uncheck **Auto Execute**
d. Click on the **Curve List** panel
e. Select the arc
f. Click **Apply**

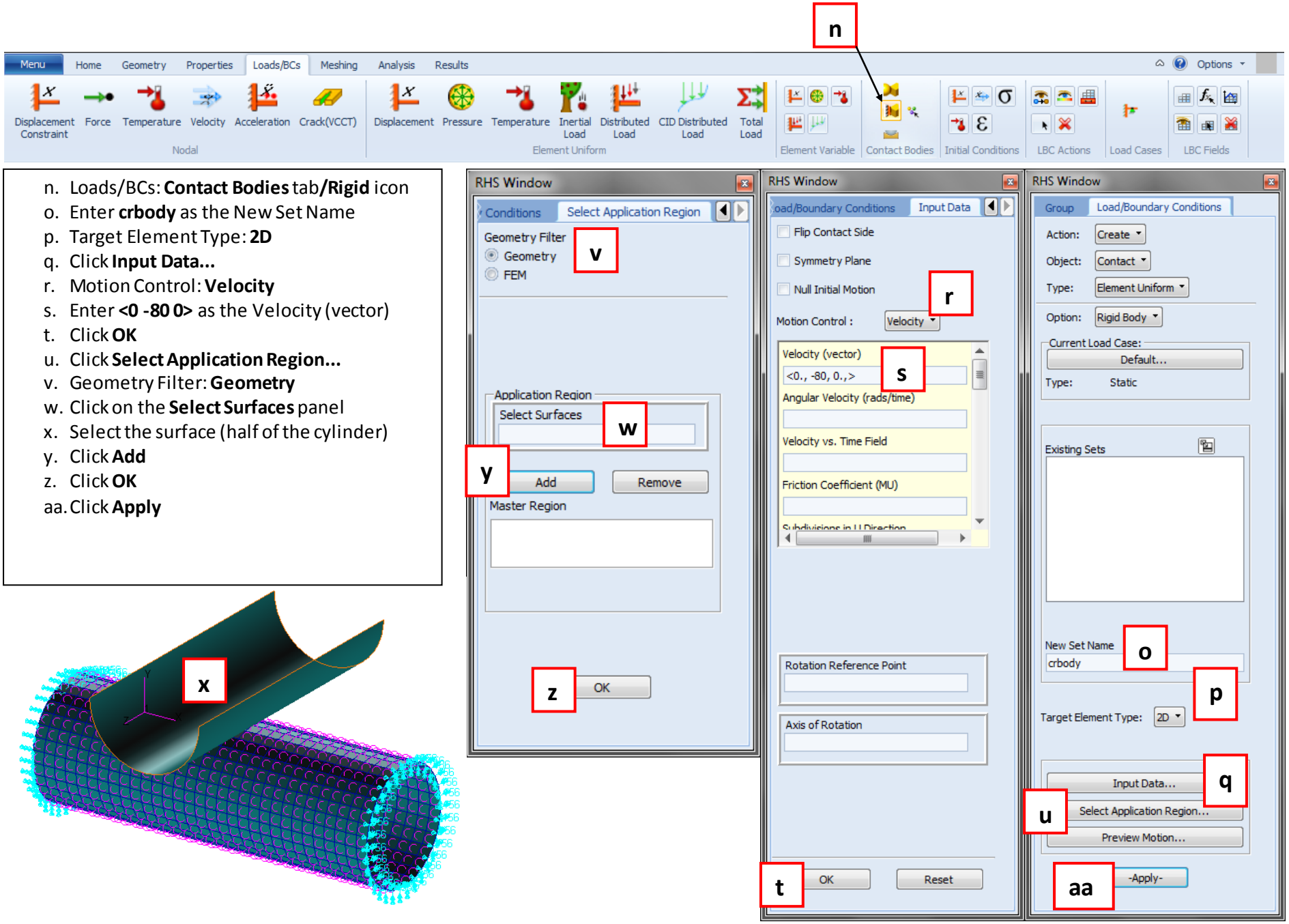


a. Post two groups: **rbody** and **pipe**
b. Click on the **Smooth shaded** icon

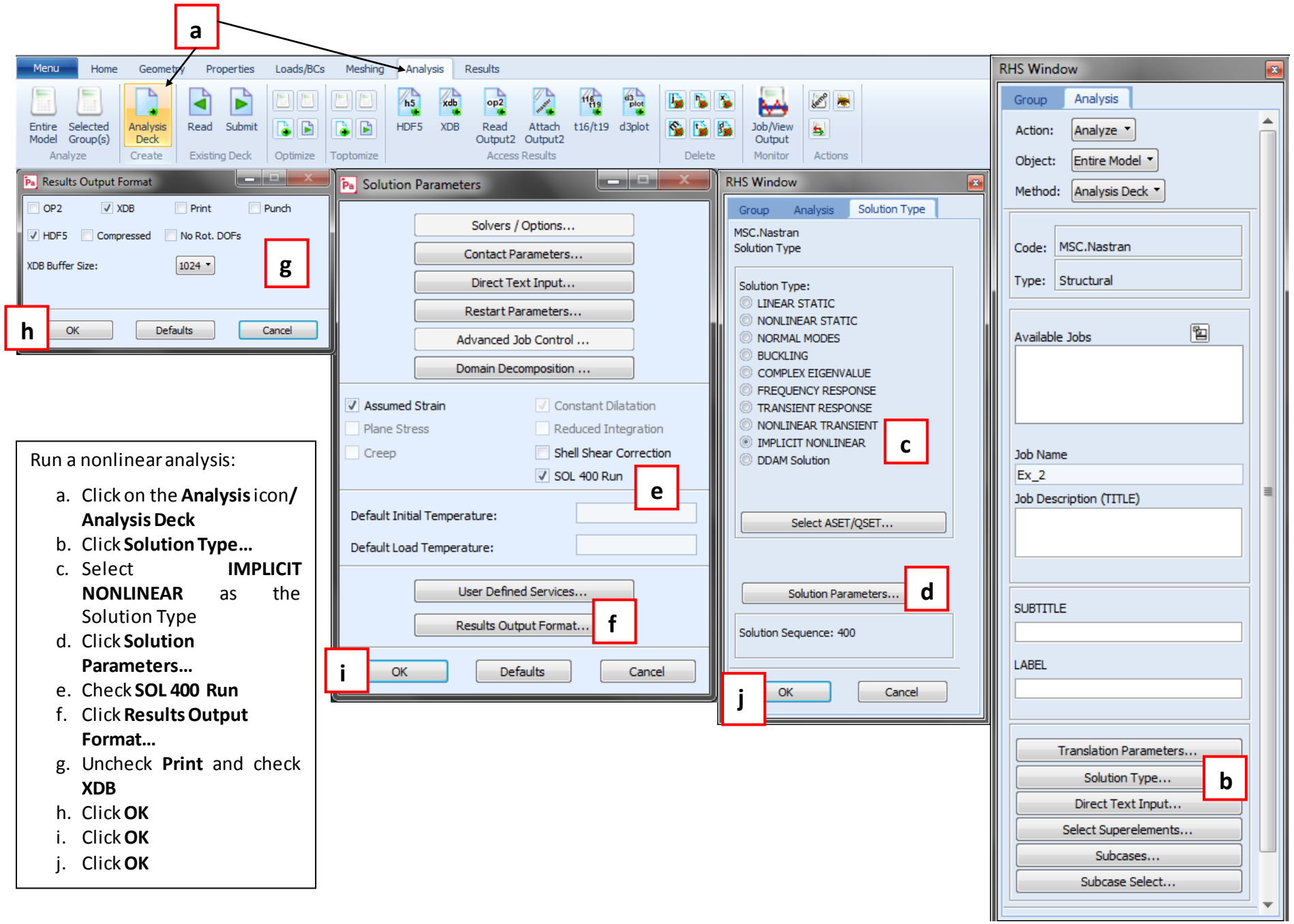
Define the contact between the rigid body and the cylindrical element:

c. Click on the **Loads/BCs** icon
d. Loads/BCs: **Contact Bodies** tab/**Deformable** icon
e. Enter **pipe** as the New Set Name
f. Target Element Type: **2D**
g. Click **Select Application Region...**
h. Select **FEM**
i. Click on the **Select 2D Elements** panel
j. Select the elements by clicking and dragging the mouse
k. Click **Add**
l. Click **OK**
m. Click **Apply**

HUMAN CAPITAL
NATIONAL COHESION STRATEGY
EUROPEAN UNION
EUROPEAN
SOCIAL FUND

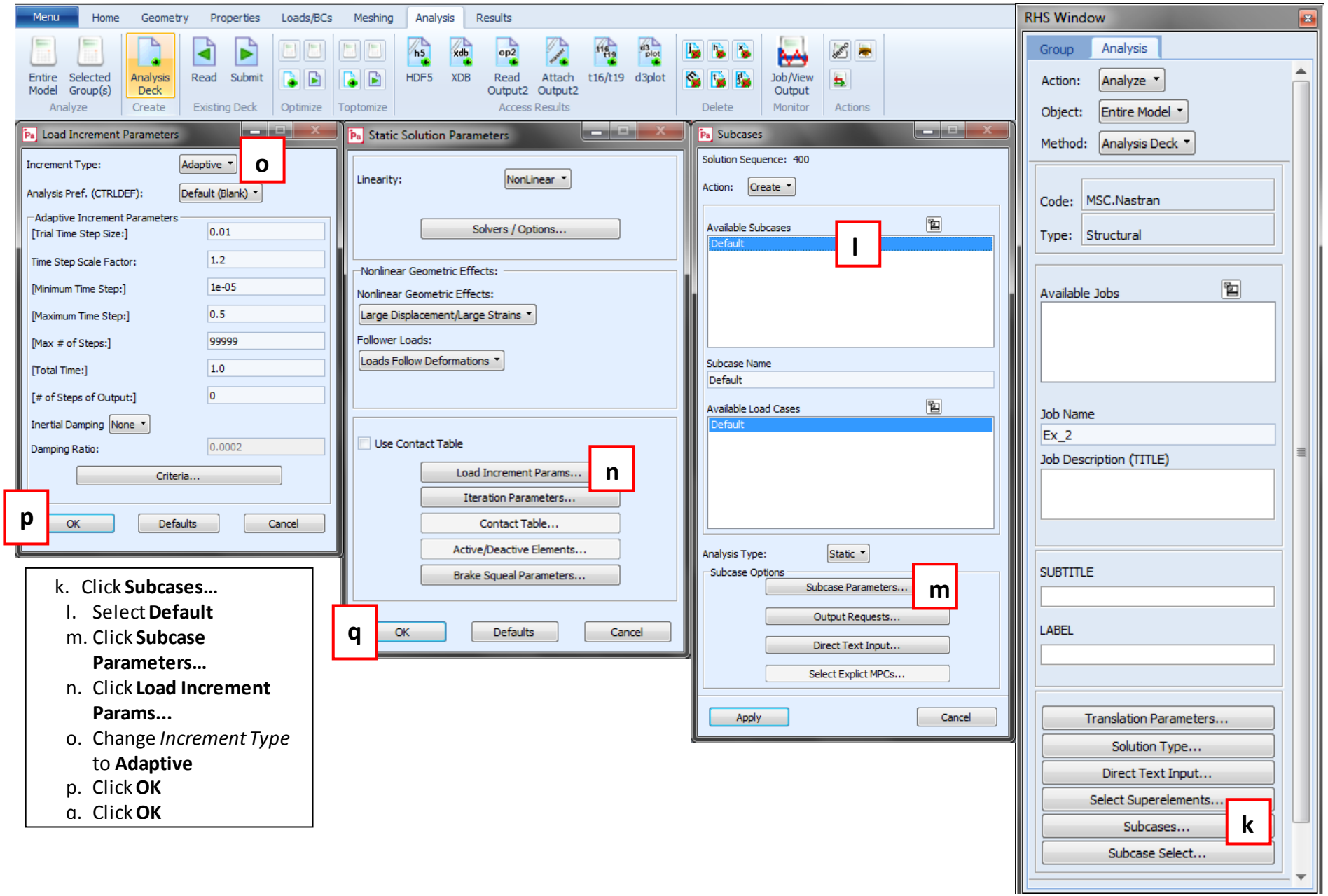


n. Loads/BCs: **Contact Bodies** tab/**Rigid** icon
o. Enter **crbody** as the New Set Name
p. Target Element Type: **2D**
q. Click **Input Data...**
r. Motion Control: **Velocity**
s. Enter **<0 -80 0>** as the Velocity (vector)
t. Click **OK**
u. Click **Select Application Region...**
v. Geometry Filter: **Geometry**
w. Click on the **Select Surfaces** panel
x. Select the surface (half of the cylinder)
y. Click **Add**
z. Click **OK**
aa. Click **Apply**



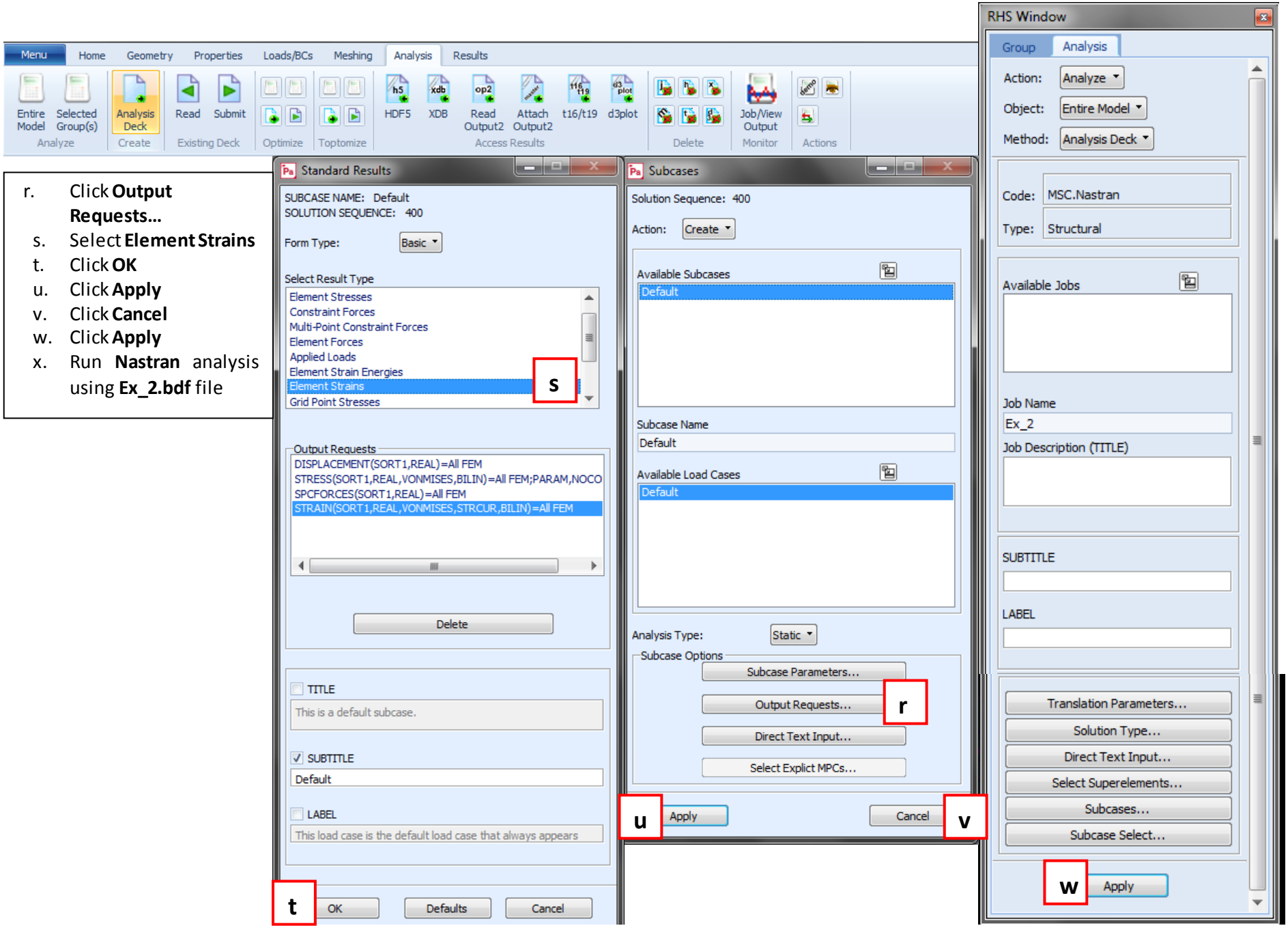
Run a nonlinear analysis:

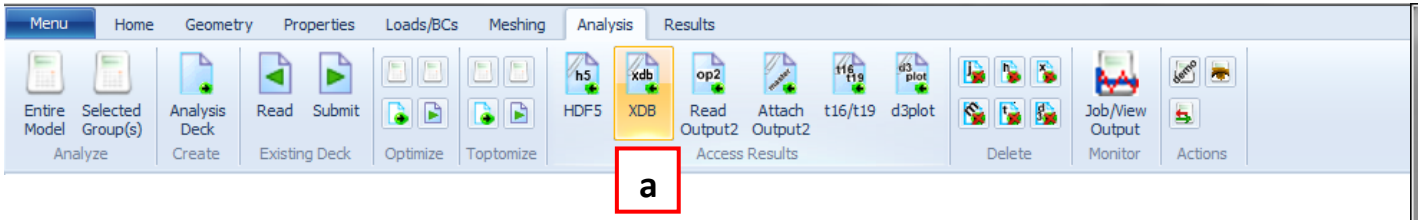
a. Click on the **Analysis** icon/ **Analysis Deck**
b. Click **Solution Type...**
c. Select **IMPLICIT NONLINEAR** as the Solution Type
d. Click **Solution Parameters...**
e. Check **SOL 400 Run**
f. Click **Results Output Format...**
g. Uncheck **Print** and check **XDB**
h. Click **OK**
i. Click **OK**
j. Click **OK**



k. Click **Subcases...**
   l. Select **Default**
   m. Click **Subcase Parameters...**
   n. Click **Load Increment Params...**
   o. Change *Increment Type* to **Adaptive**
   p. Click **OK**
   q. Click **OK**

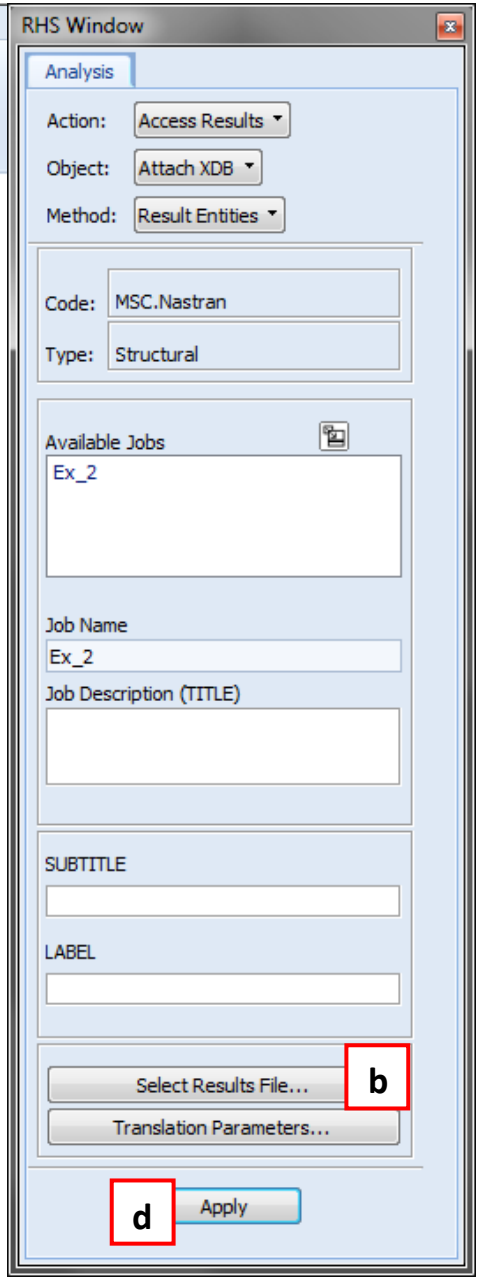
r. Click **Output Requests…**
s. Select **Element Strains**
t. Click **OK**
u. Click **Apply**
v. Click **Cancel**
w. Click **Apply**
x. Run **Nastran** analysis using **Ex_2.bdf** file

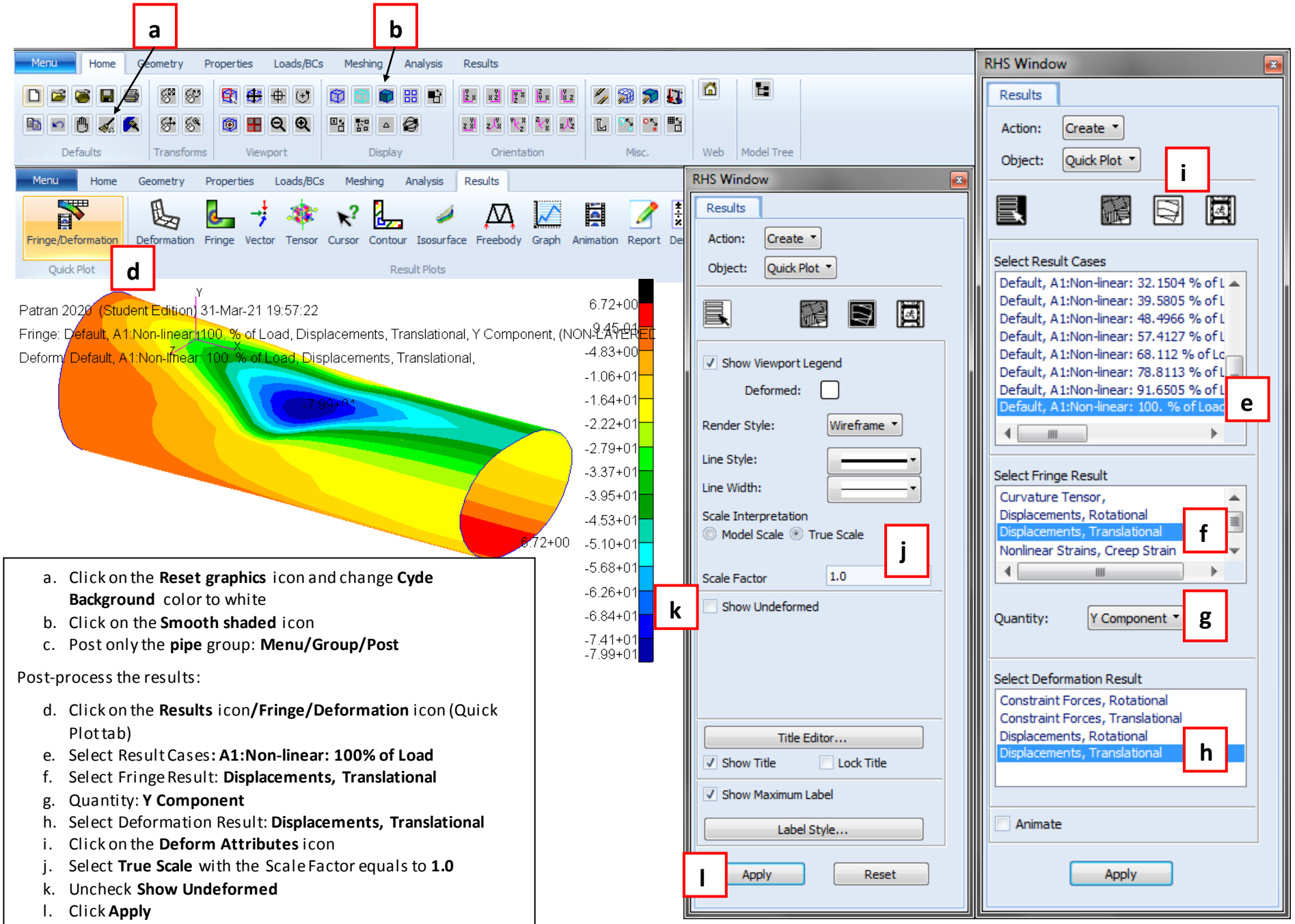




Attach the results file, when the analysis job is completed:

a. Click on the **Analysis** tab**/XDB** icon
b. Click **Select Results File…**
c. Select **Ex_2.xdb** file and click **OK**
d. Click **Apply**

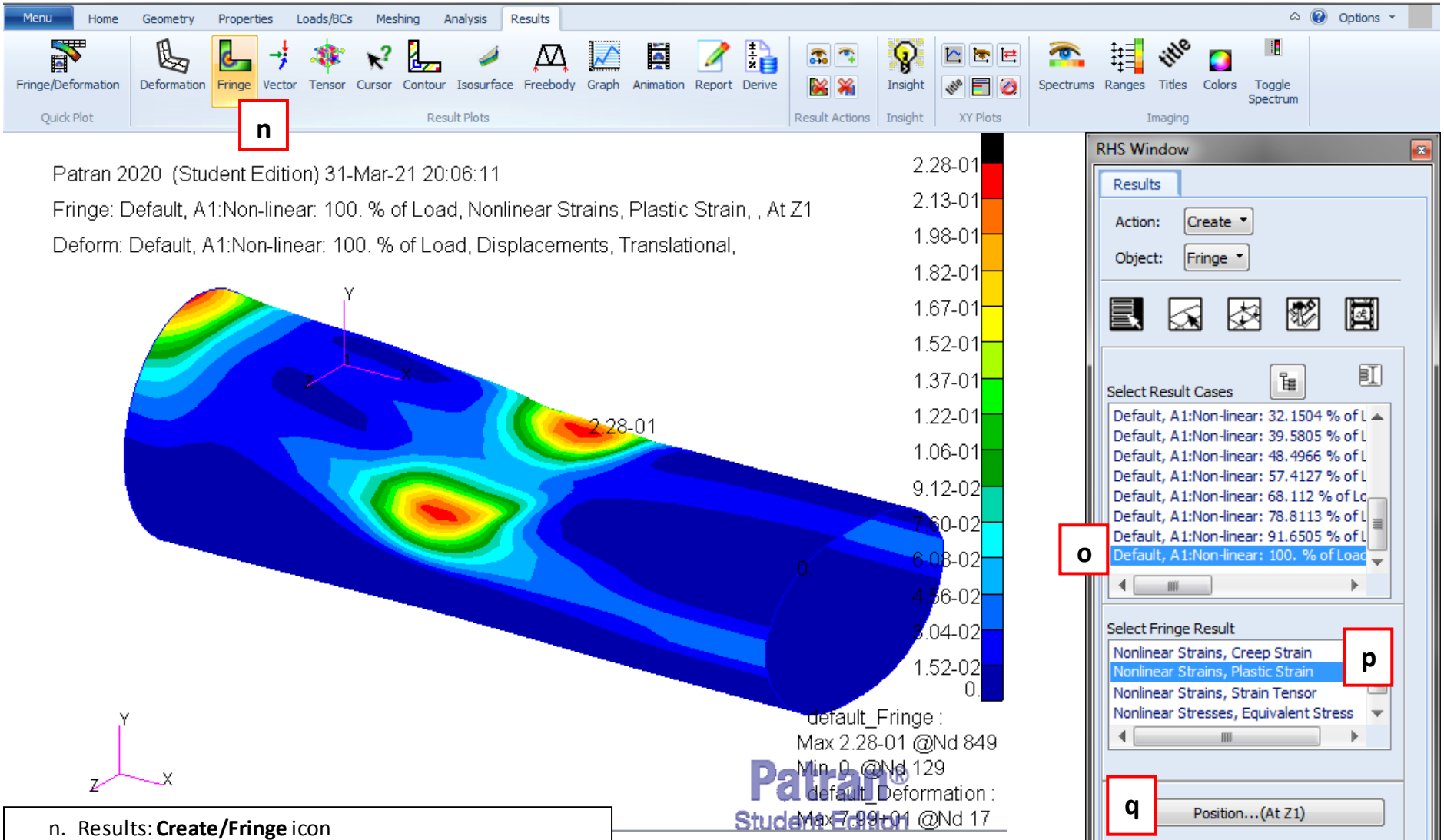




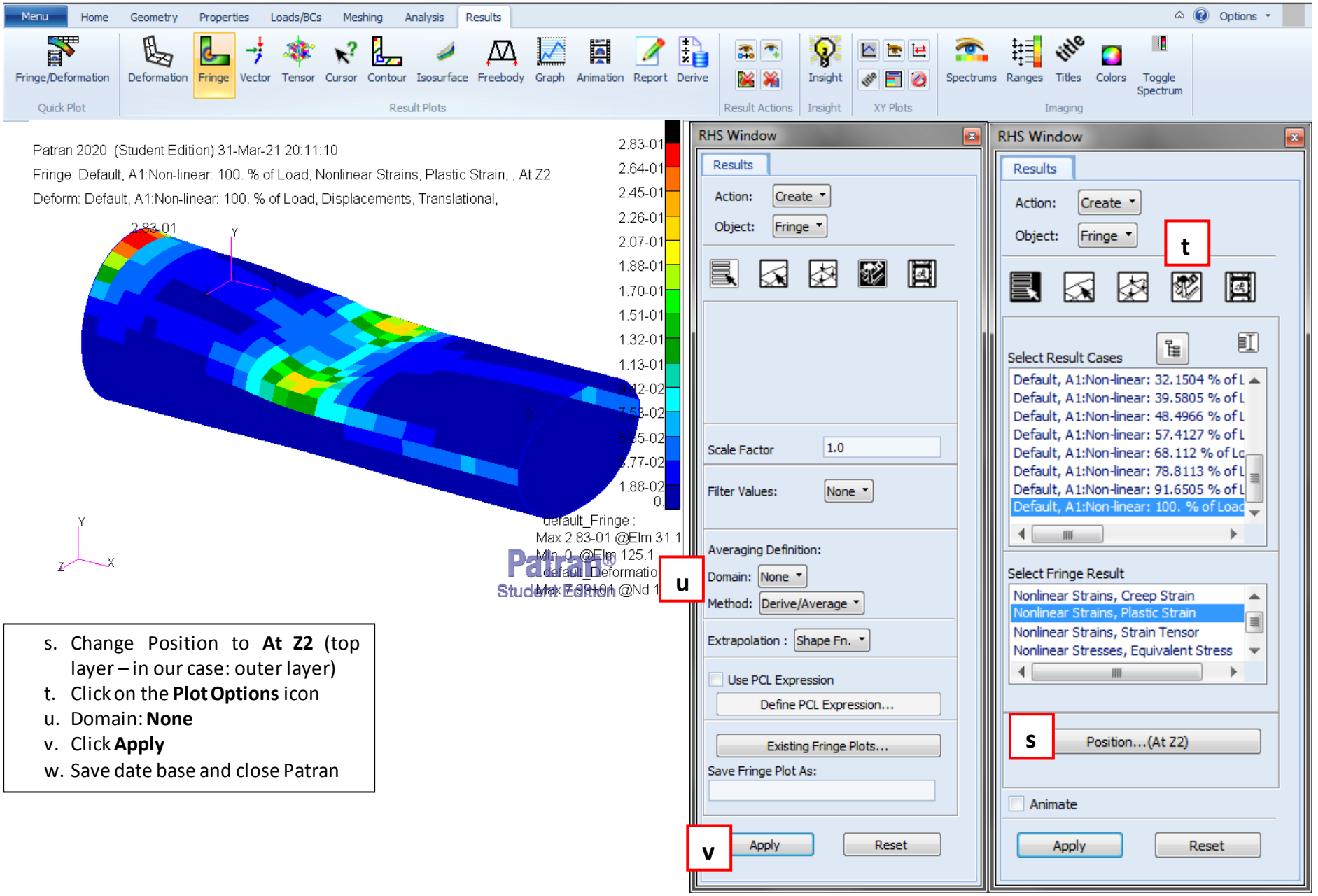
a. Click on the **Reset graphics** icon and change **Cyde Background** color to white
b. Click on the **Smooth shaded** icon
c. Post only the **pipe** group: **Menu/Group/Post**

Post-process the results:

d. Click on the **Results** icon**/Fringe/Deformation** icon (Quick Plot tab)
e. Select Result Cases**: A1:Non-linear: 100% of Load**
f. Select Fringe Result: **Displacements, Translational**
g. Quantity: **Y Component**
h. Select Deformation Result: **Displacements, Translational**
i. Click on the **Deform Attributes** icon
j. Select **True Scale** with the Scale Factor equals to **1.0**
k. Uncheck **Show Undeformed**
l. Click **Apply**



n. Results: **Create/Fringe** icon
o. Select Result Cases: **A1:Non-linear: 100% of Load**
p. Select **Nonlinear Strains, Plastic Strain**
q. Position: **At Z1** (bottom layer – in our case: inner layer)
r. Click **Apply**



s. Change Position to **At Z2** (top layer – in our case: outer layer)
t. Click on the **Plot Options** icon
u. Domain: **None**
v. Click **Apply**
w. Save date base and close Patran