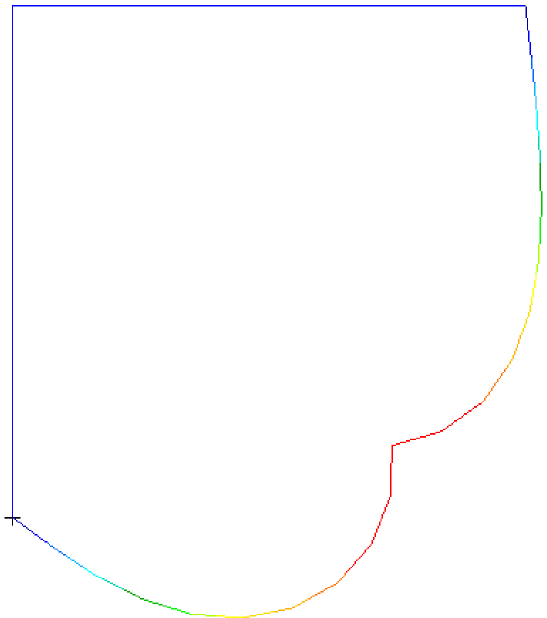
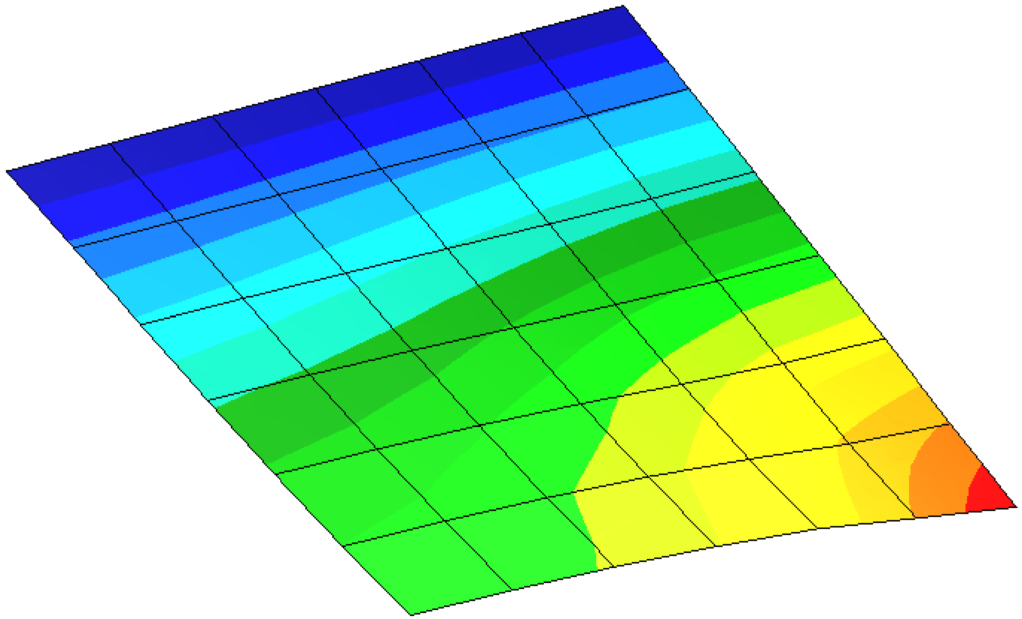
NONLINEAR MECHANICS OF STRUCTURES

EXERCISE 3





HUMAN CAPITAL
NATIONAL COHESION STRATEGY

EUROPEAN UNION
EUROPEAN
SOCIAL FUND

PROBLEM DESCRIPTION

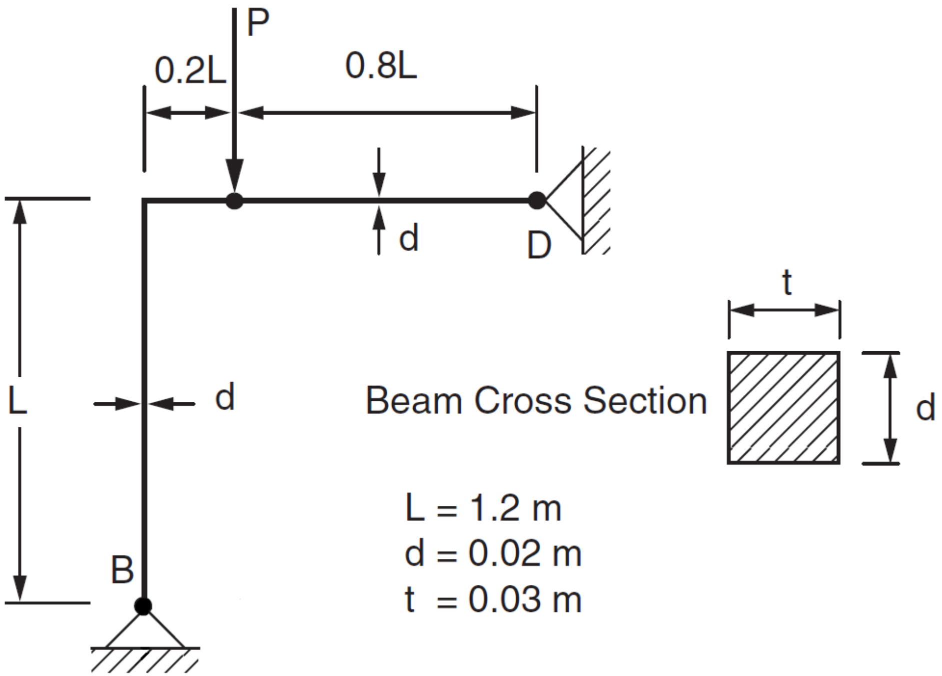


Fig. 1 Lee's Frame Buckling Problem

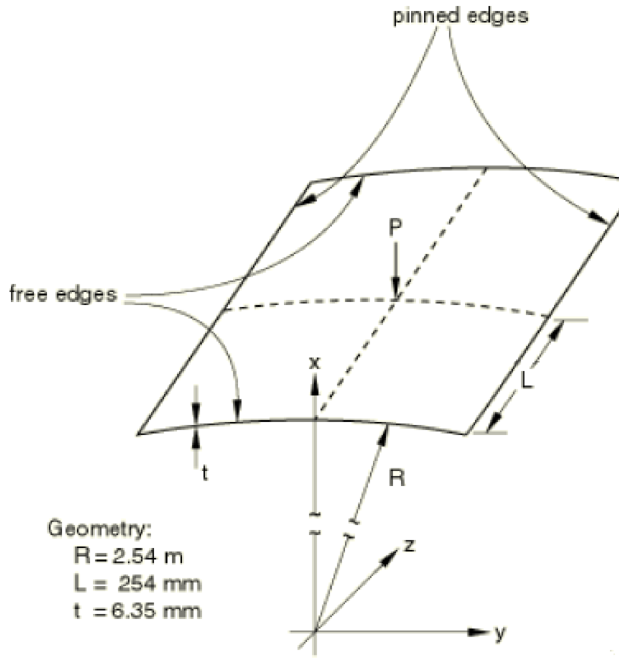
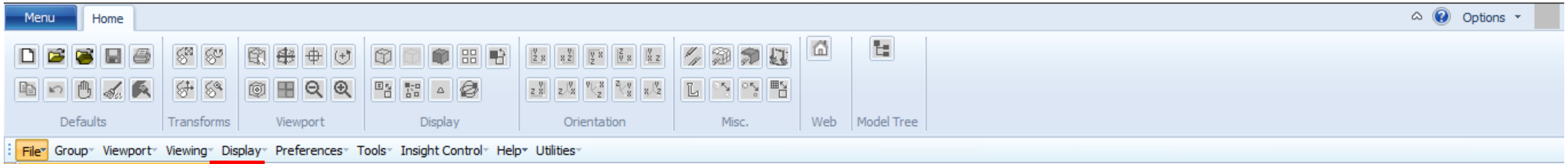
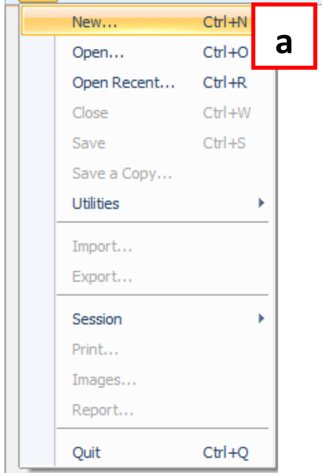


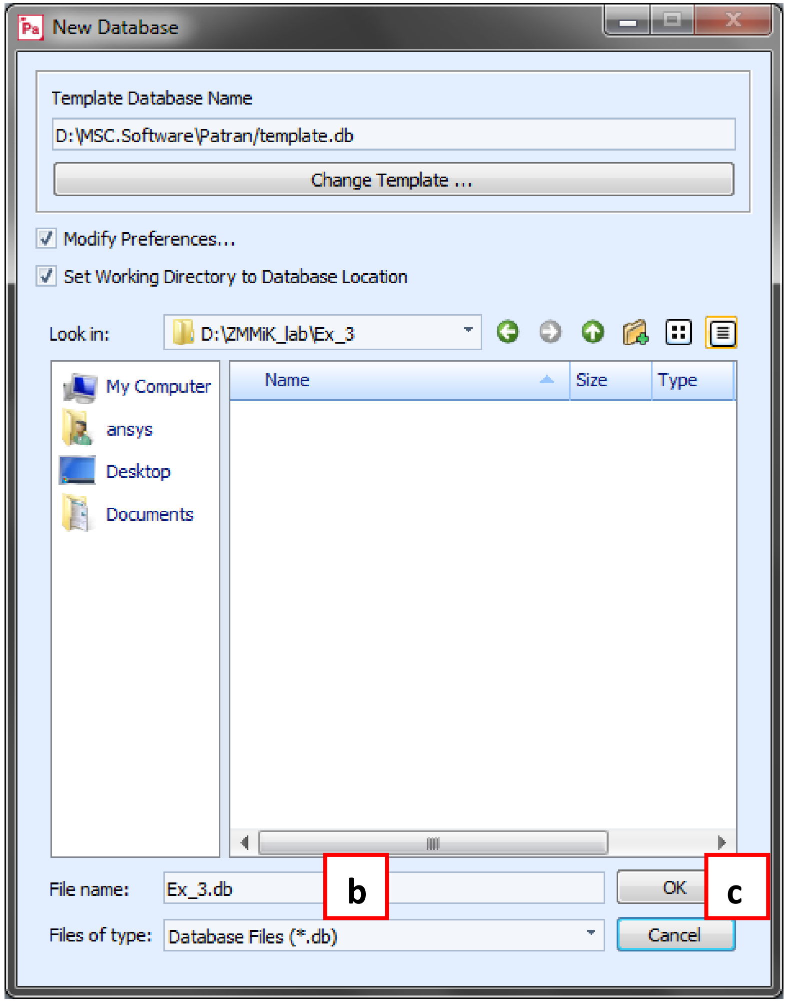
Fig. 2 Shallow, cylindrical roof under a point load

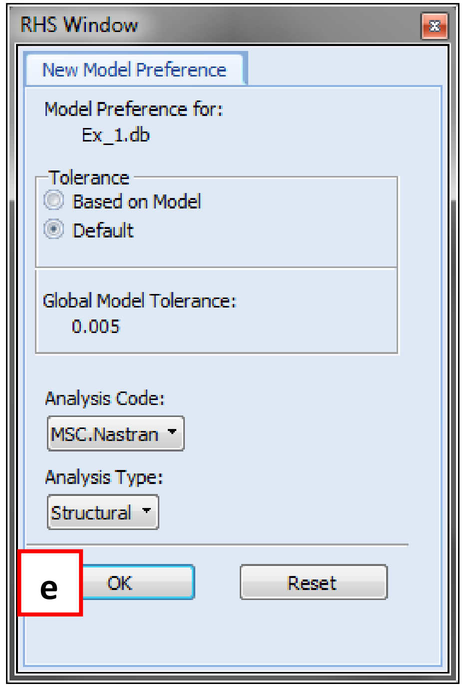
The goal of this exercise is to conduct the nonlinear analyses of two well known benchmark tests which exhibit snap-through behavior. In the first part *Lee's Frame Buckling Problem* is analyzed. In the second part – the behavior of a shallow, cylindrical roof under a point load is assessed.

Units: mm, N, MPa



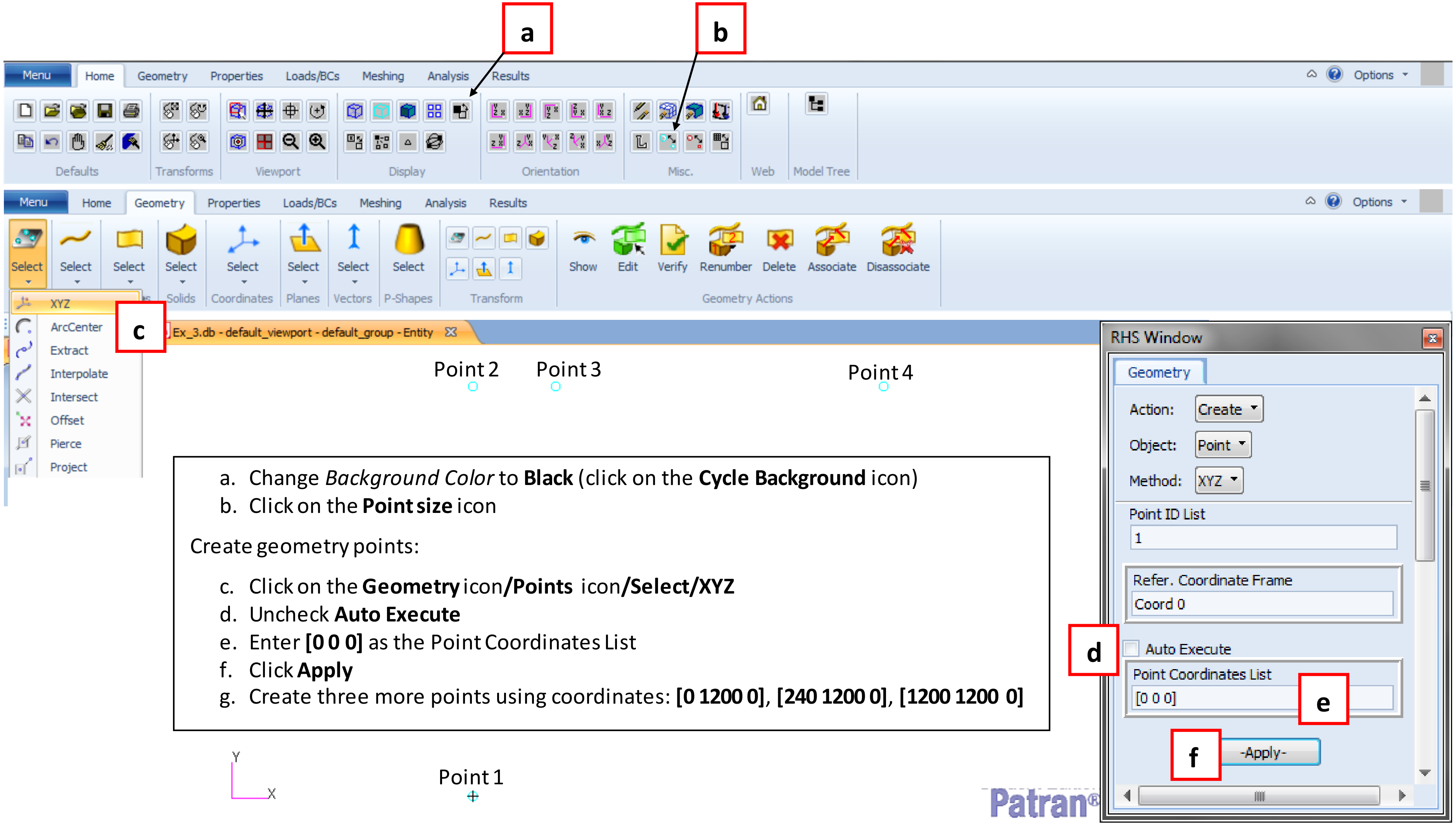






Create a new database:

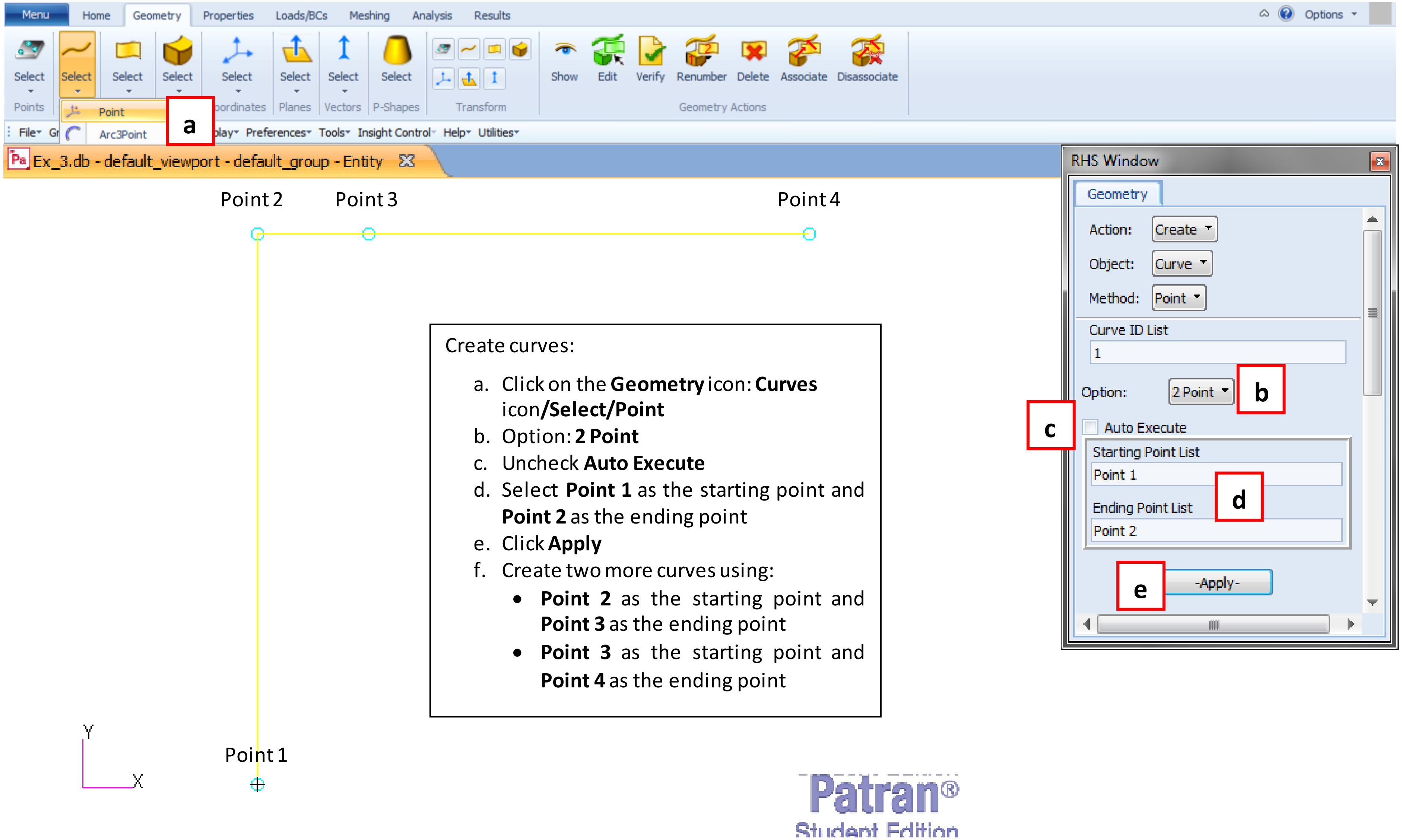
a. **File / New...**
b. Enter **Ex_3a.db** as the File name
c. Click **OK**
d. Select **Default**
e. Click **OK**



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

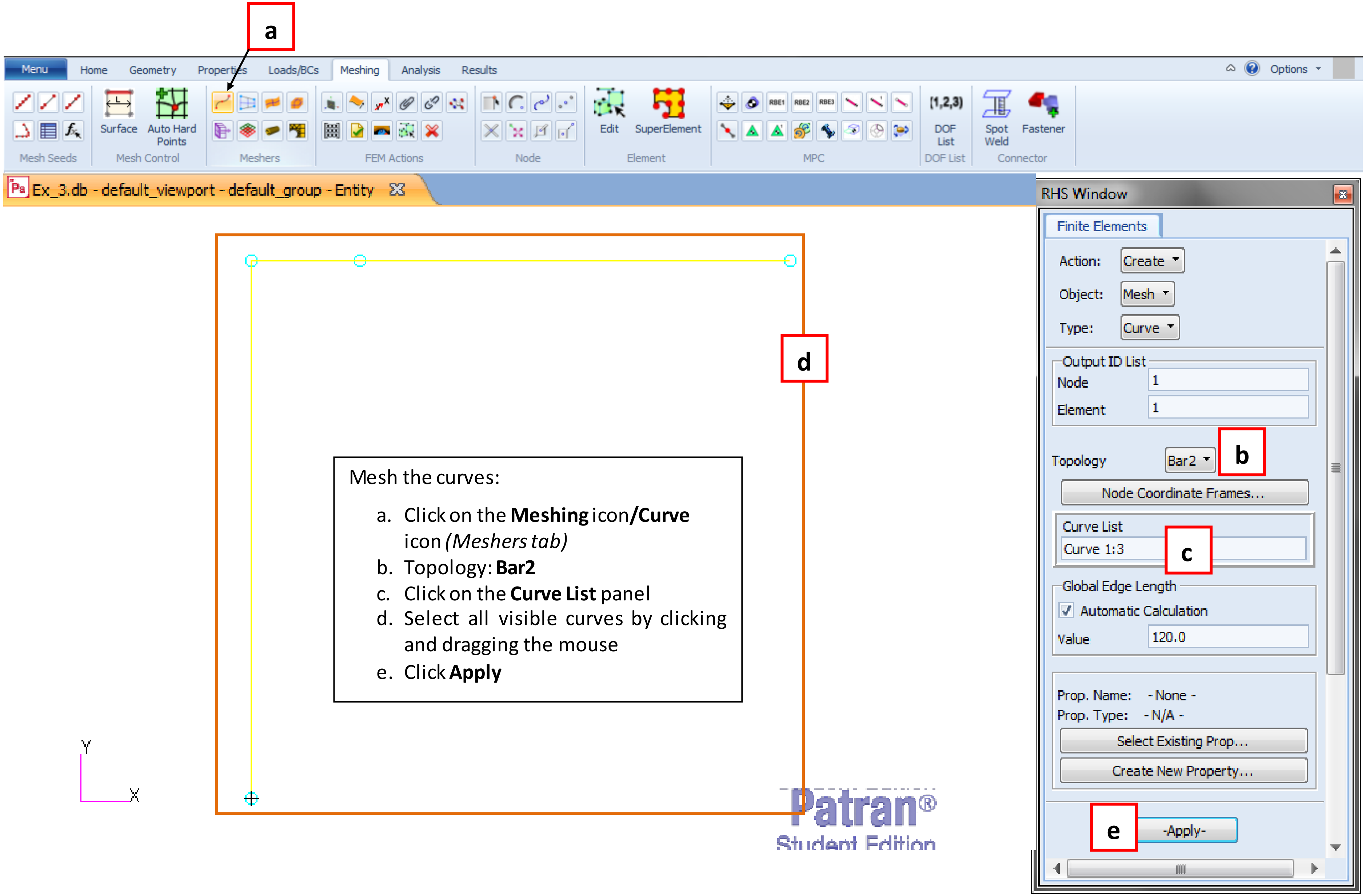
Create geometry points:

c. Click on the **Geometry** icon**/Points** icon**/Select/XYZ**
d. Uncheck **Auto Execute**
e. Enter **[0 0 0]** as the Point Coordinates List
f. Click **Apply**
g. Create three more points using coordinates: **[0 1200 0]**, **[240 1200 0]**, **[1200 1200 0]**



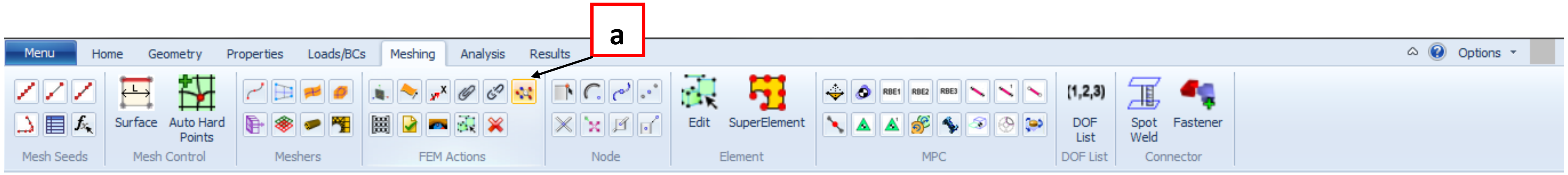
Create curves:

a. Click on the **Geometry** icon: **Curves** icon**/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**
f. Create two more curves using:
   - **Point 2** as the starting point and **Point 3** as the ending point
   - **Point 3** as the starting point and **Point 4** as the ending point



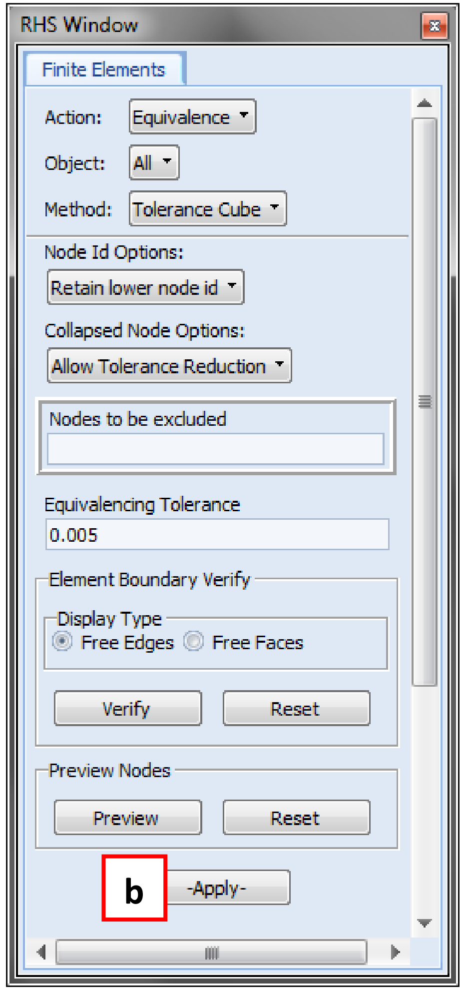
Mesh the curves:

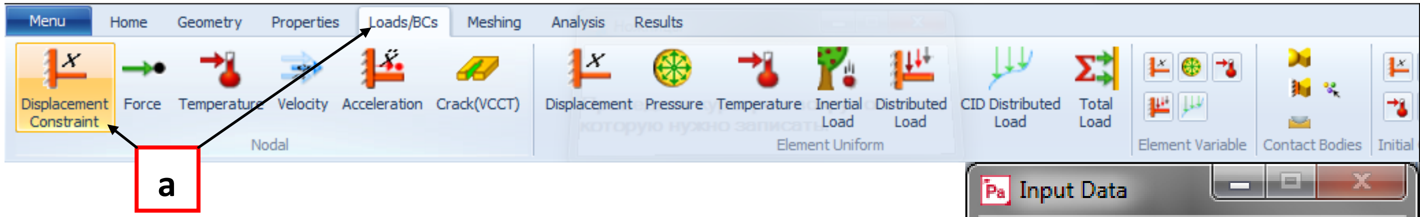
a. Click on the **Meshing** icon/**Curve** icon *(Meshers tab)*
b. Topology: **Bar2**
c. Click on the **Curve List** panel
d. Select all visible curves by clicking and dragging the mouse
e. Click **Apply**



Delete duplicate nodes:

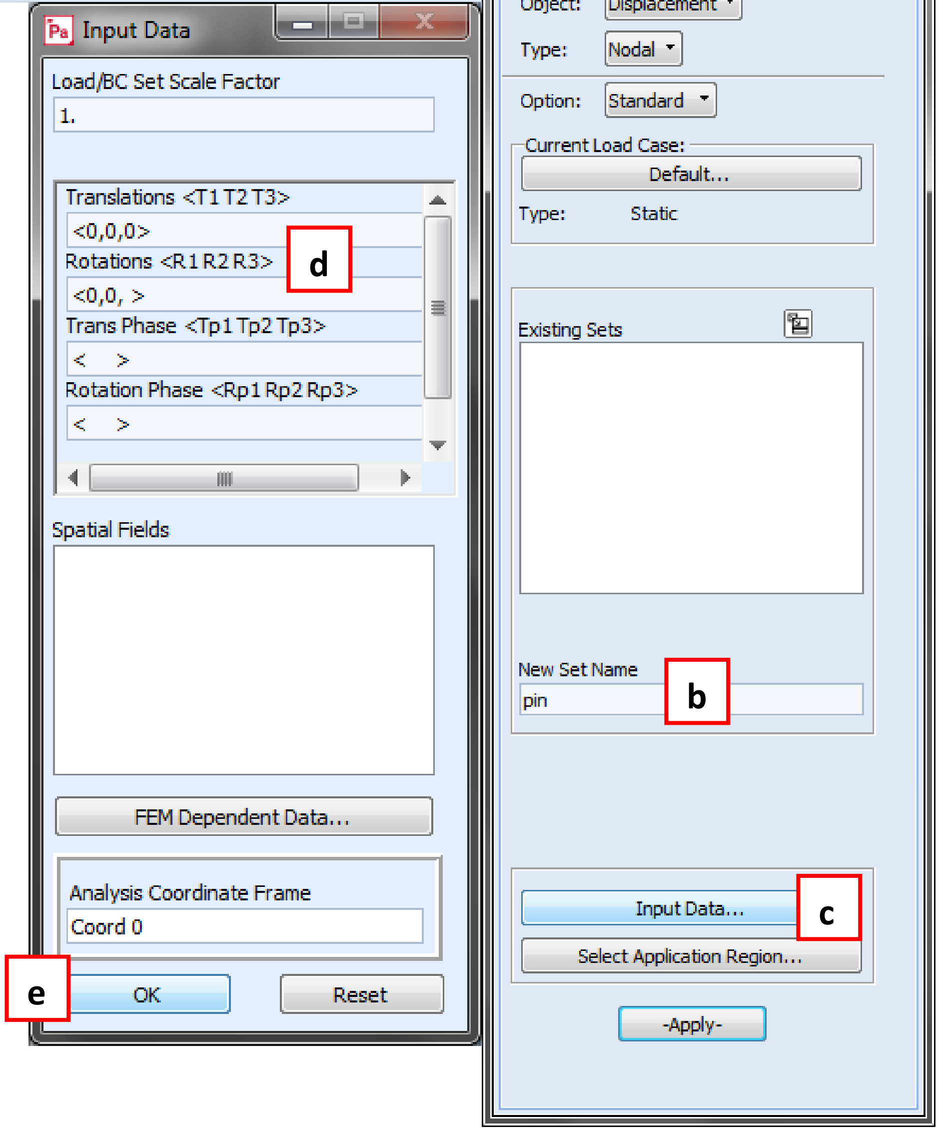
a. Meshing: **Equivalence** icon *(FEM Actions tab)*
b. Click **Apply**

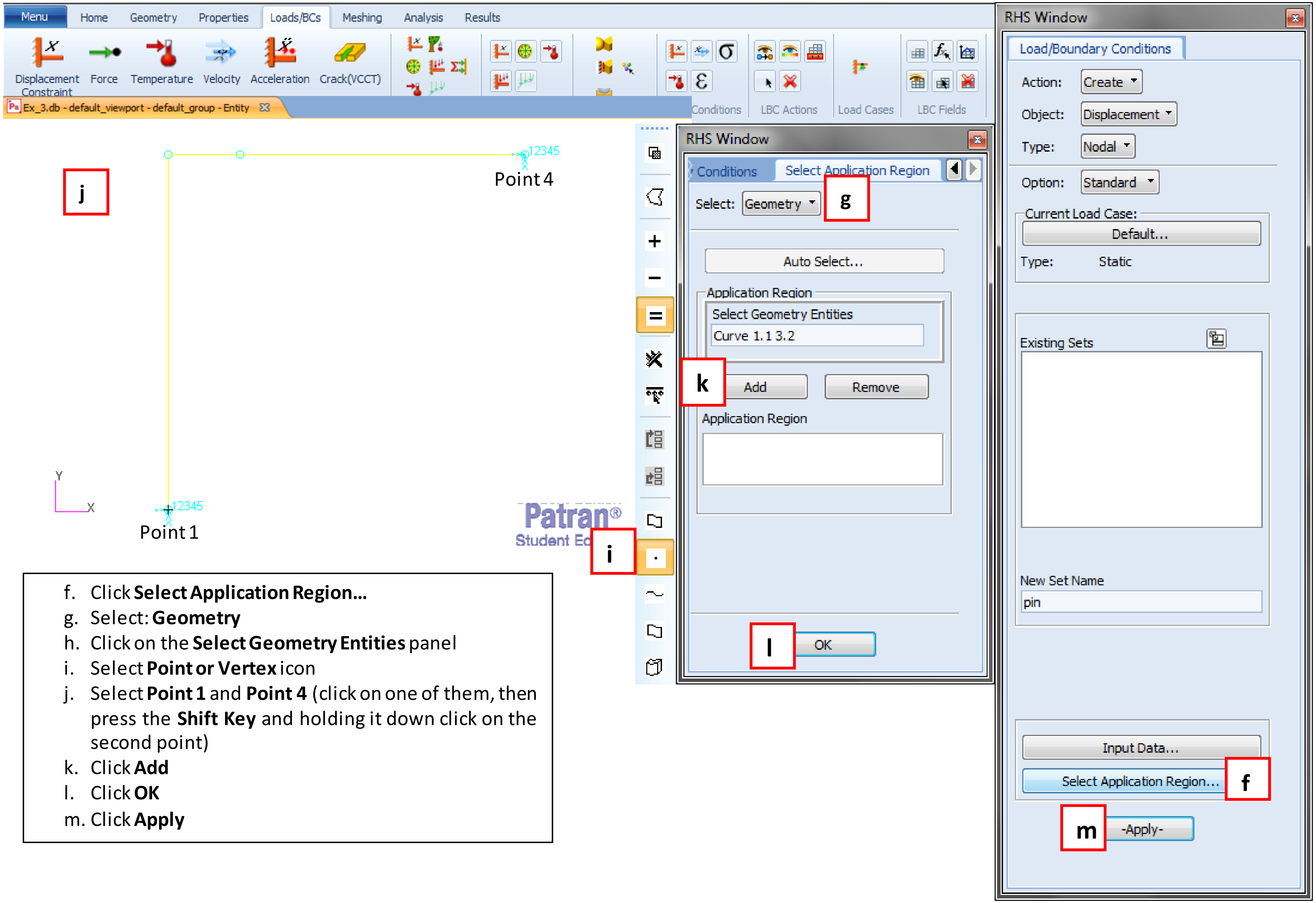




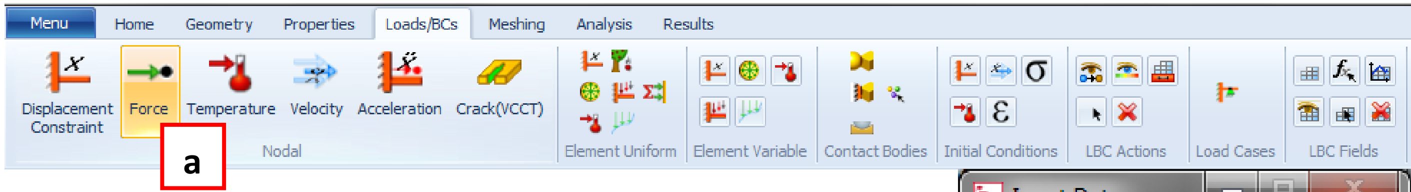
Apply the boundary conditions:

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



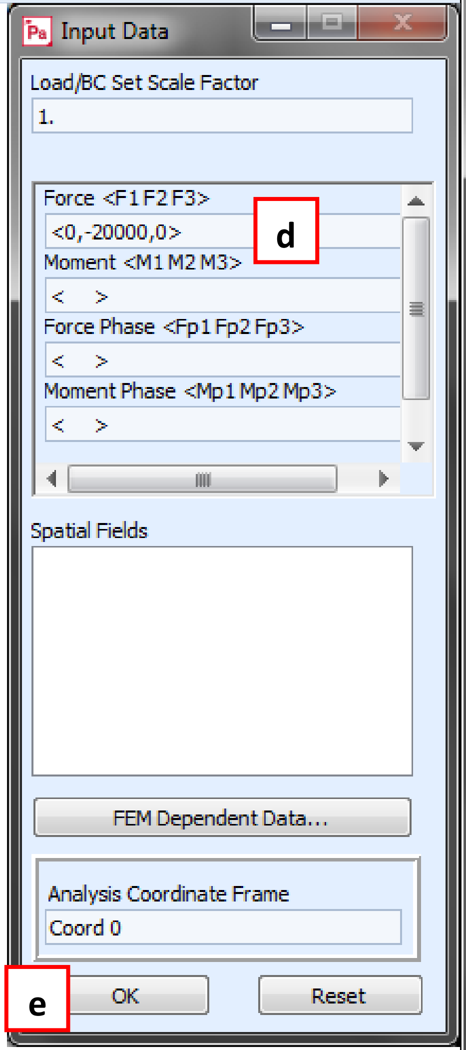


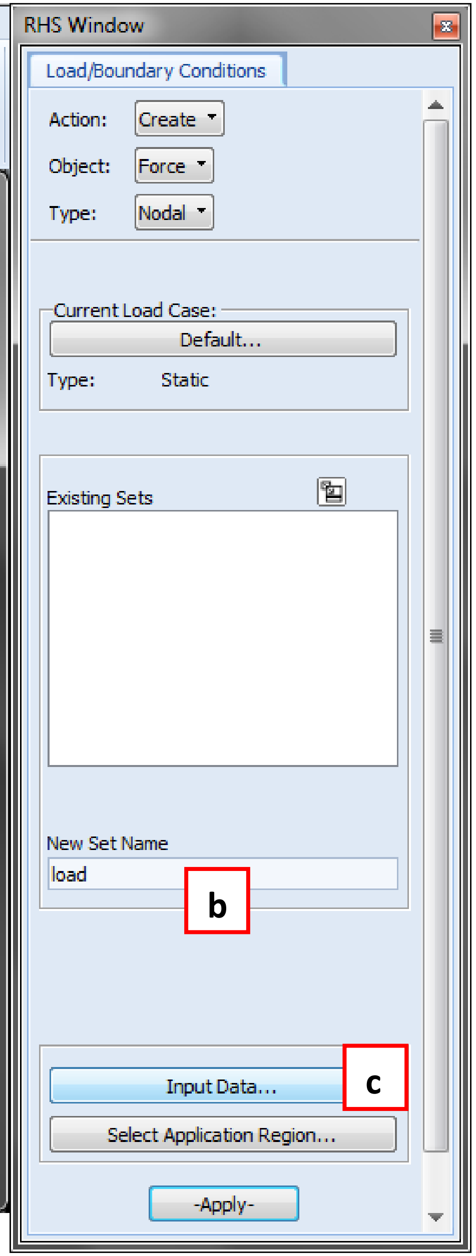
f. Click **Select Application Region...**
g. Select: **Geometry**
h. Click on the **Select Geometry Entities** panel
i. Select **Point or Vertex** icon
j. Select **Point 1** and **Point 4** (click on one of them, then press the **Shift Key** and holding it down click on the second point)
k. Click **Add**
l. Click **OK**
m. Click **Apply**

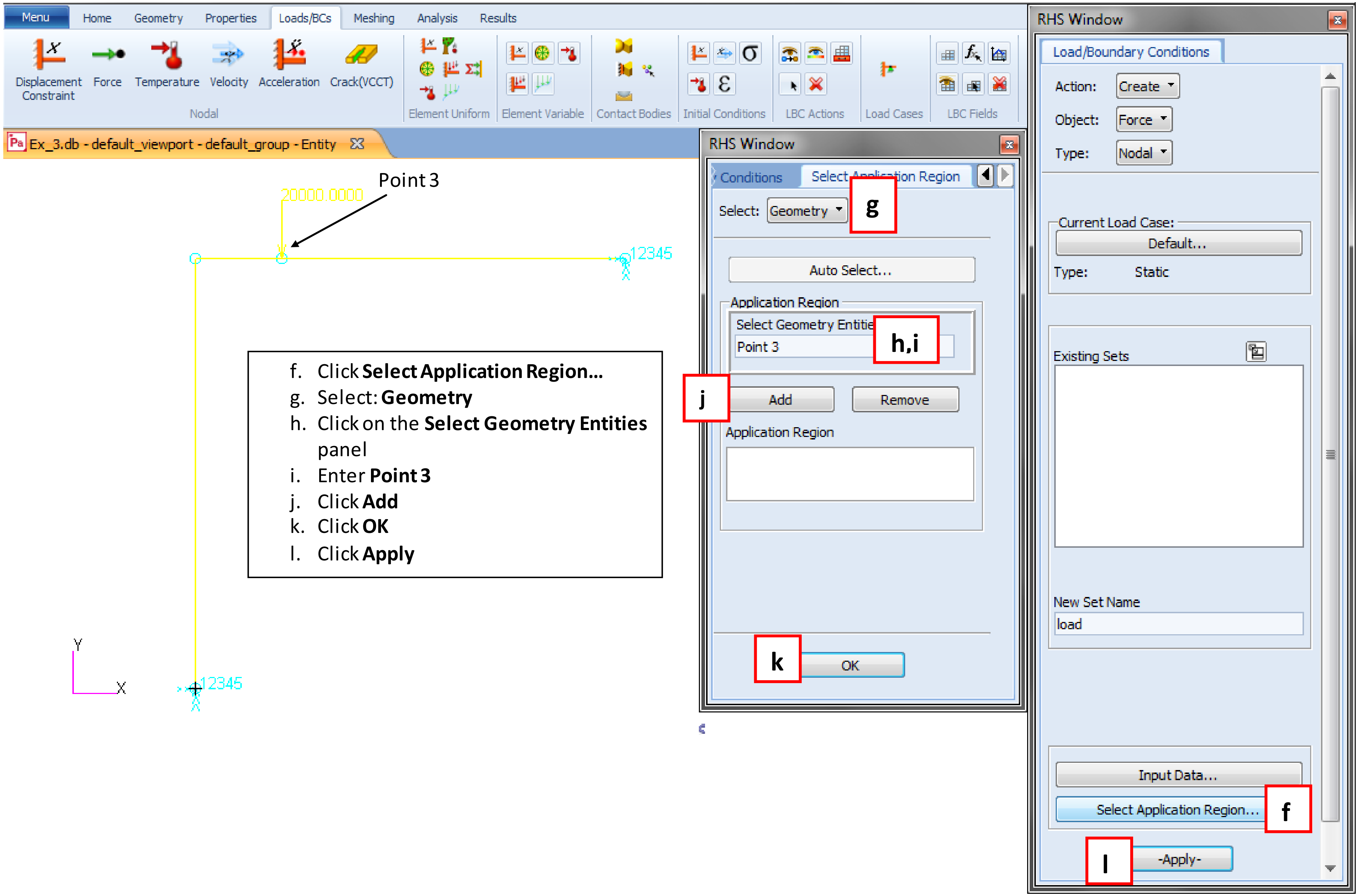


Apply the force:

a. Loads/BCs: **Force** icon
b. Enter **load** as the New Set Name
c. Click **Input Data...**
d. Enter **<0,-20000,0>** for the Force
e. Click **OK**

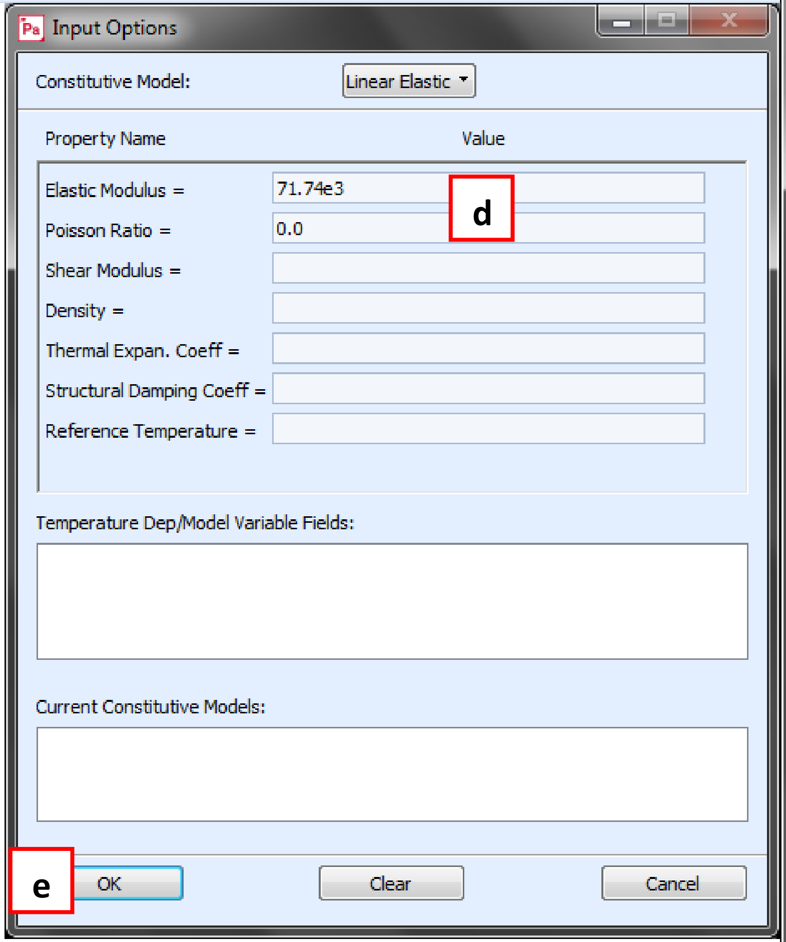


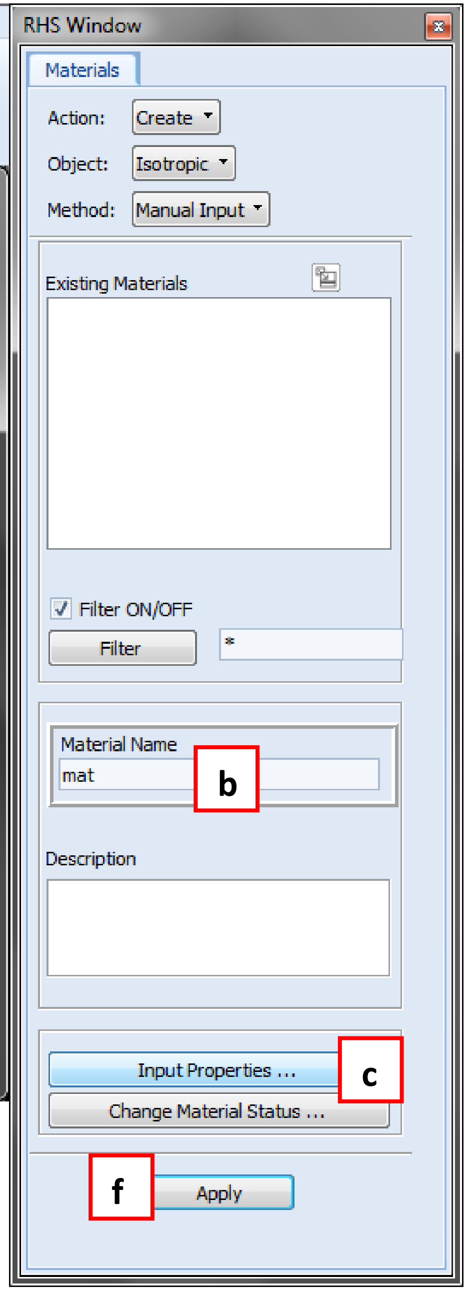


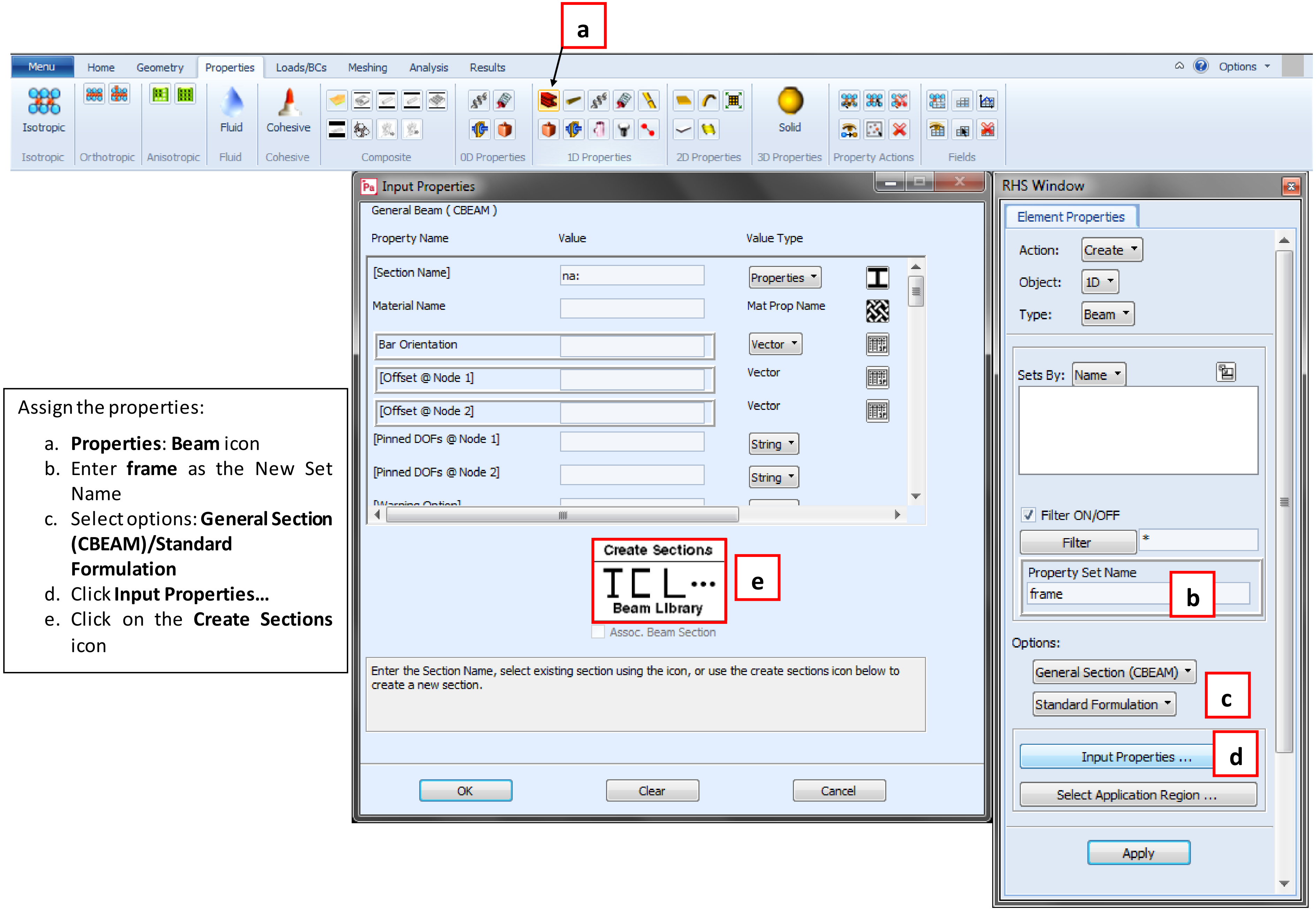

Point 3
20000.0000
12345
12345
Y
X
f. Click Select Application Region…
g. Select: Geometry
h. Click on the Select Geometry Entities panel
i. Enter Point 3
j. Click Add
k. Click OK
l. Click Apply
RHS Window
Conditions
Select Application Region
Select: Geometry
Auto Select…
Application Region
Select Geometry Entitie
Point 3
Add
Remove
Application Region
OK
RHS Window
Load/Boundary Conditions
Action: Create
Object: Force
Type: Nodal
Current Load Case:
Default…
Type: Static
Existing Sets
New Set Name
load
Input Data…
Select Application Region…
-Apply-
Ex_3.db - default_viewport - default_group - Entity
Menu
Home
Geometry
Properties
Loads/BCs
Meshing
Analysis
Results
Displacement Constraint
Force
Temperature
Velocity
Acceleration
Crack(VCCT)
Nodal
Element Uniform
Element Variable
Contact Bodies
Initial Conditions
LBC Actions
Load Cases
LBC Fields
g
h,i
j
k
f
l

Define a rial:

a. Click on the Click on the **Properties** icon/**Isotropic** icon
b. Enter **mat** as the Material Name
c. Click **Input Properties...**
d. Enter **71.74e3** as the Elastic Modulus and **0.0** as the Poisson Ratio
e. Click **OK**
f. Click **Apply**

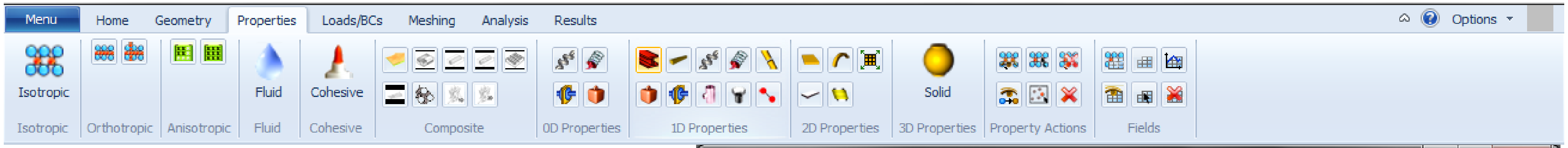






Assign the properties:

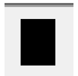
a. **Properties**: **Beam** icon
b. Enter **frame** as the New Set Name
c. Select options: **General Section (CBEAM)/Standard Formulation**
d. Click **Input Properties...**
e. Click on the **Create Sections** icon



f. Create Sections: **Create/Standard Shape/NASTRAN Standard**

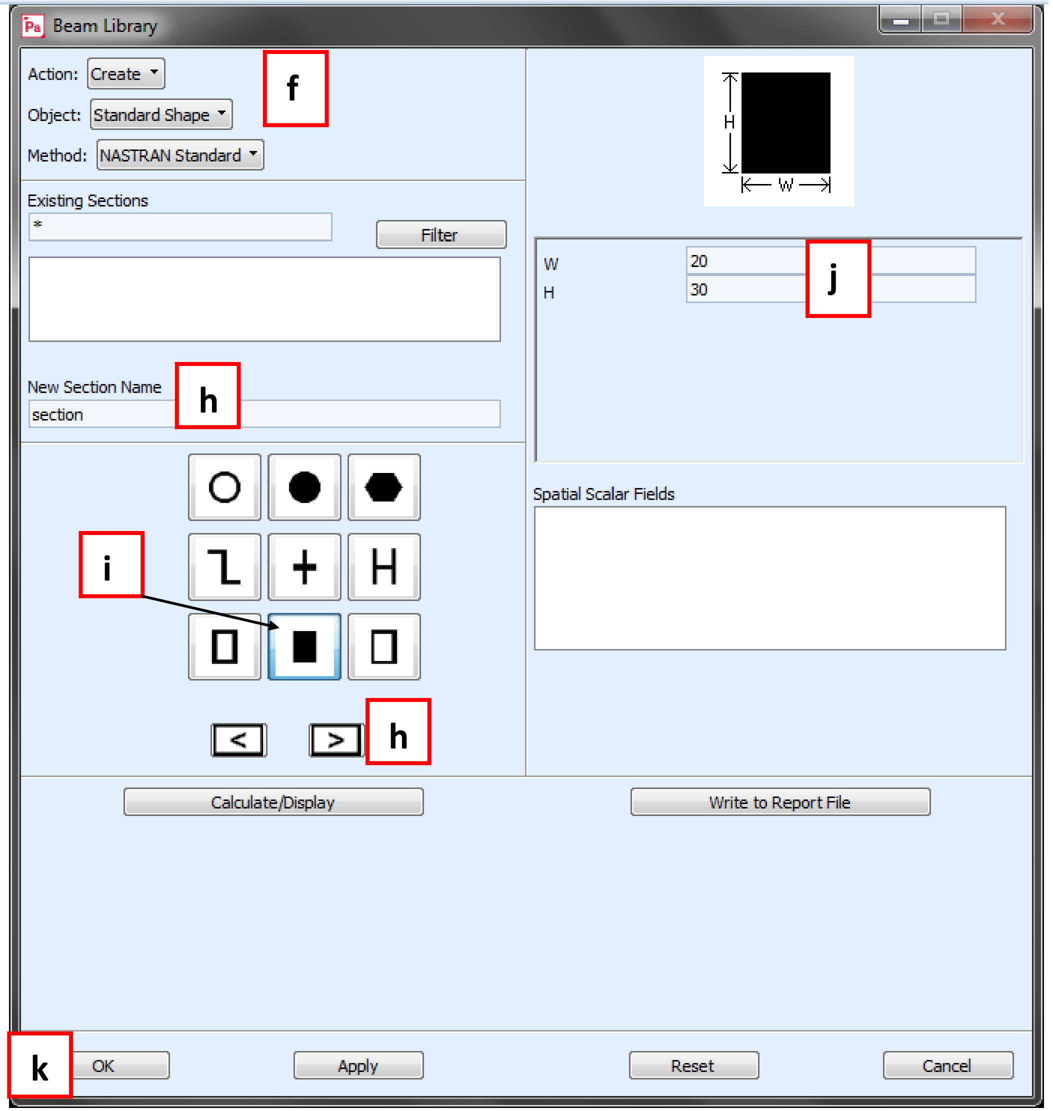
g. Enter **section** as the New Section Name

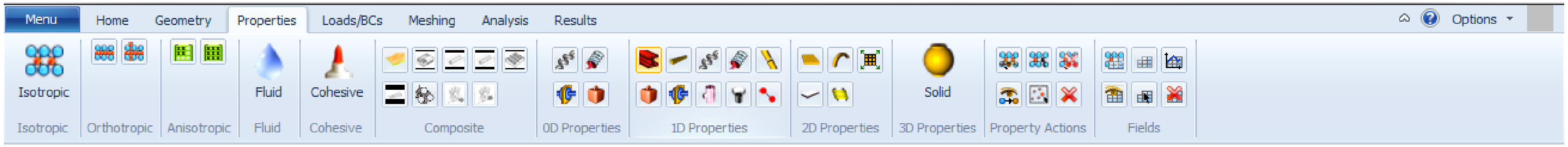
h. Click 

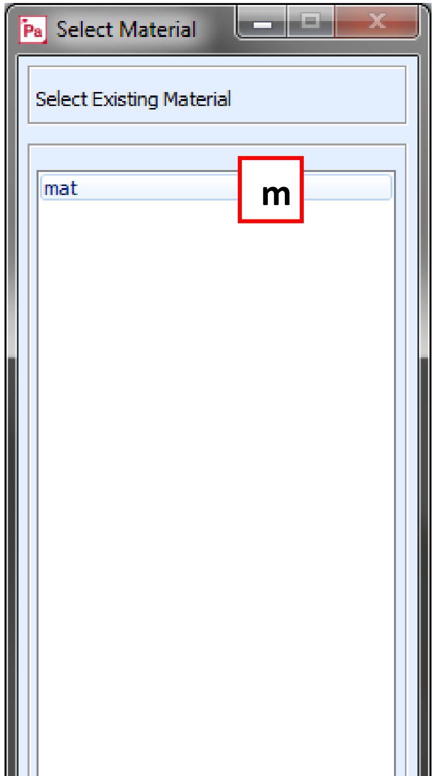
i. Click 

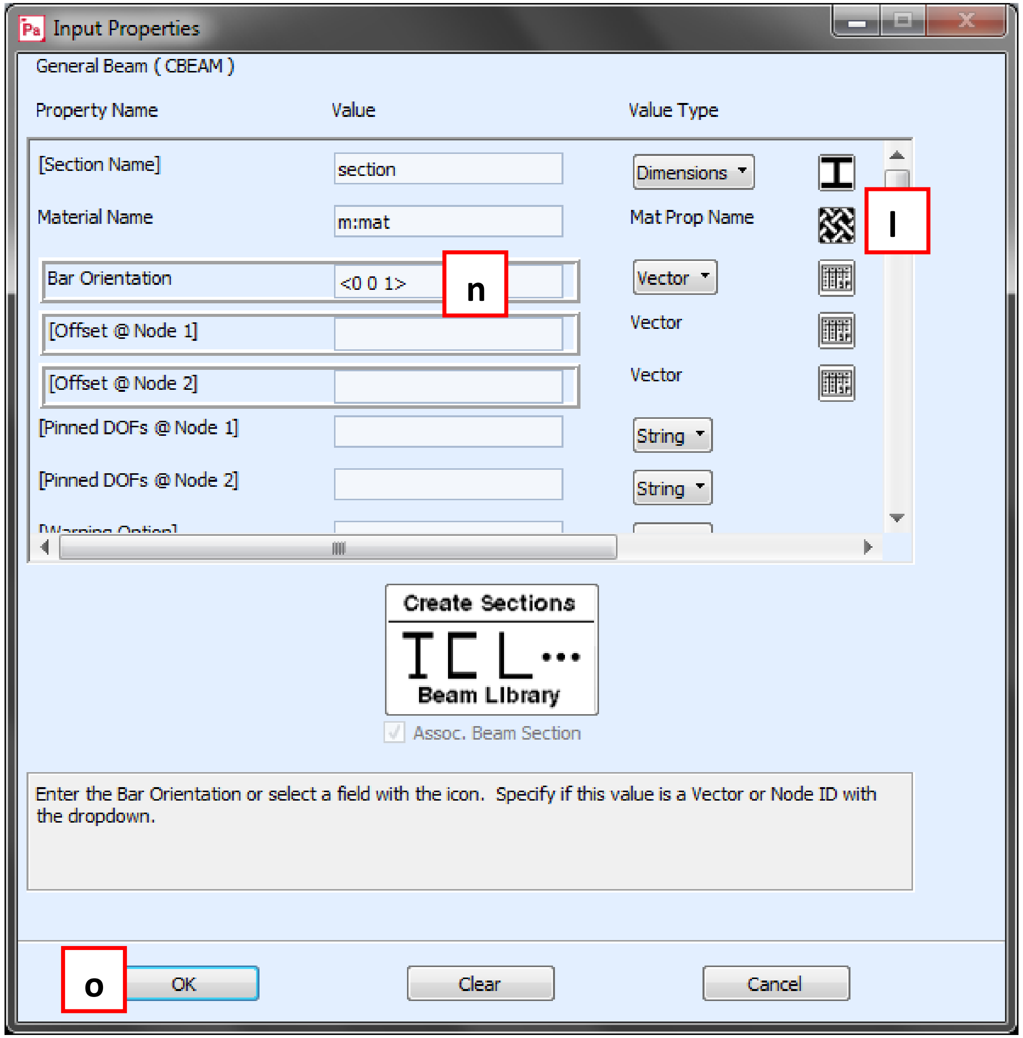
j. Enter **20** as the value of W and **30** as the value of H

k. Click **OK**

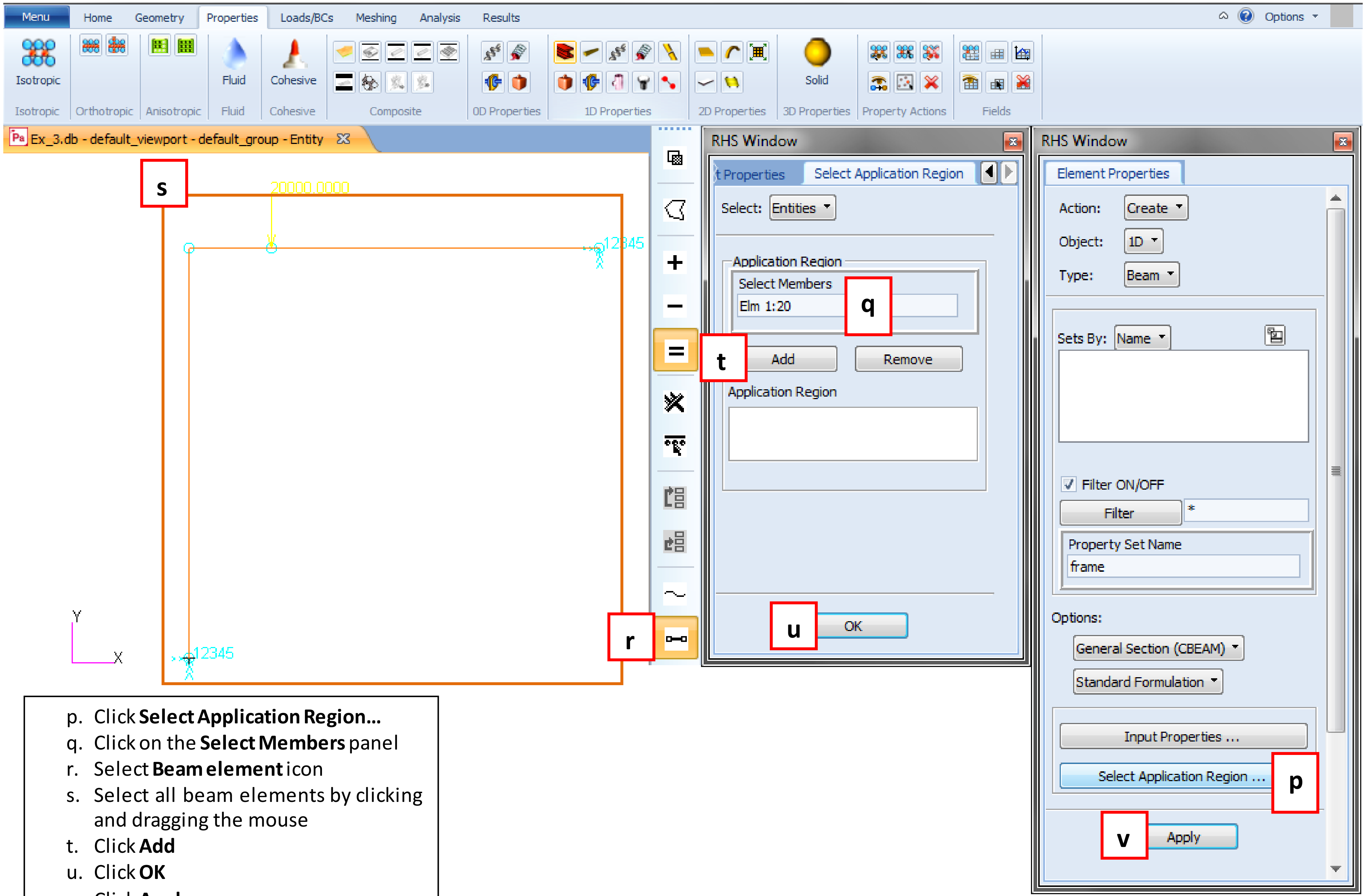




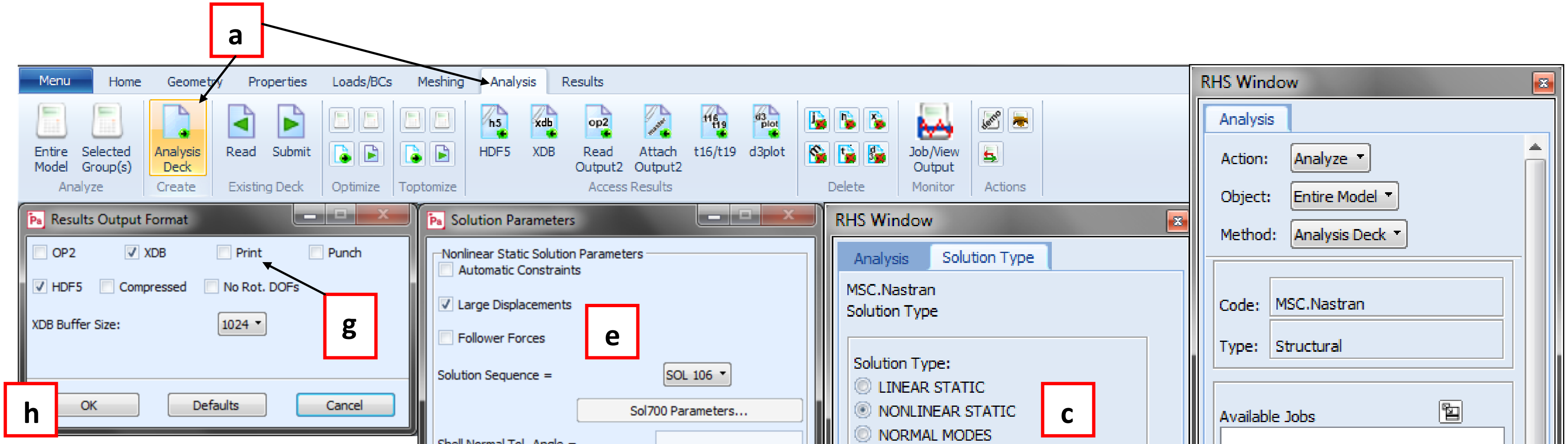




l. Click on the **Mat Prop Name** icon
m. Select **mat**
n. Enter **<0 0 1>** for the Bar Orientation
o. Click **OK**

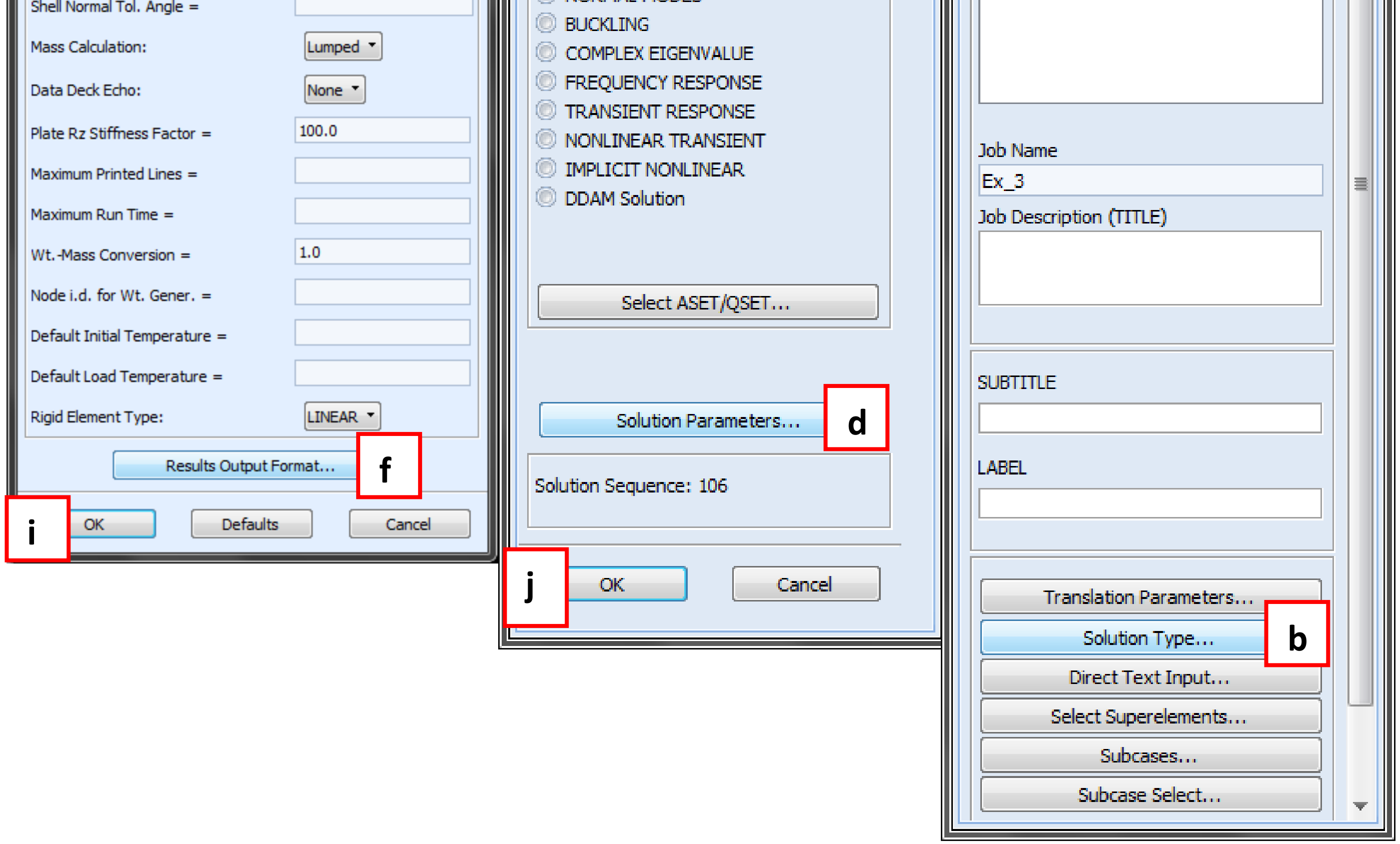


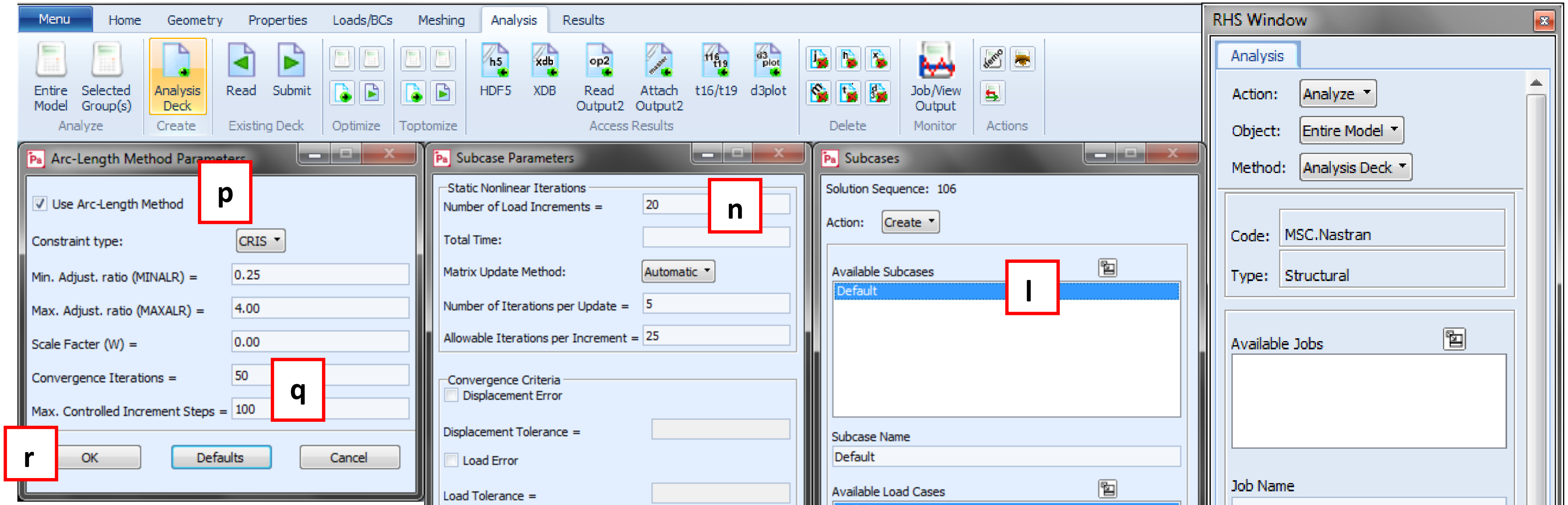
p. Click **Select Application Region...**
q. Click on the **Select Members** panel
r. Select **Beam element** icon
s. Select all beam elements by clicking and dragging the mouse
t. Click **Add**
u. Click **OK**
v. Click **Apply**



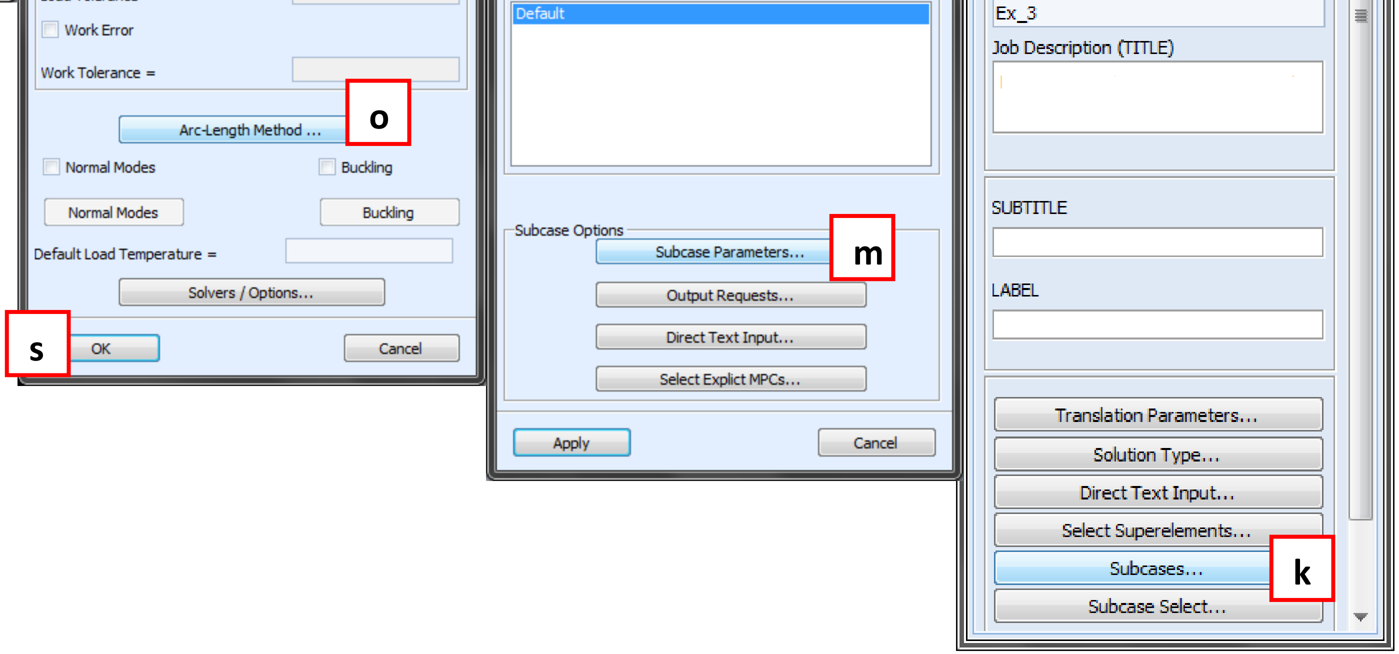
Run a nonlinear analysis:

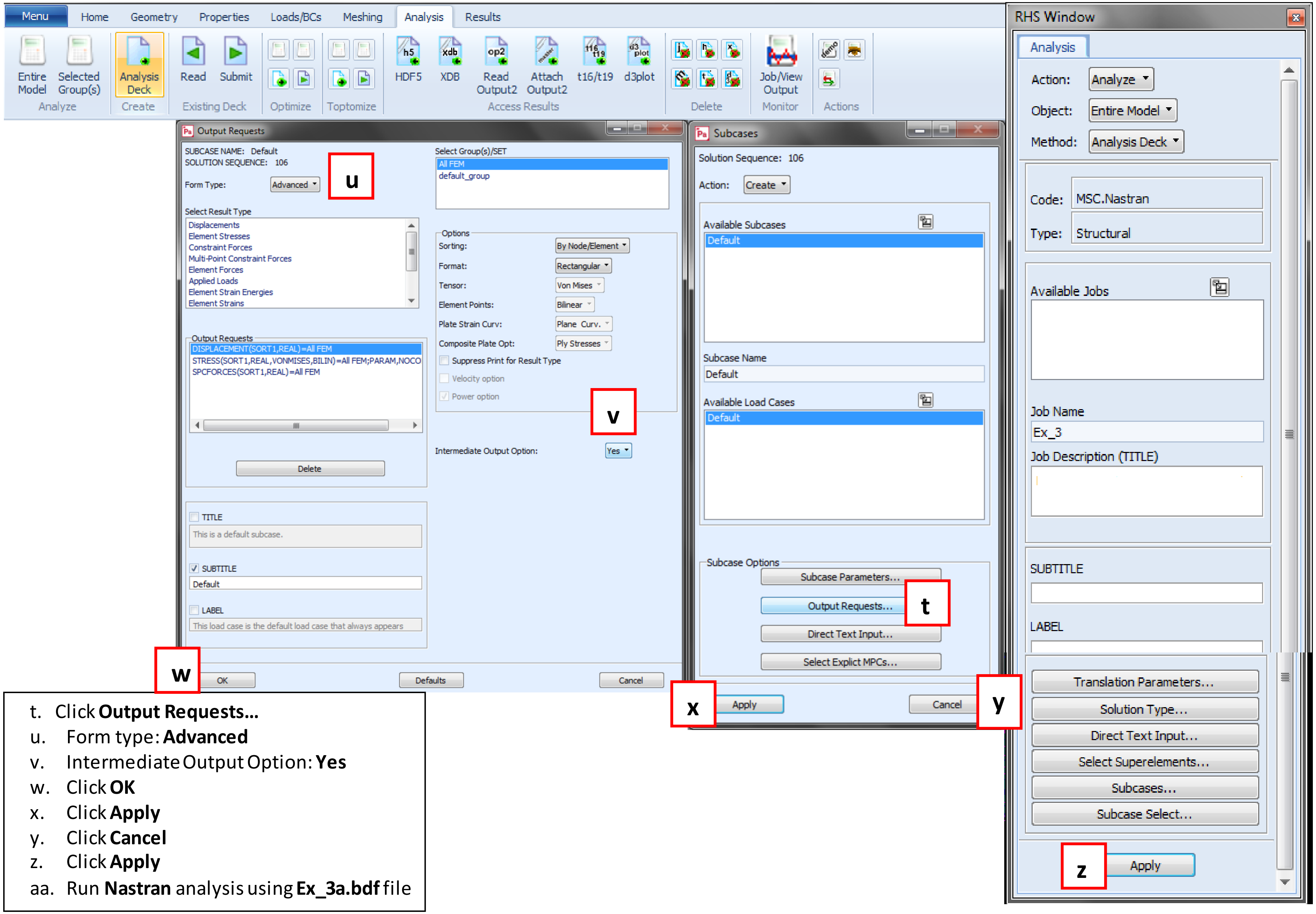
a. Click on the **Analysis/Analysis Deck** icon
b. Click **Solution Type...**
c. Select **NONLINEAR STATIC** as the Solution Type
d. Click **Solution Parameters...**
e. Uncheck **Follower Forces**
f. Click **Results Output Format...**
g. Uncheck **Print** and check **XDB**
h. Click **OK**
i. Click **OK**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



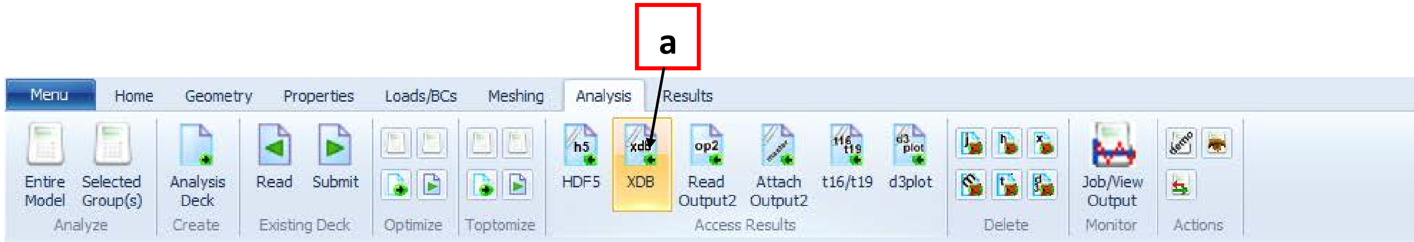


1. k. Click **Subcases...**
2. l. Select **Default**
3. m. Click **Subcase Parameters...**
4. n. Enter **20** as the Number of Load Increments
5. o. Click **Arc-Length Method**
6. p. Select **CRIS** and check **Use Arc-Length Method**
7. q. Enter **50** as the Convergence Iterations and **100** as the Max. Controlled Increment Steps
8. r. Click **OK**
9. s. Click **OK**



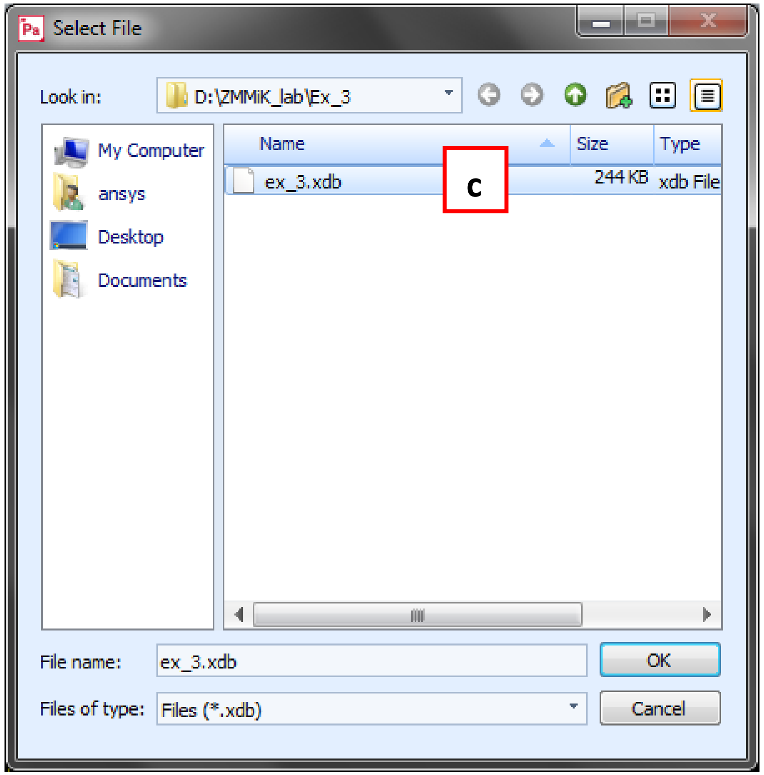


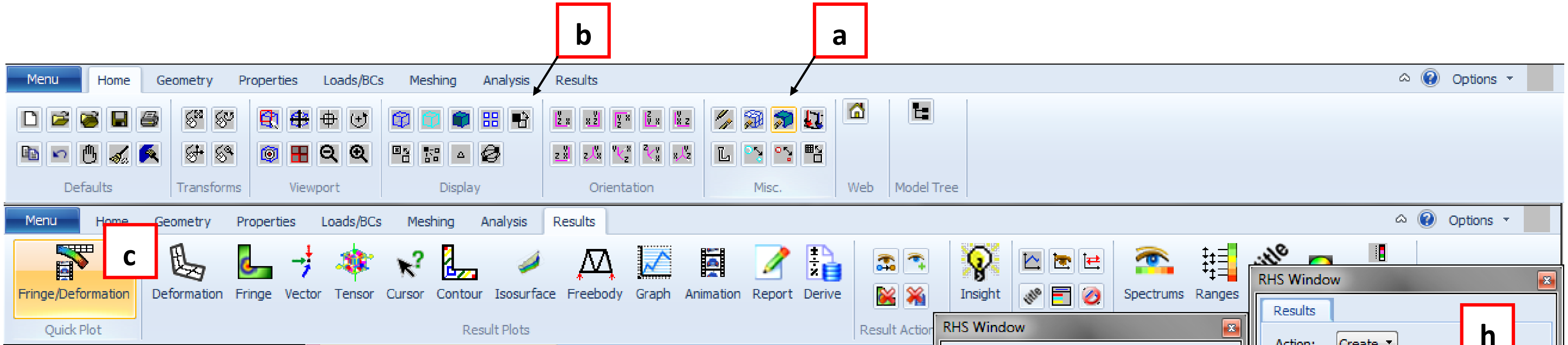
t. Click **Output Requests…**
u. Form type: **Advanced**
v. Intermediate Output Option: **Yes**
w. Click **OK**
x. Click **Apply**
y. Click **Cancel**
z. Click **Apply**
aa. Run **Nastran** analysis using **Ex_3a.bdf** file



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

a. Analysis: **XDB** icon
b. Click **Select Results File…**
c. Select **Ex_3a.xdb** file and click **OK**
d. Click **Apply**



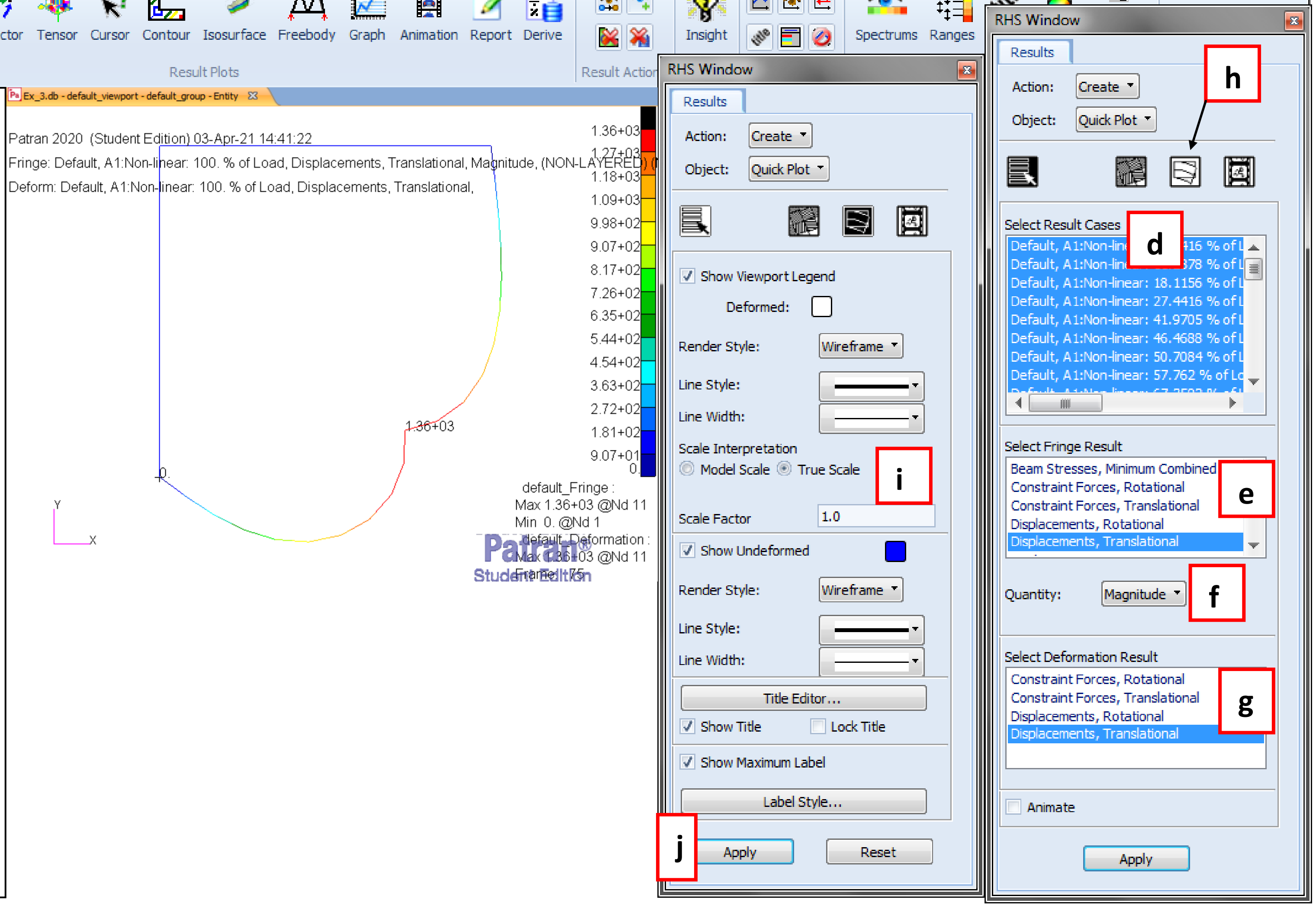


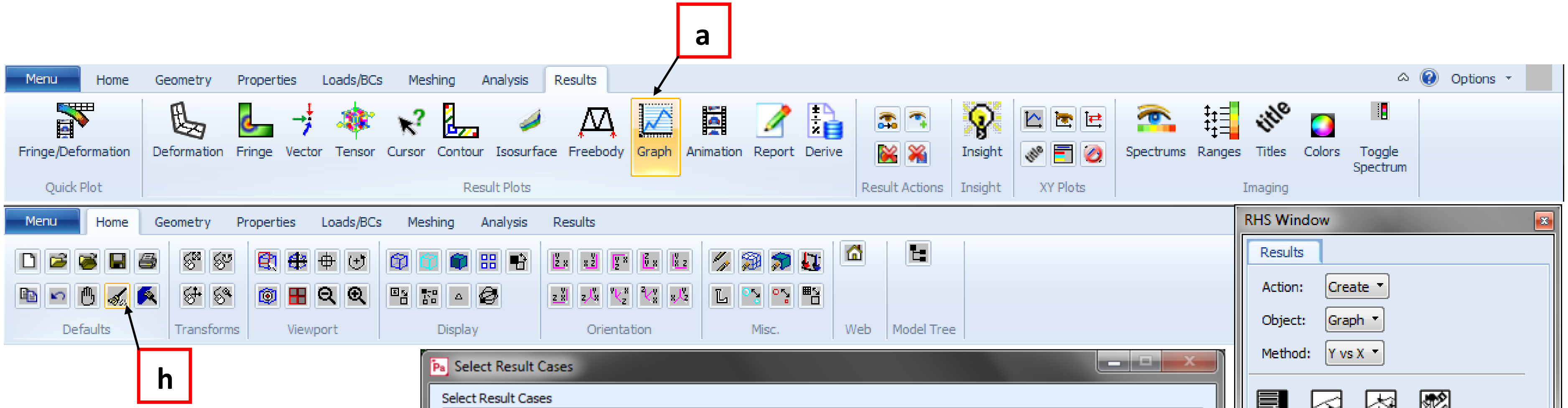
- a. Click on the **Plot/Erase Geometry** icon

Post-process the results:

- b. Change background color to white
- c. Click on the **Results** icon: **Fringe/Deformation** *(Quick Plot)*
- d. Select **all** steps
- e. Select Fringe Result: **Displacements, Translational**
- f. Quantity: **Magnitude**
- g. Select Deformation Result: **Displacements, Translational**
- h. Click on the **Deform Attributes** icon
- i. Select **True Scale** with the Scale Factor equals to **1.0**
- **j.** Click **Apply**

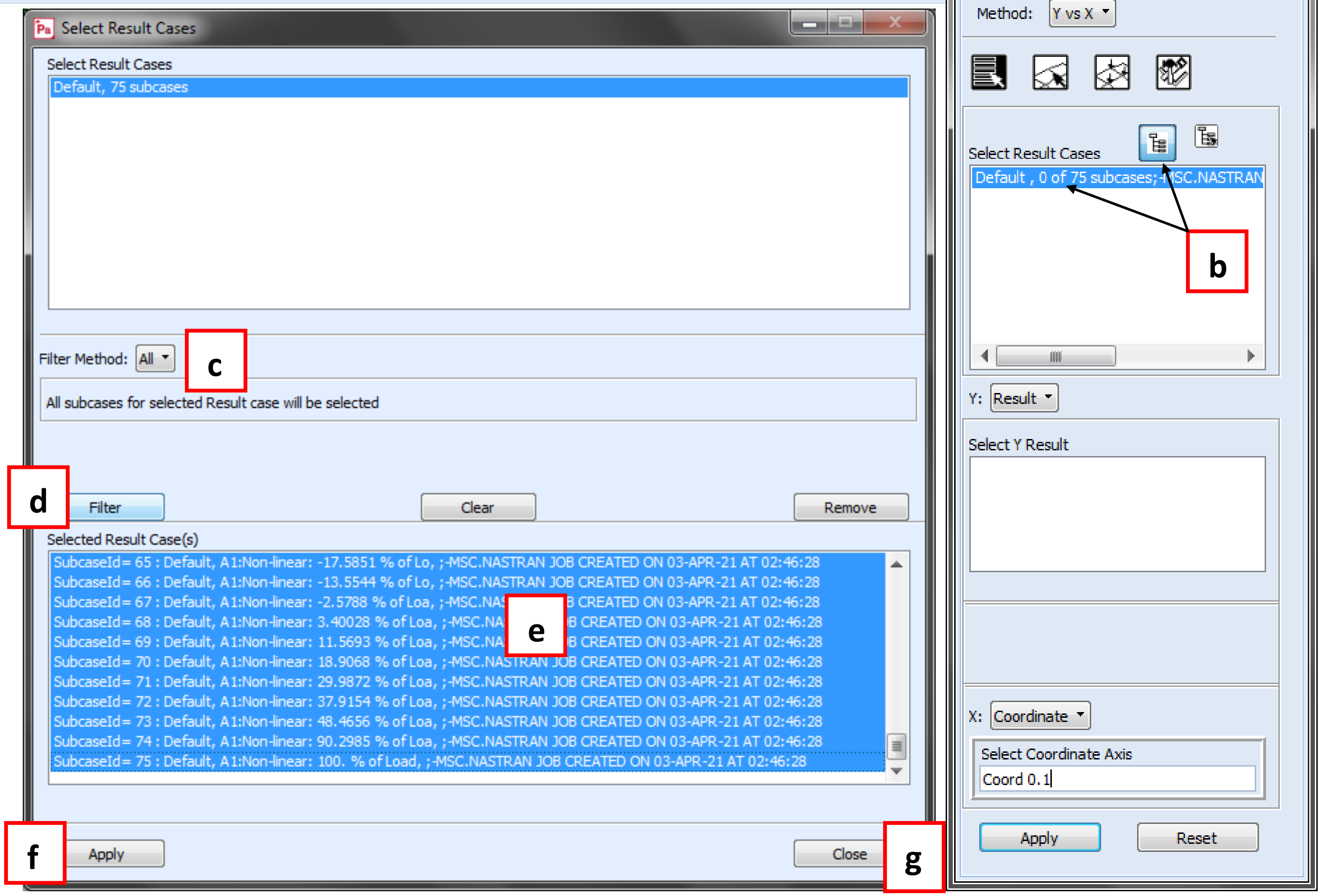
Remark: To capture the plot use **File / Images…**





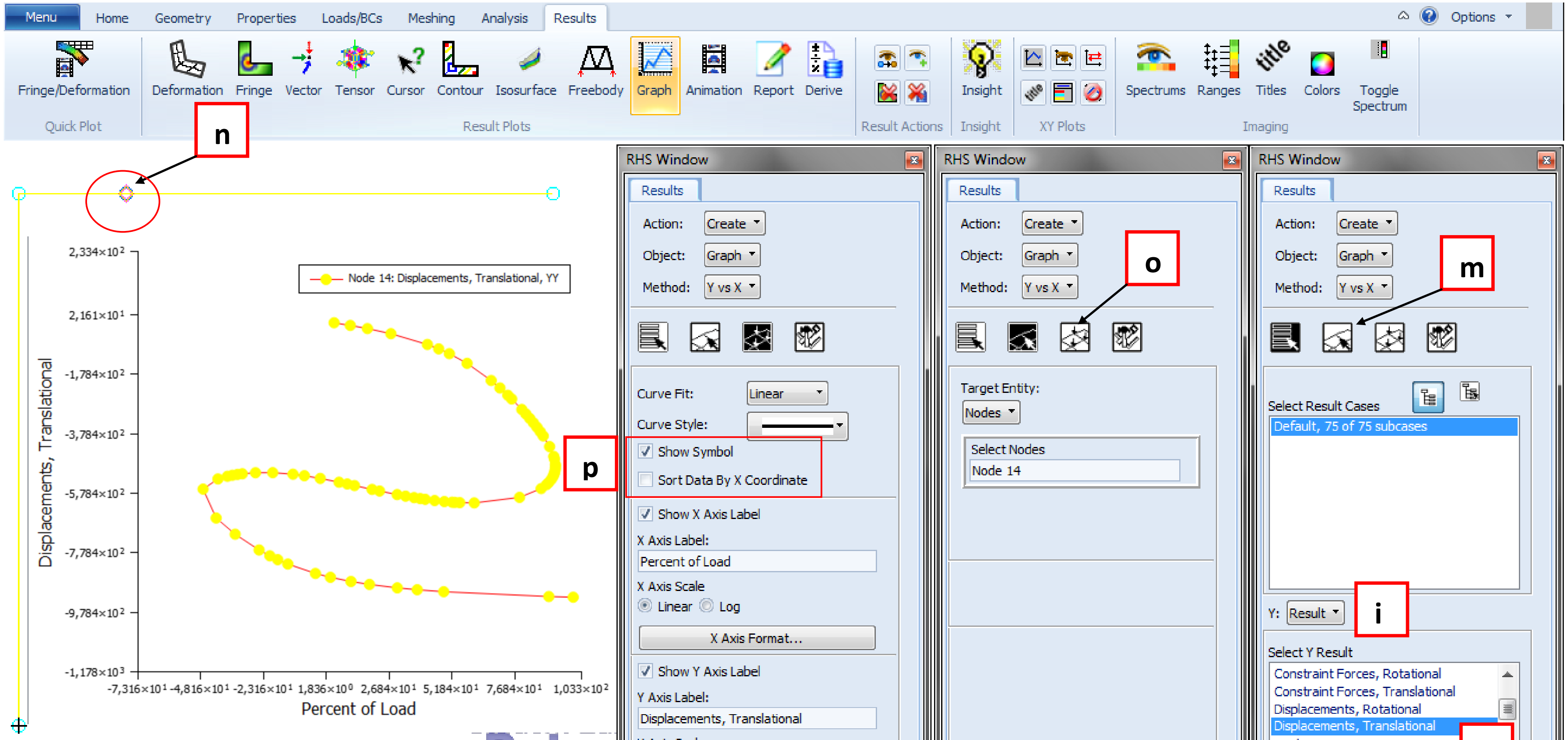
Create a graph (an equilibrium path):

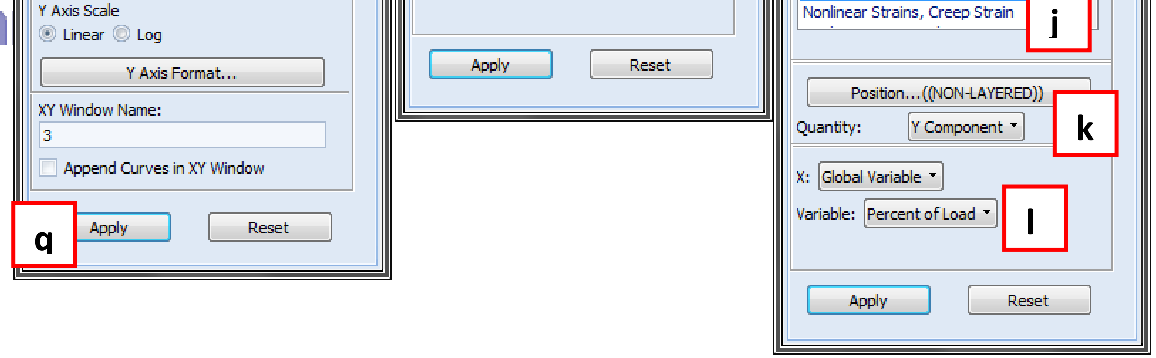
a. Results: **Graph** icon
b. Select and click on the **SC1: 0 of 75 subcases;**
c. Filter Method: **All**
d. Click **Filter**
e. Select **all** steps
f. Click **Apply**
g. Click **Close**
h. Click **Reset Graphics**



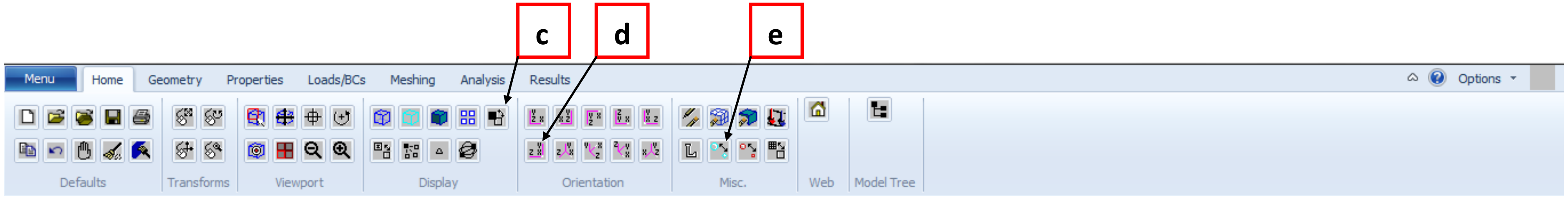
HUMAN CAPITAL
NATIONAL COHESION STRATEGY

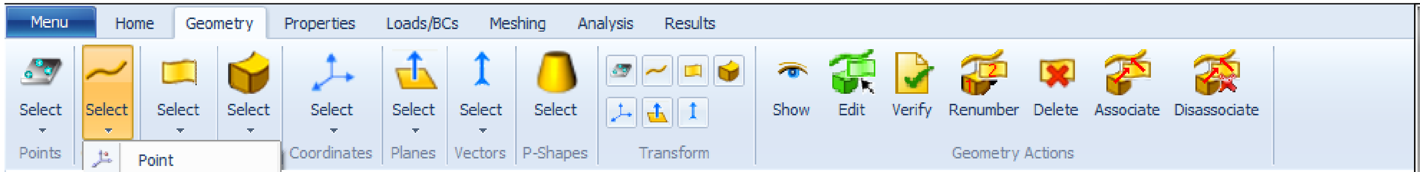
EUROPEAN UNION
EUROPEAN
SOCIAL FUND

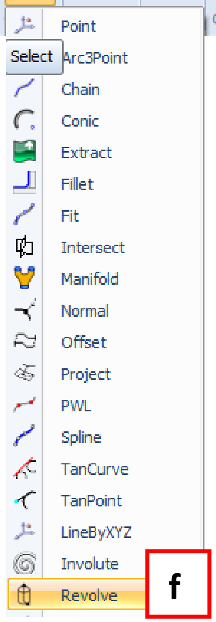




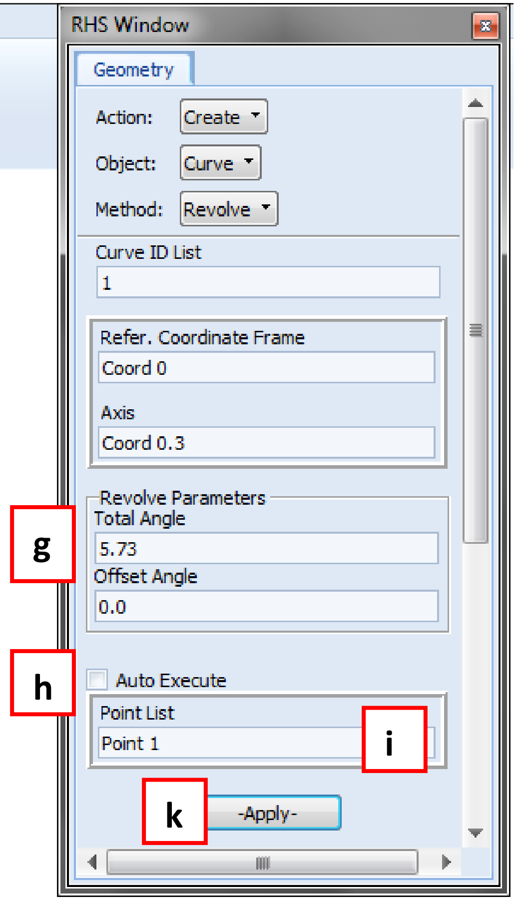
i. Y: **Result**
j. Select **Displacements, Translational** as the Y Result
k. Quantity: **Y Component**
l. X: **Global Variable**; Variable: **Percent of Load**
m. Click on the **Target Entities** icon
n. Select the node to which the force was applied
o. Click on the **Display Attributes** icon
p. Check **Show Symbol** and Uncheck **Sort Data By X Coordinate**
q. Click **Apply**







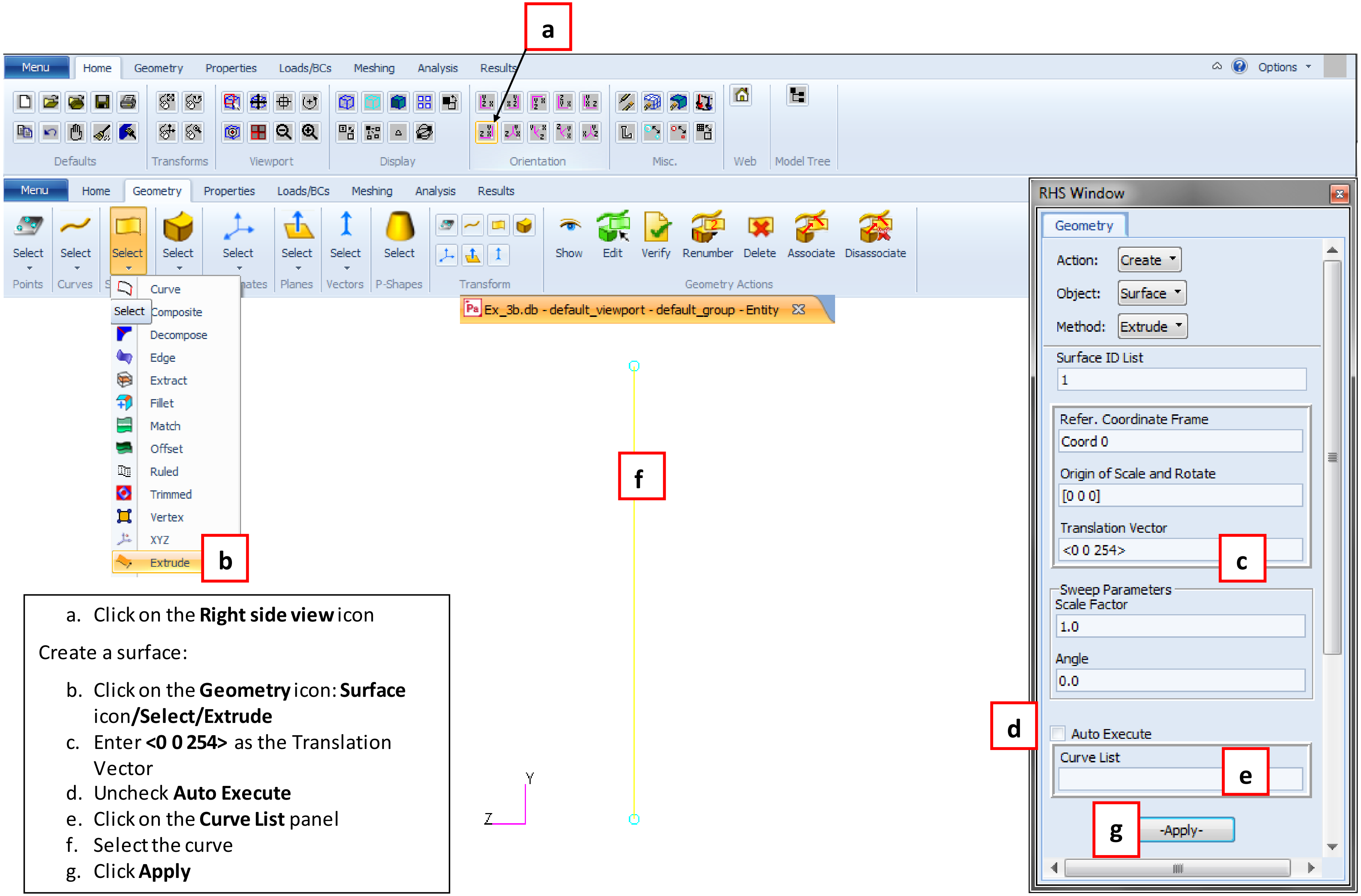
PART 2: Snap-through of a shallow, cylindrical roof under a point load



a. Save and close the previous database
b. Create a new database (**Ex_3b.db**)
c. Change *Viewport Color* to **Black** (click on the **Cycle Background** icon)
d. Click on the **Point size** icon
e. Create a point using coordinates: **[2540 0 0]**

Create a curve:

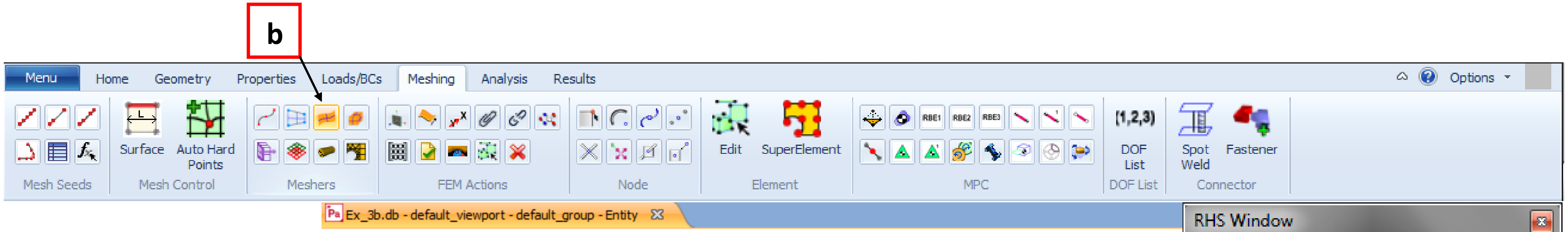
f. Click on the **Geometry** icon: **Curves** icon**/Select/Revolve**
g. Enter **5.73** as the Total Angle
h. Uncheck **Auto Execute**
i. Click on the **Point List** panel
j. Select the point created in step **e**
k. Click **Apply**



a. Click on the **Right side view** icon

Create a surface:

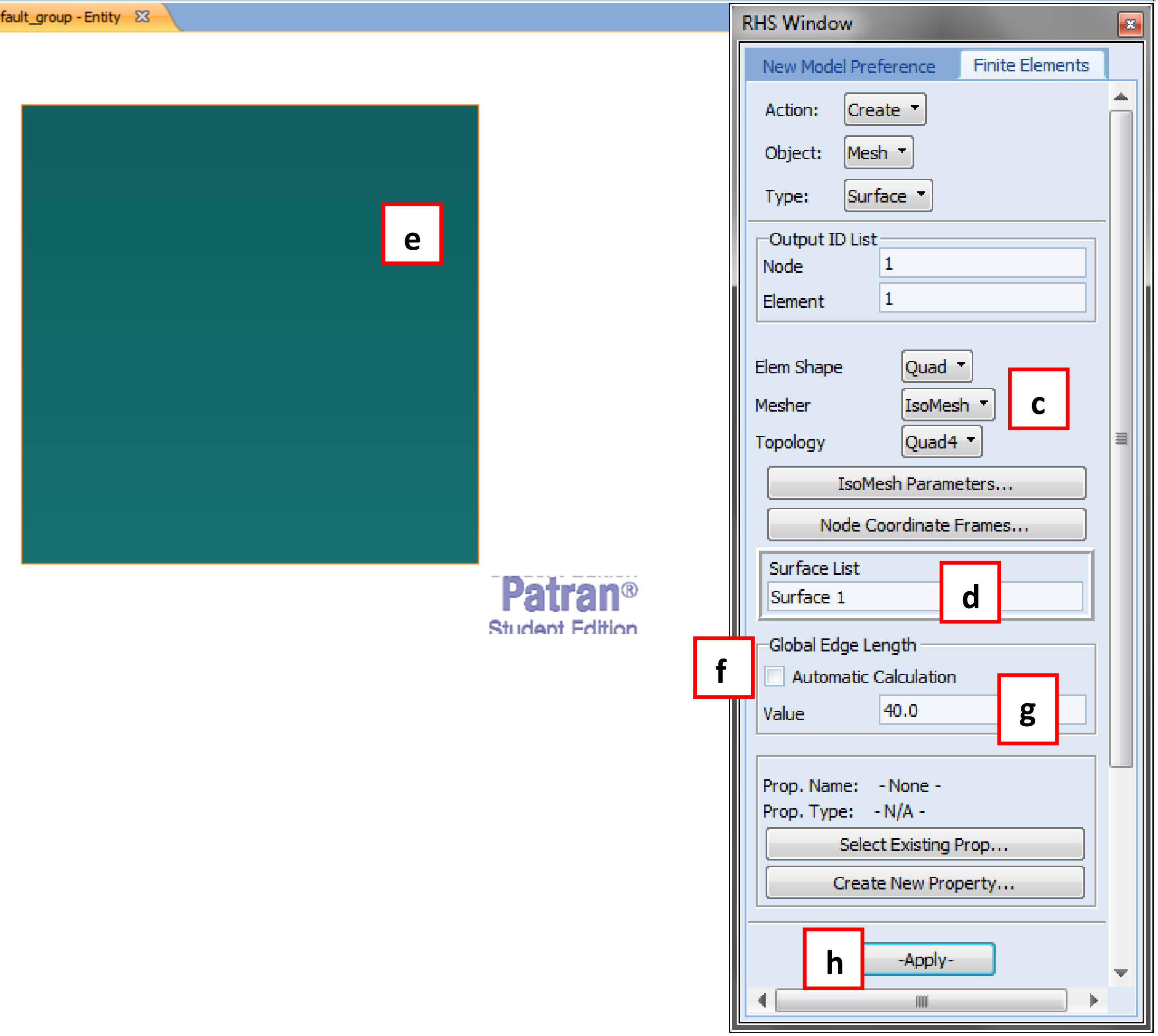
b. Click on the **Geometry** icon: **Surface** icon**/Select/Extrude**
c. Enter **<0 0 254>** as the Translation Vector
d. Uncheck **Auto Execute**
e. Click on the **Curve List** panel
f. Select the curve
g. Click **Apply**



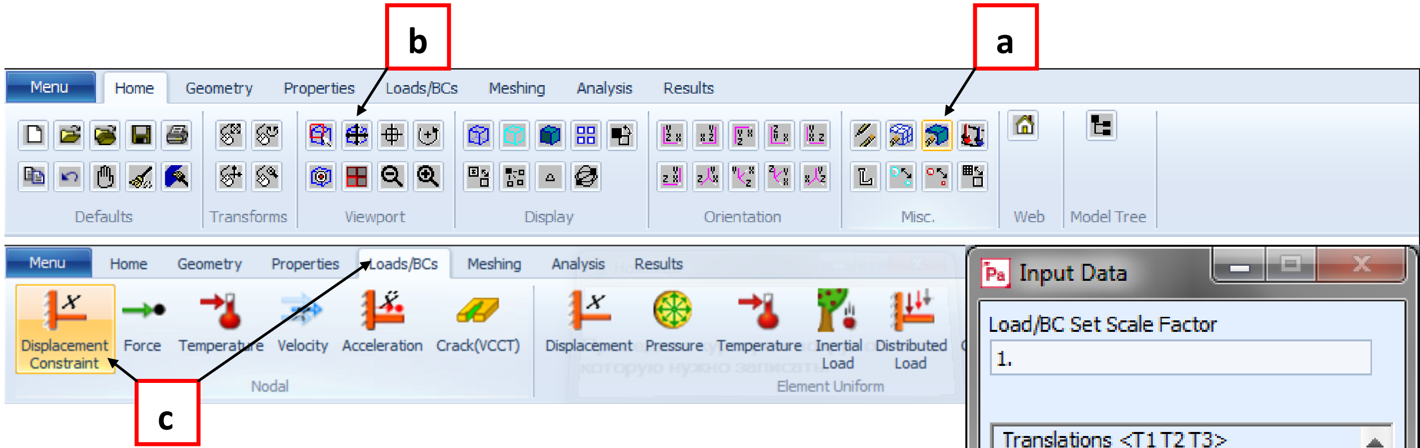
a. Click on the **Smooth shaded** icon

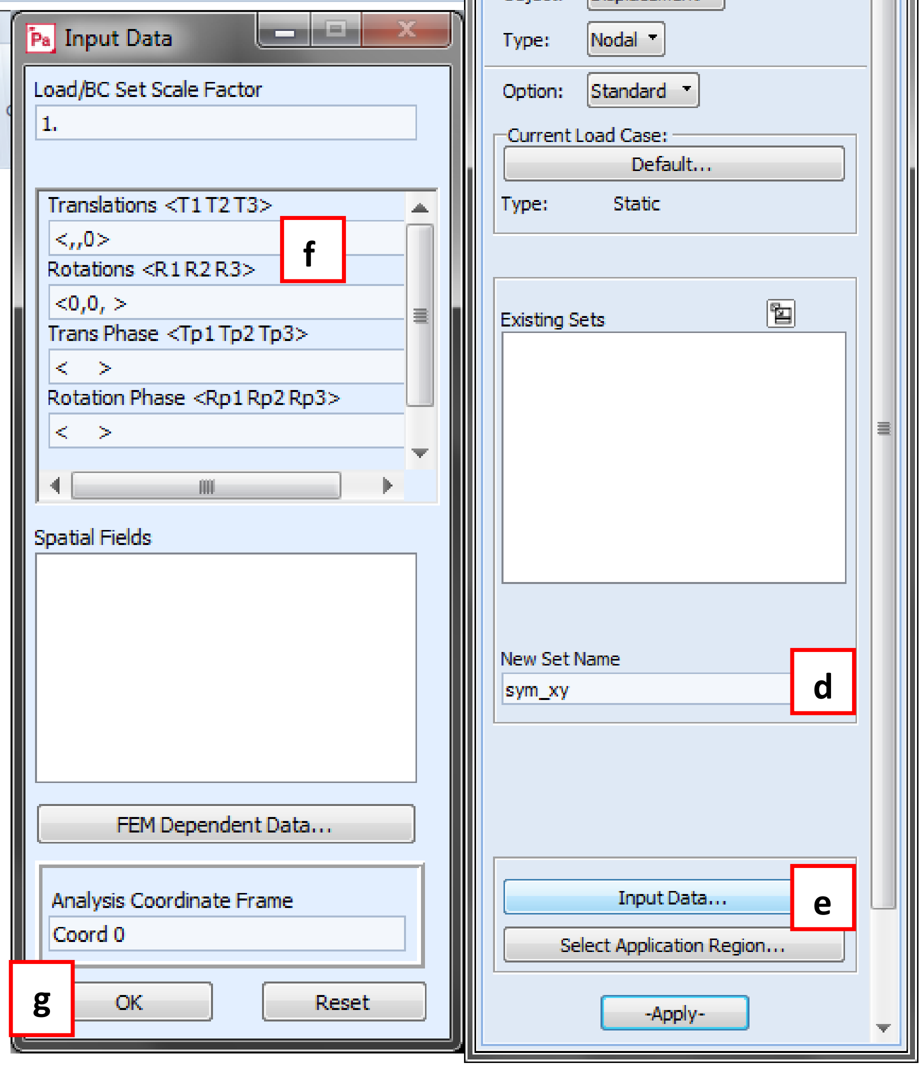
Mesh the surface:

b. Click on the **Meshing** icon/**Surface** icon *(Meshers tab)*
c. Elem Shape: **Quad**; Mesher: **IsoMesh**; Topology: **Quad4**
d. Click on the **Surface List** panel
e. Select the surface
f. Uncheck **Automatic Calculation**
g. Enter **40** as the Value of the Global Edge Length
h. Click **Apply**



HUMAN CAPITAL
NATIONAL COHESION STRATEGY
EUROPEAN UNION
EUROPEAN SOCIAL FUND

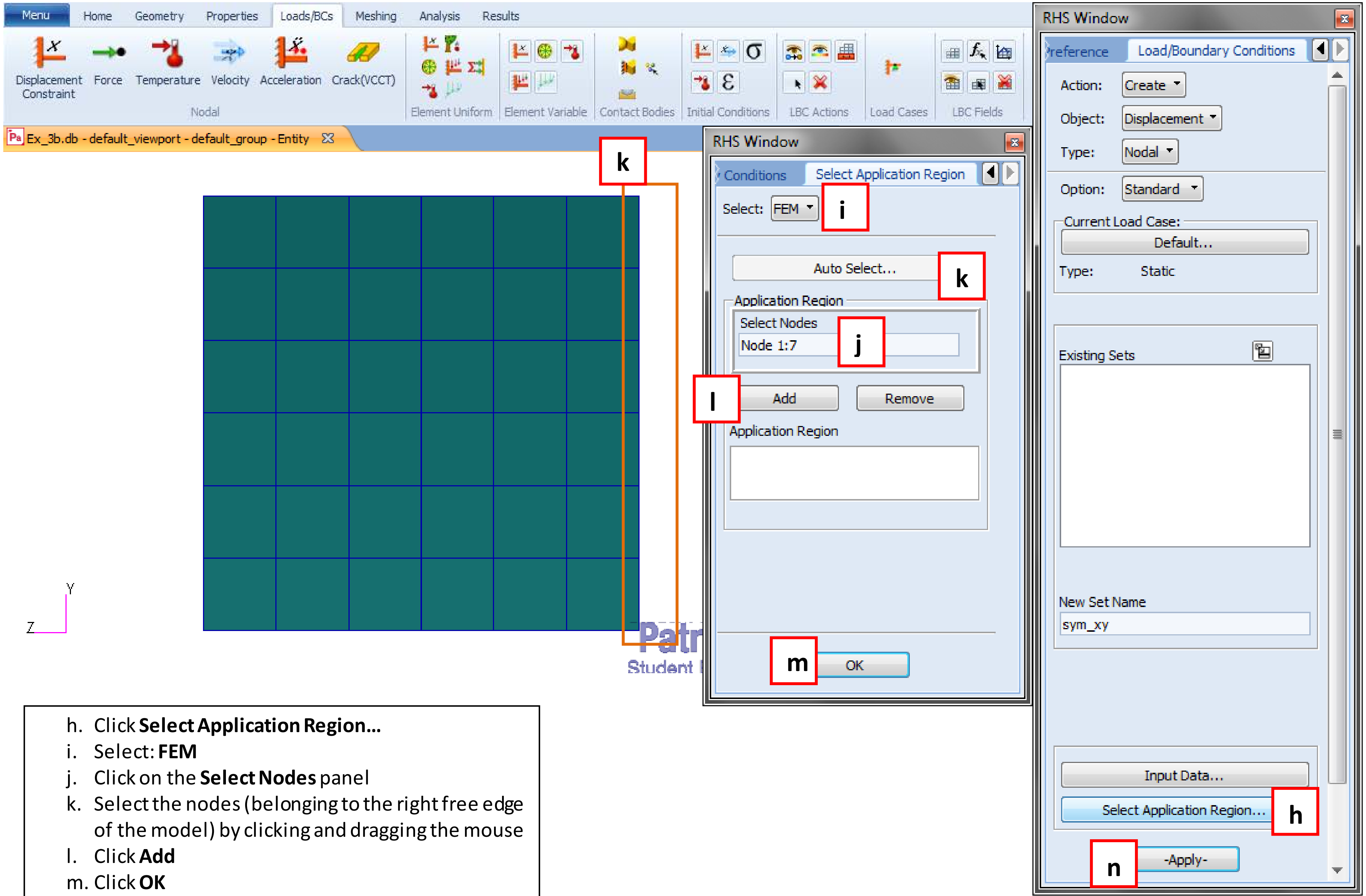




a. Click on the **Plot/Erase Geometry** icon
b. Click on the **Fit view** icon

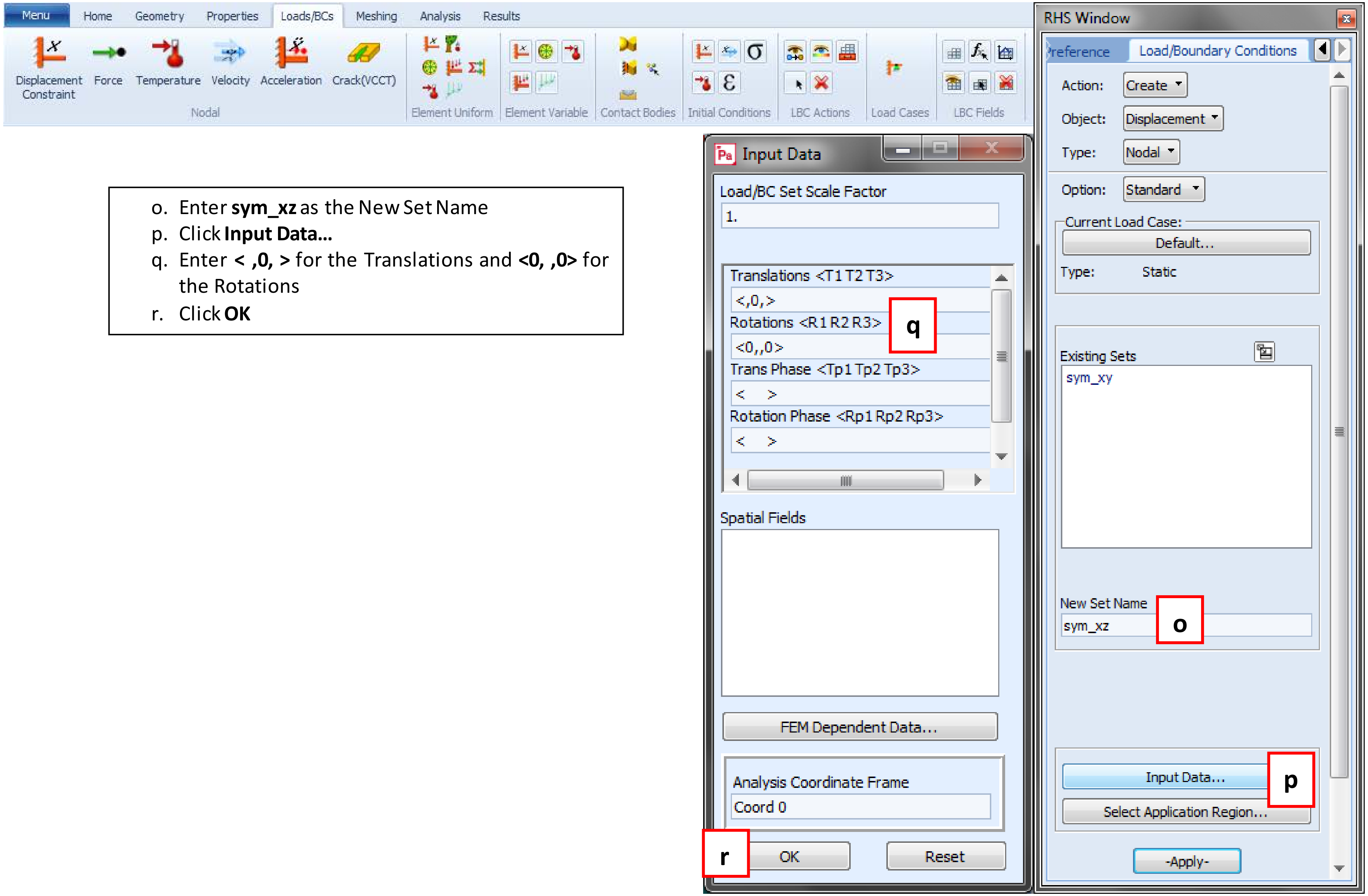
Apply the boundary conditions:

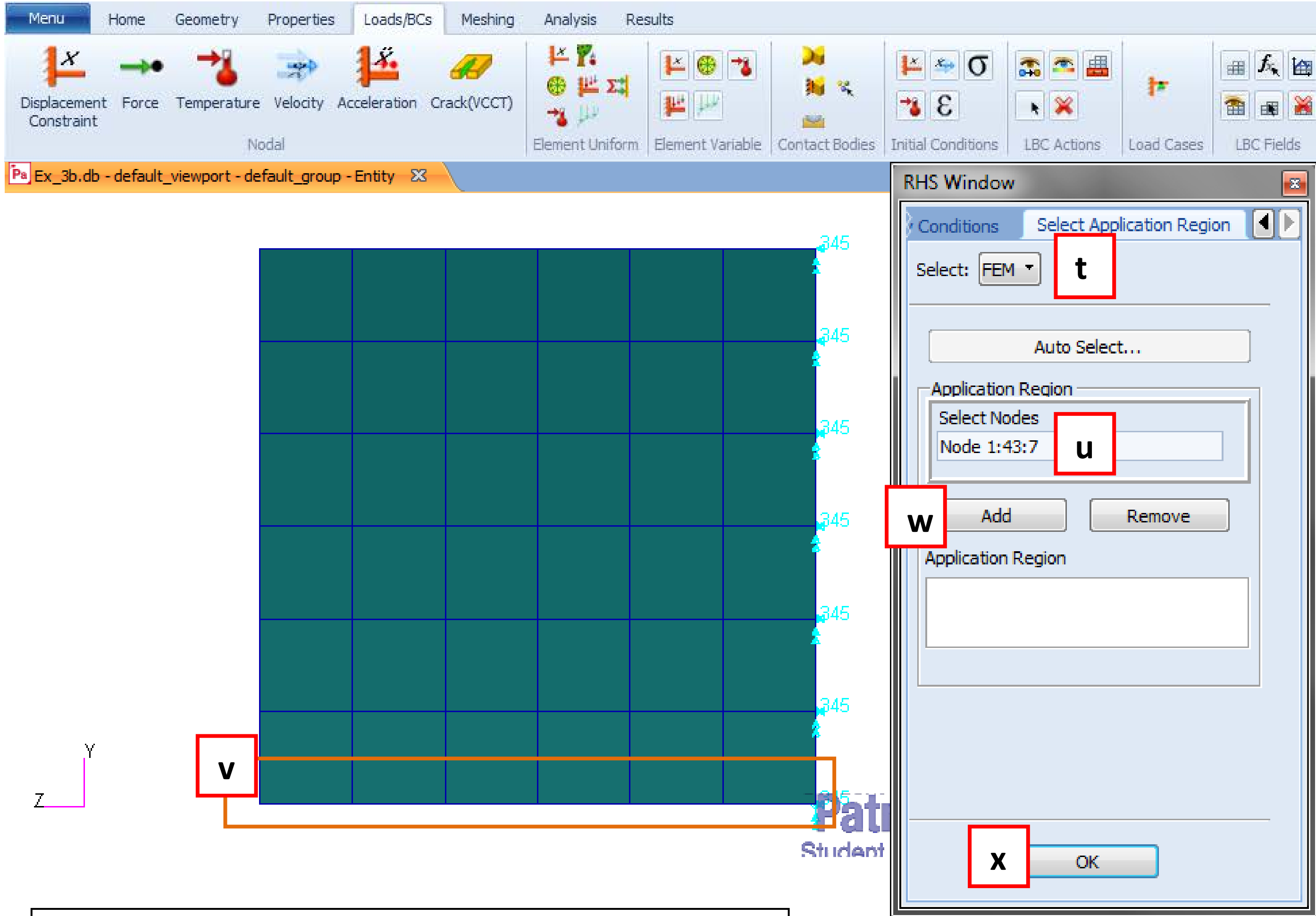
c. Click on the **Loads/BCs** icon/**Displacement Constraint** icon
d. Enter **sym_xy** as the New Set Name
e. Click **Input Data...**
f. Enter **<,,0>** for the Translations and **<0,0, >** for the Rotations
g. Click **OK**

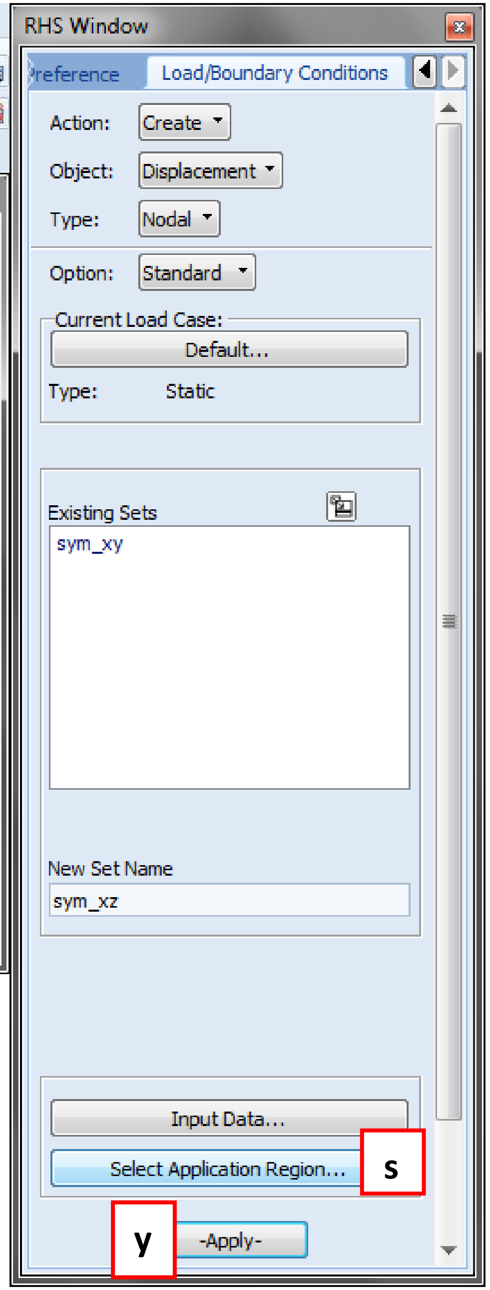


h. Click **Select Application Region...**
i. Select: **FEM**
j. Click on the **Select Nodes** panel
k. Select the nodes (belonging to the right free edge of the model) by clicking and dragging the mouse
l. Click **Add**
m. Click **OK**
n. Click **Apply**

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







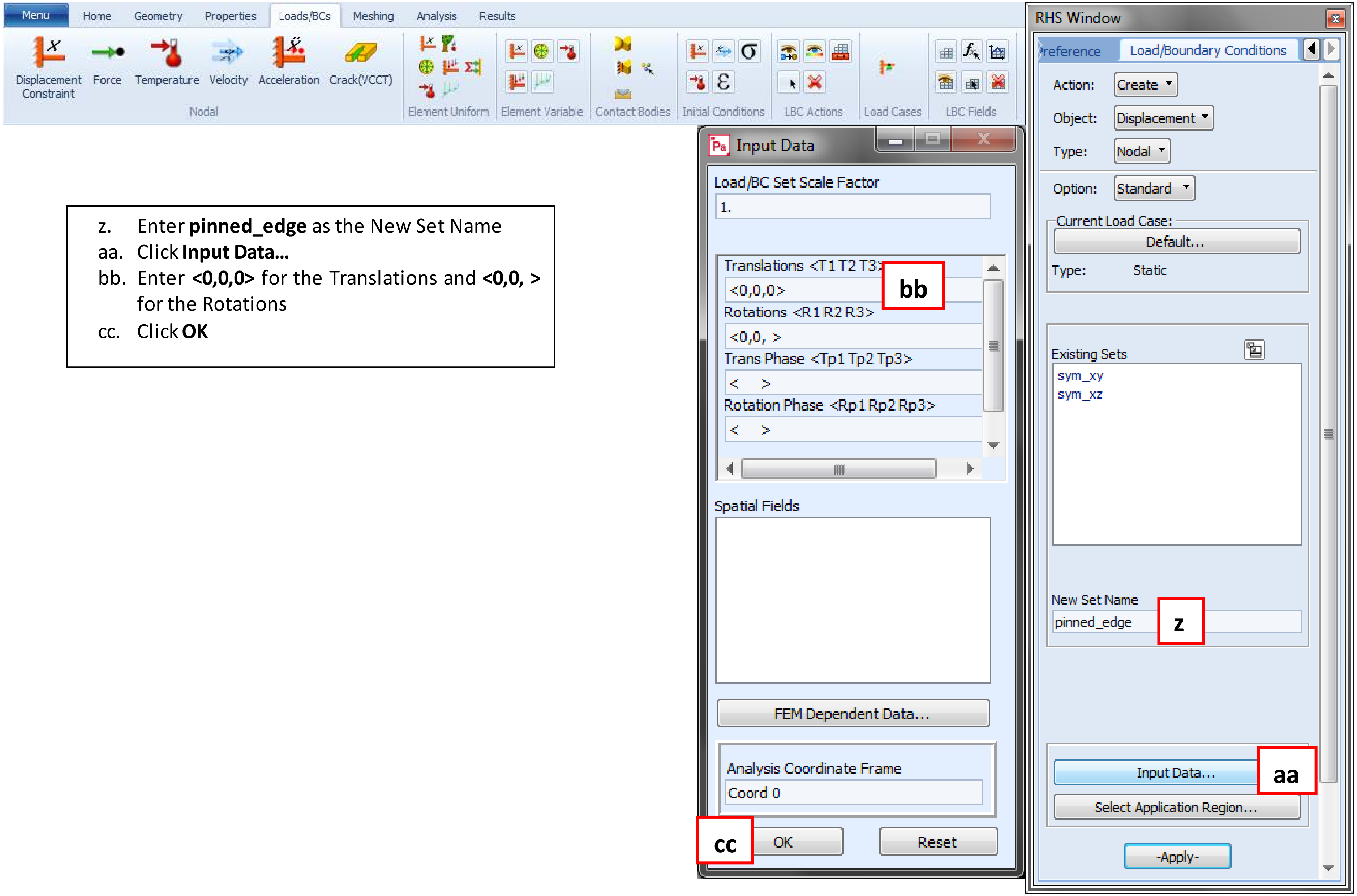
s. Click **Select Application Region...**
t. Select: **FEM**
u. Click on the **Select Nodes** panel
v. Select the nodes (belonging to the bottom free edge of the model) by clicking and dragging the mouse
w. Click **Add**
x. Click **OK**
y. Click **Apply**

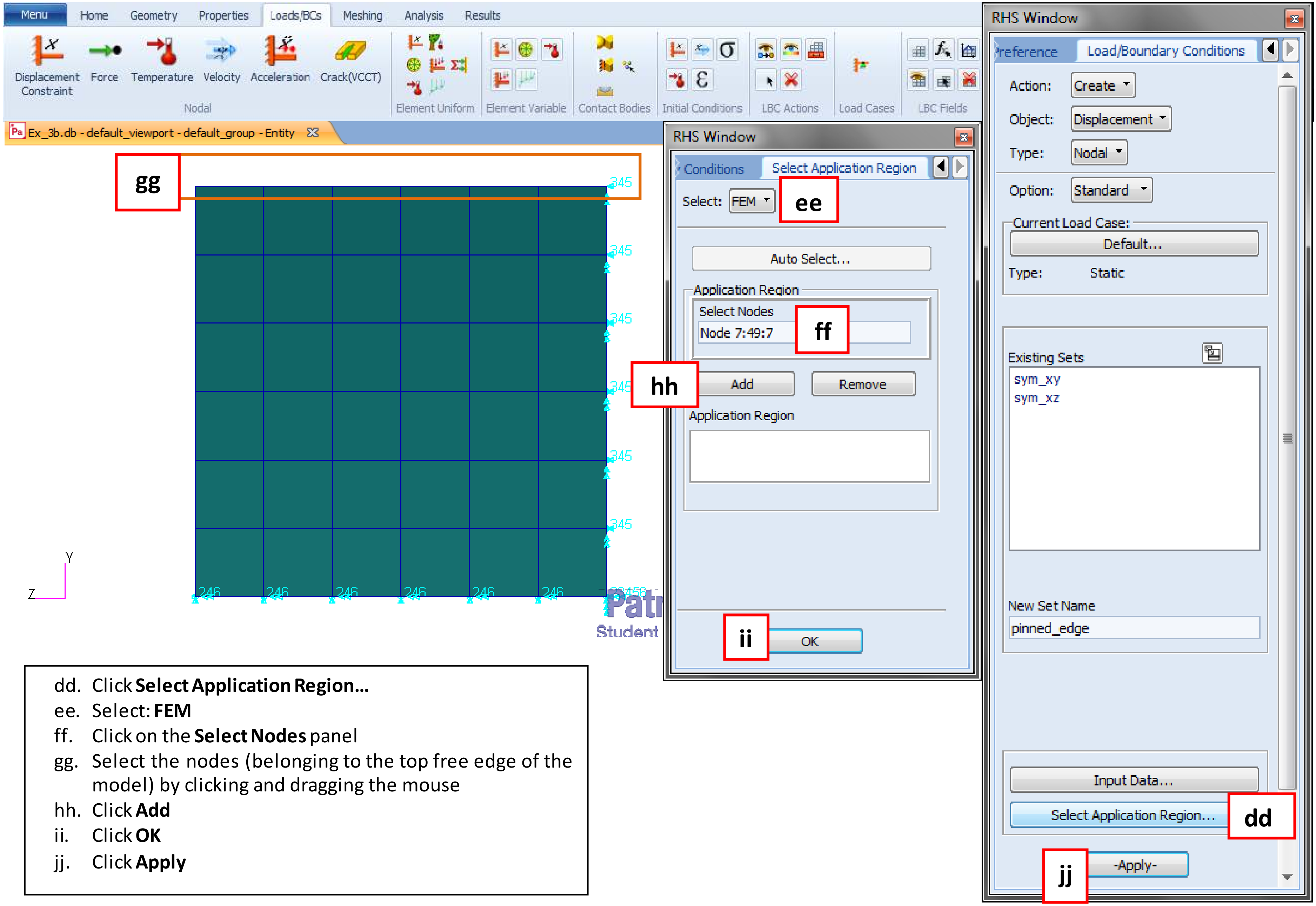
z. Enter **pinned_edge** as the New Set Name

aa. Click **Input Data...**

bb. Enter **<0,0,0>** for the Translations and **<0,0, >** for the Rotations

cc. Click **OK**





dd. Click **Select Application Region...**

ee. Select: **FEM**

ff. Click on the **Select Nodes** panel

gg. Select the nodes (belonging to the top free edge of the model) by clicking and dragging the mouse

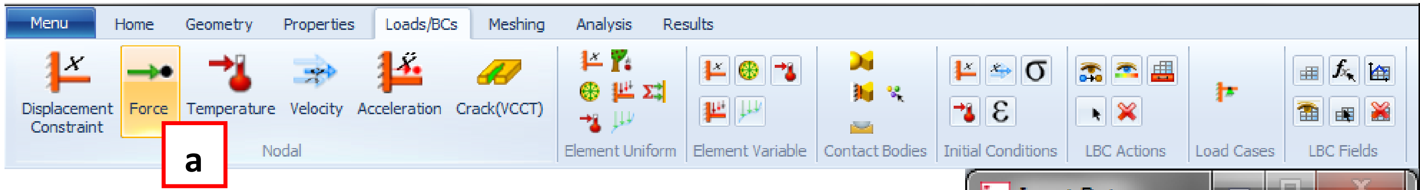
hh. Click **Add**

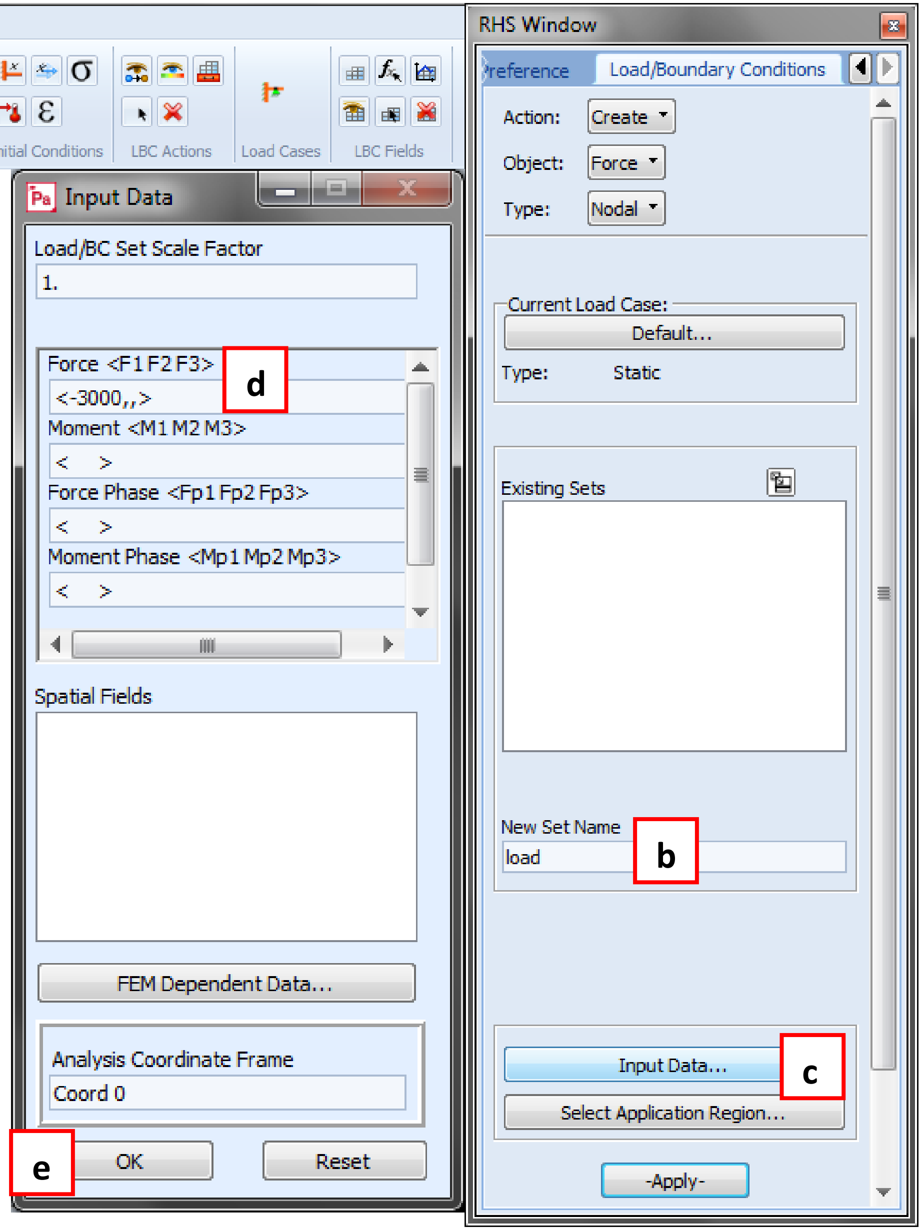
ii. Click **OK**

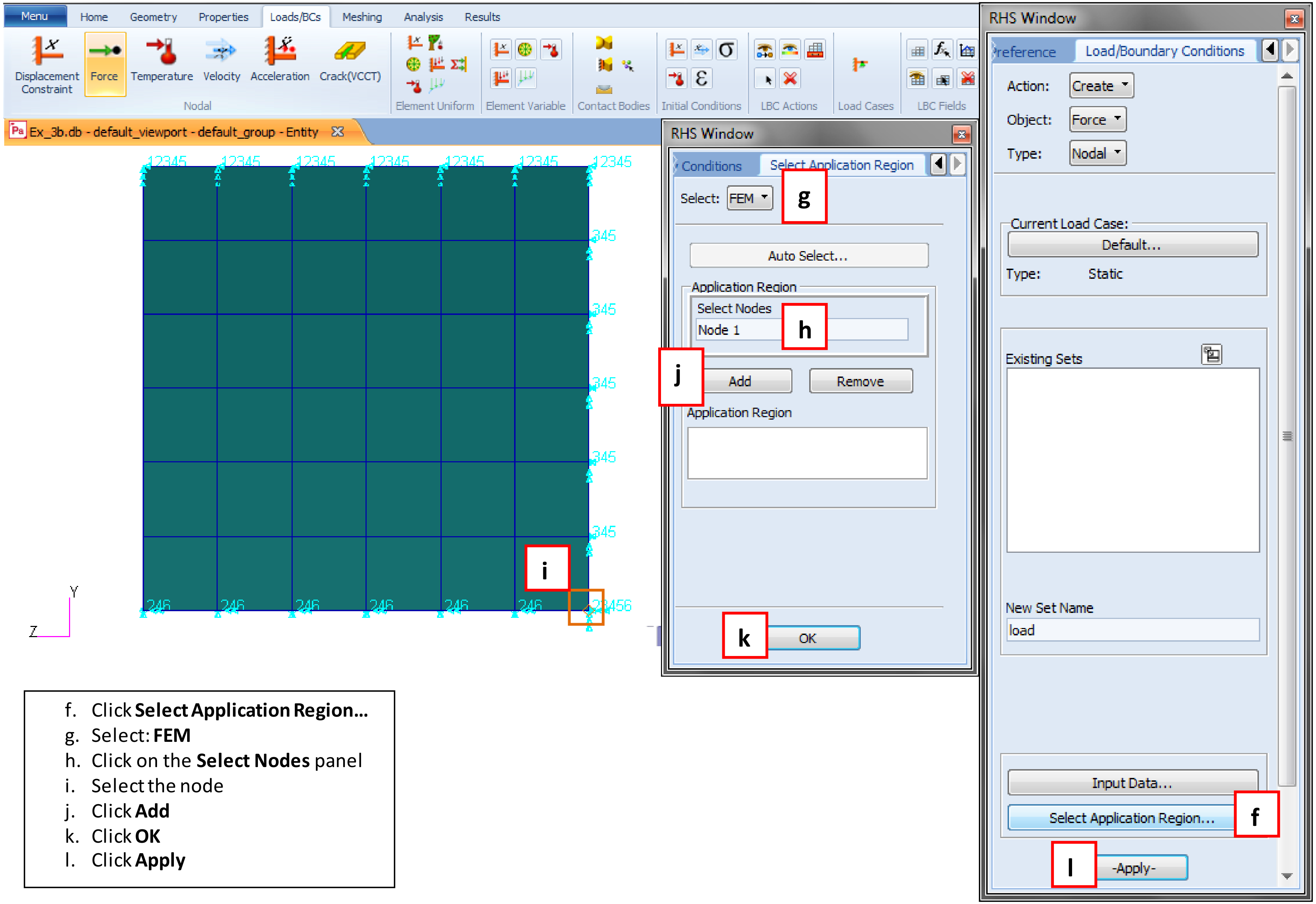
jj. Click **Apply**

Apply the force:

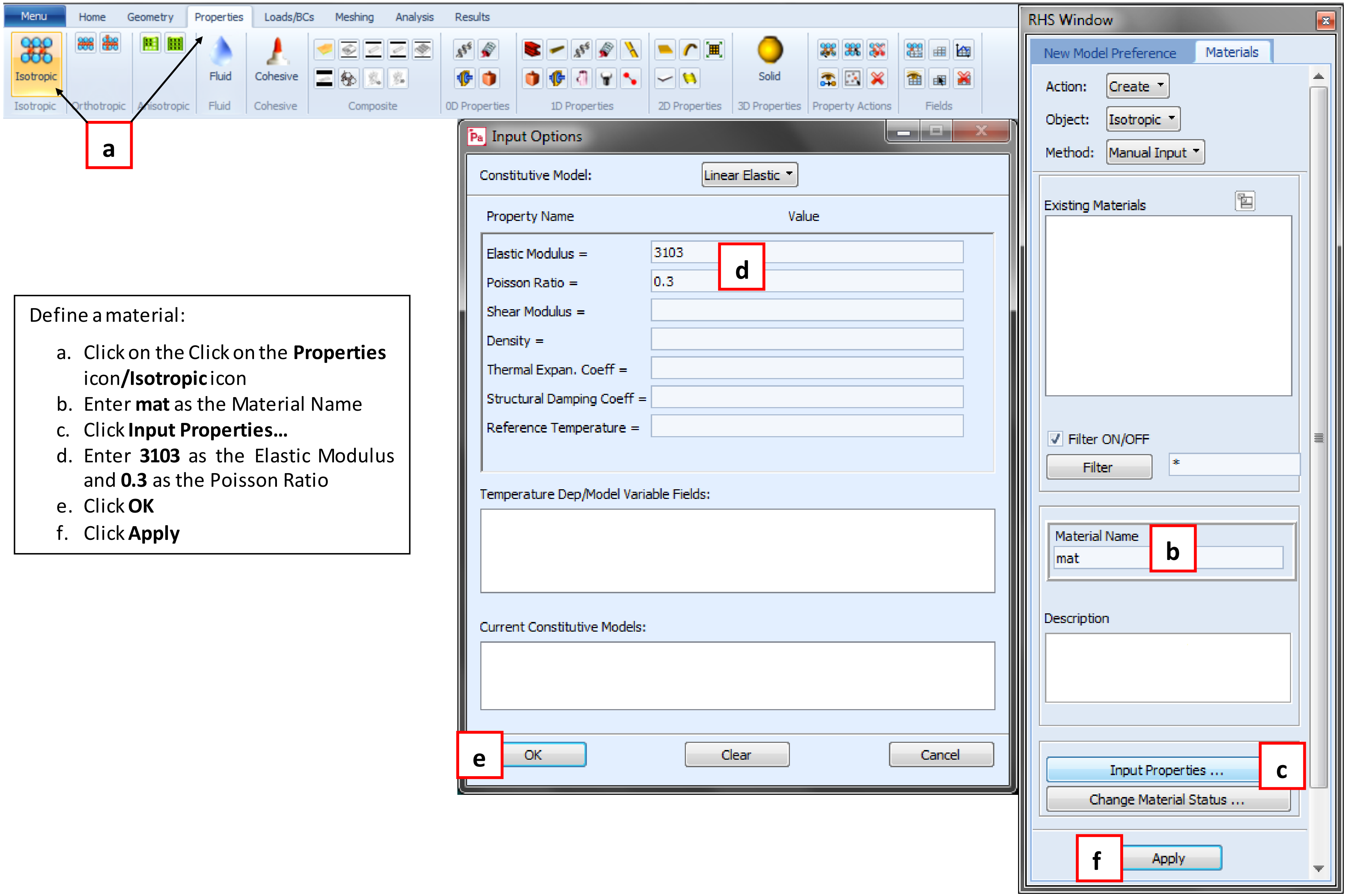
a. Loads/BCs: **Force** icon
b. Enter **load** as the New Set Name
c. Click **Input Data...**
d. Enter **<-3000,,>** for the Force
e. Click **OK**





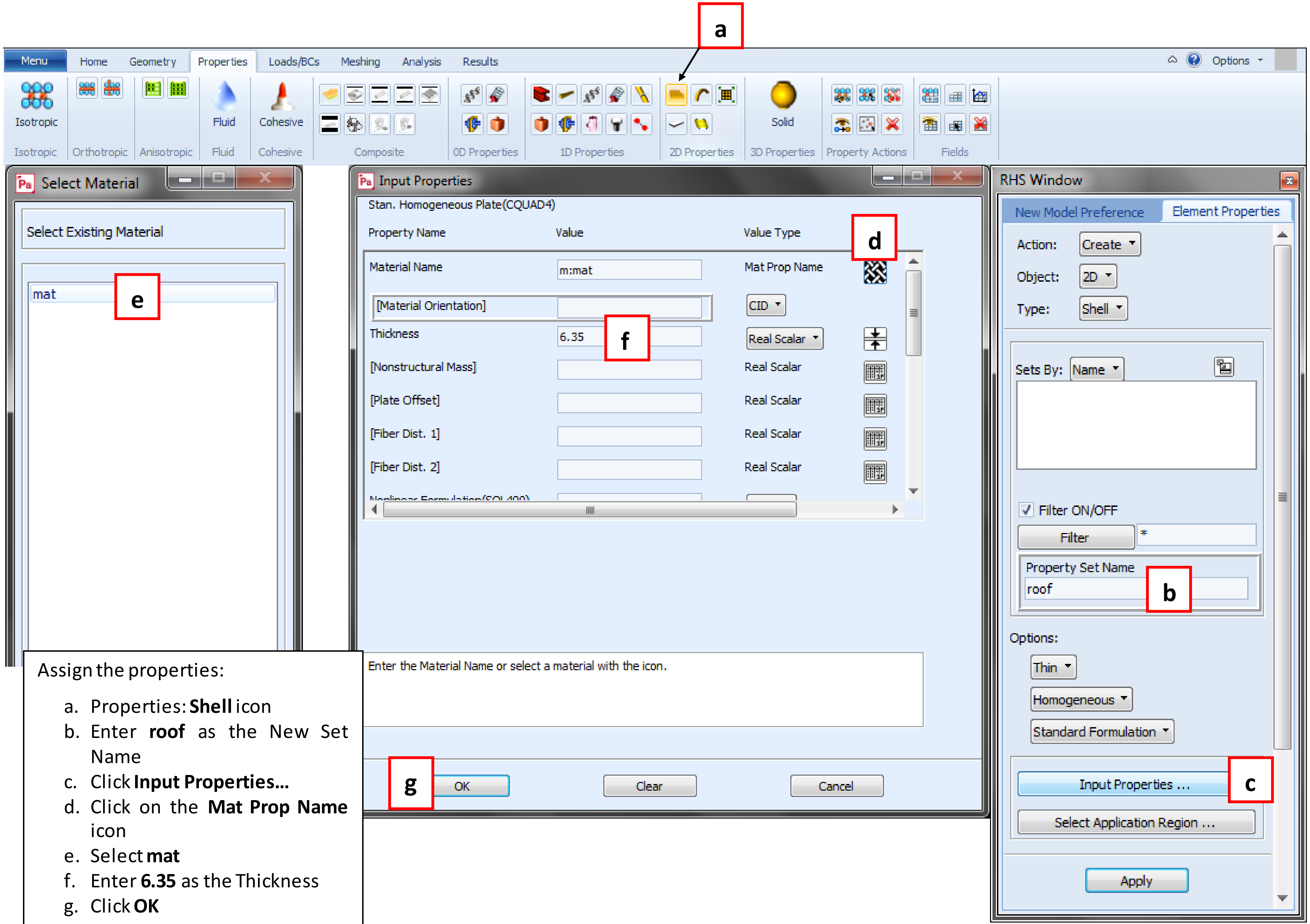


f. Click **Select Application Region...**
g. Select: **FEM**
h. Click on the **Select Nodes** panel
i. Select the node
j. Click **Add**
k. Click **OK**
l. Click **Apply**



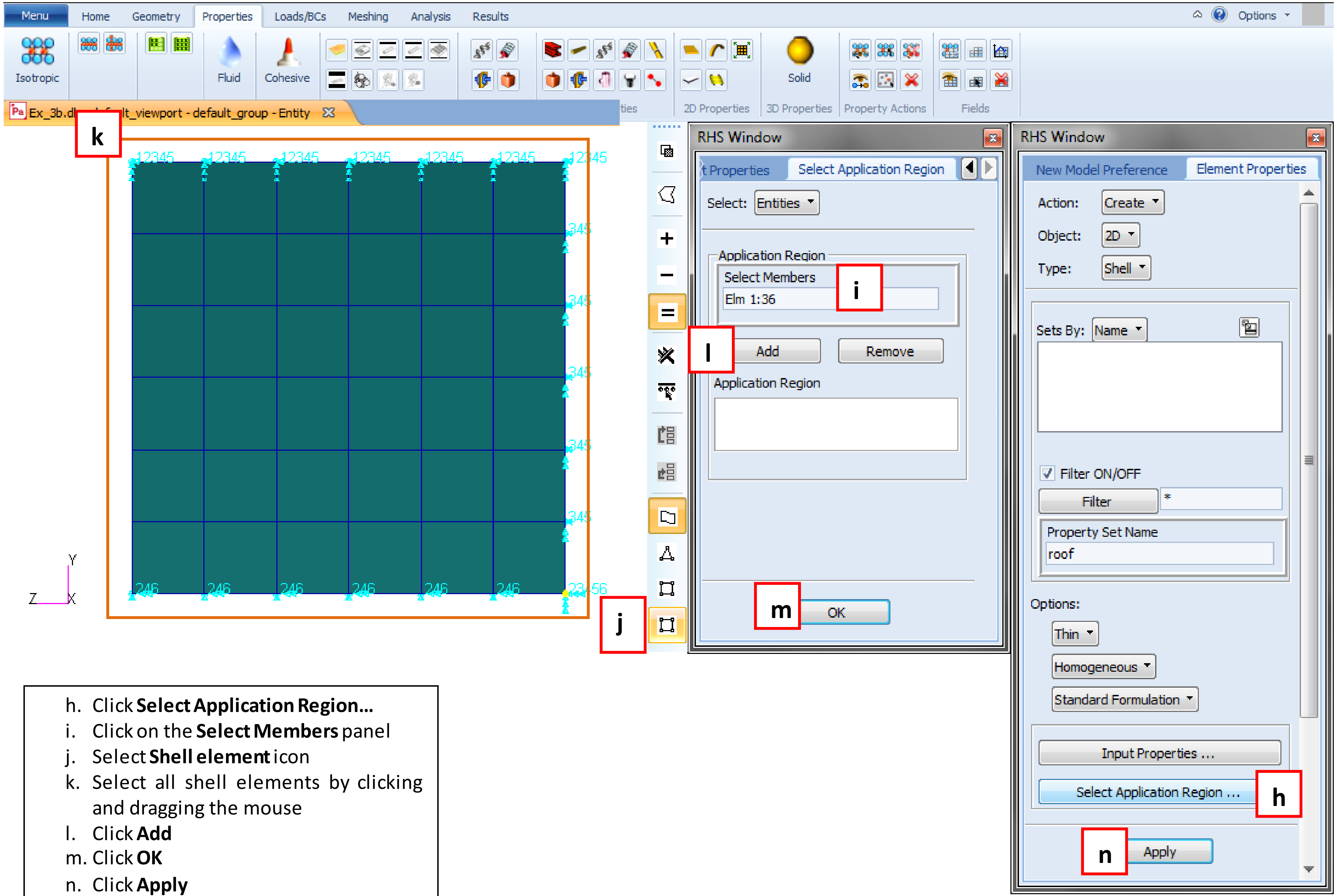
Define a material:

a. Click on the Click on the **Properties** icon/**Isotropic** icon
b. Enter **mat** as the Material Name
c. Click **Input Properties...**
d. Enter **3103** as the Elastic Modulus and **0.3** as the Poisson Ratio
e. Click **OK**
f. Click **Apply**

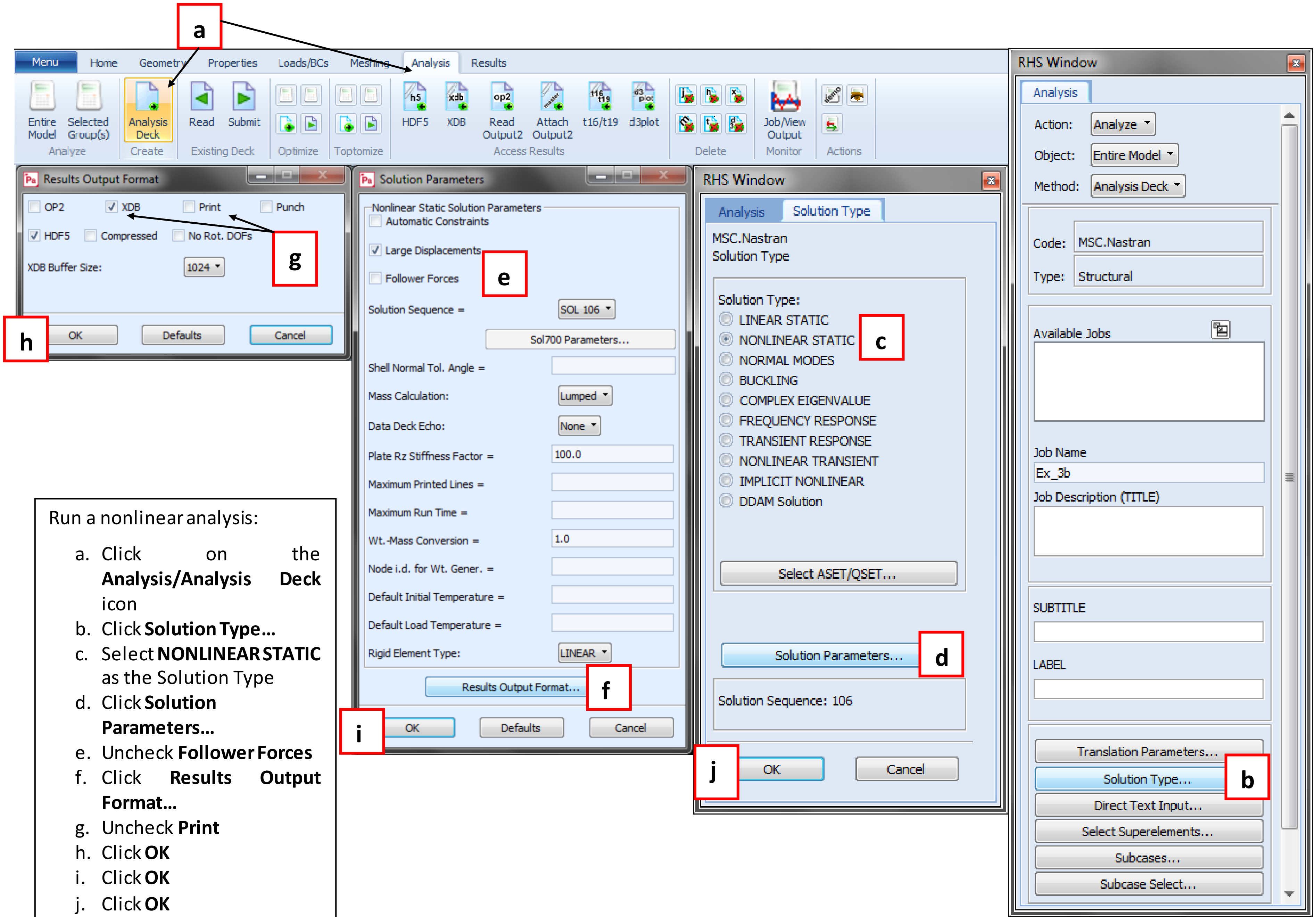


Assign the properties:

a. Properties: **Shell** icon
b. Enter **roof** as the New Set Name
c. Click **Input Properties...**
d. Click on the **Mat Prop Name** icon
e. Select **mat**
f. Enter **6.35** as the Thickness
g. Click **OK**

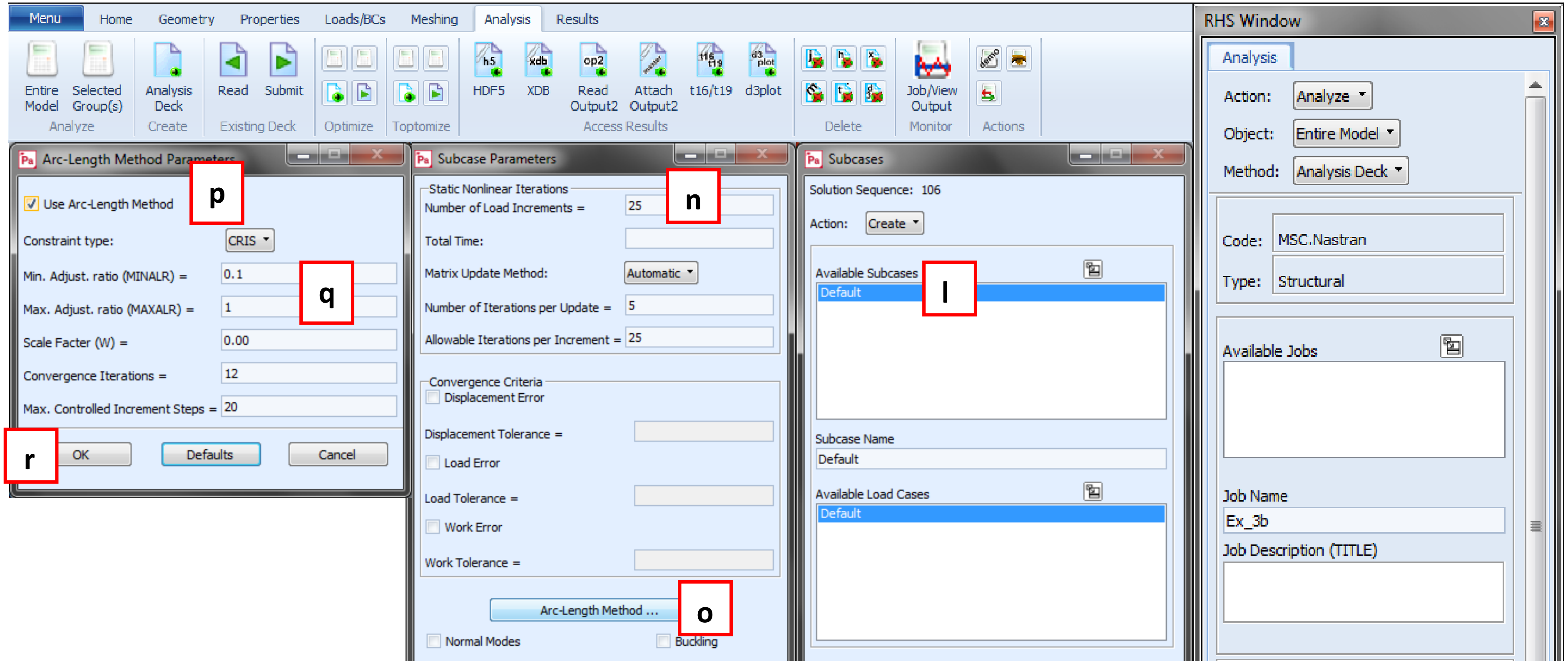


h. Click **Select Application Region...**
i. Click on the **Select Members** panel
j. Select **Shell element** icon
k. Select all shell elements by clicking and dragging the mouse
l. Click **Add**
m. Click **OK**
n. Click **Apply**

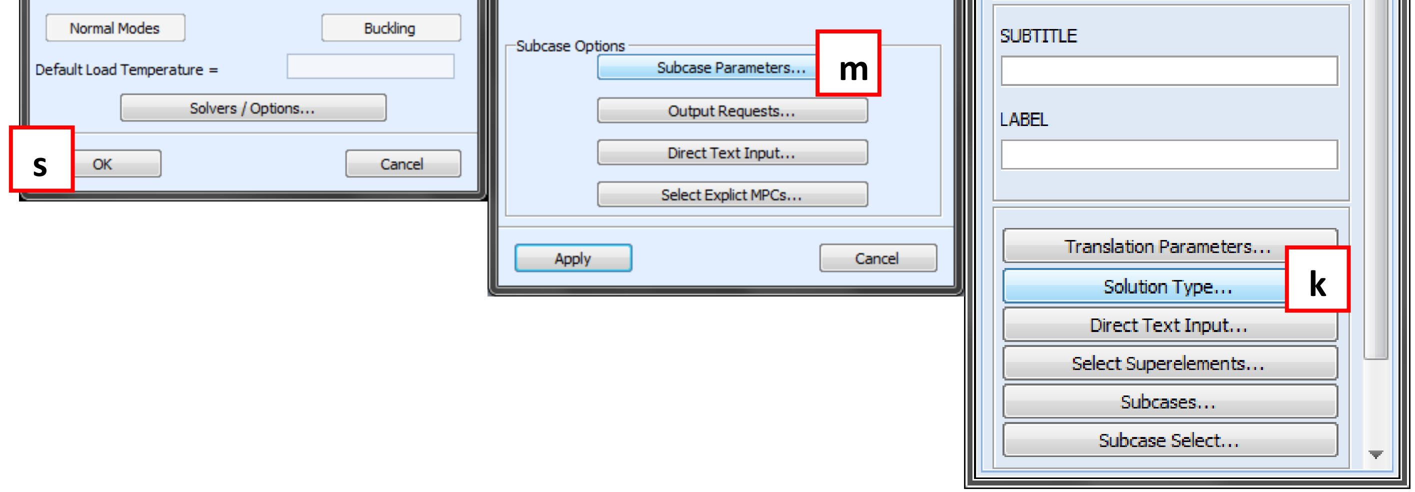


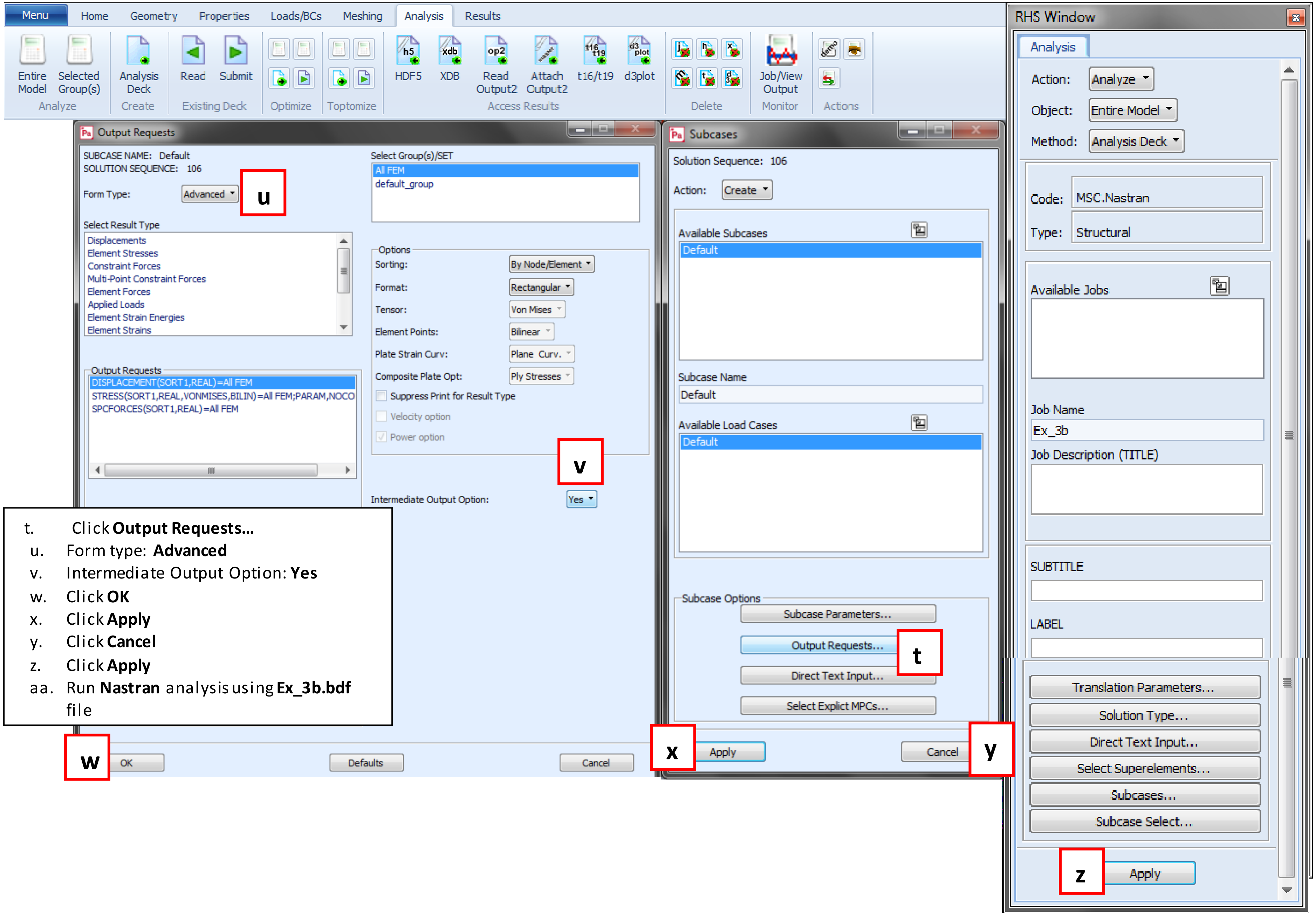
Run a nonlinear analysis:

a. Click on the **Analysis/Analysis Deck** icon
b. Click **Solution Type...**
c. Select **NONLINEAR STATIC** as the Solution Type
d. Click **Solution Parameters...**
e. Uncheck **Follower Forces**
f. Click **Results Output Format...**
g. Uncheck **Print**
h. Click **OK**
i. Click **OK**
j. Click **OK**

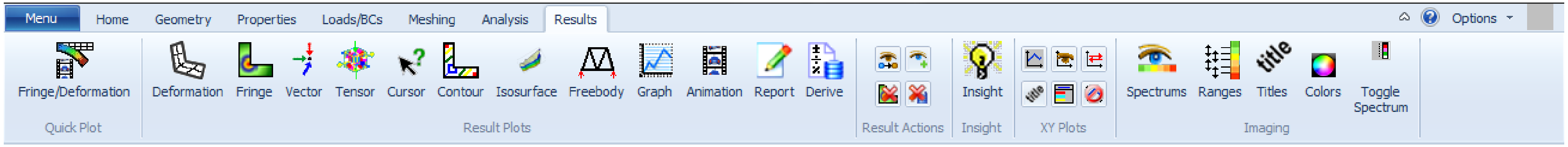


k. Click **Subcases...**
l. Select **Default**
m. Click **Subcase Parameters...**
n. Enter **25** as the Number of Load Increments
o. Click **Arc-Length Method**
p. Select **CRIS** and check **Use Arc-Length Method**
q. Enter **0.1** as the Min. Adjust. ratio and **1** as the Max. Adjust. ratio
r. Click **OK**
s. Click **OK**

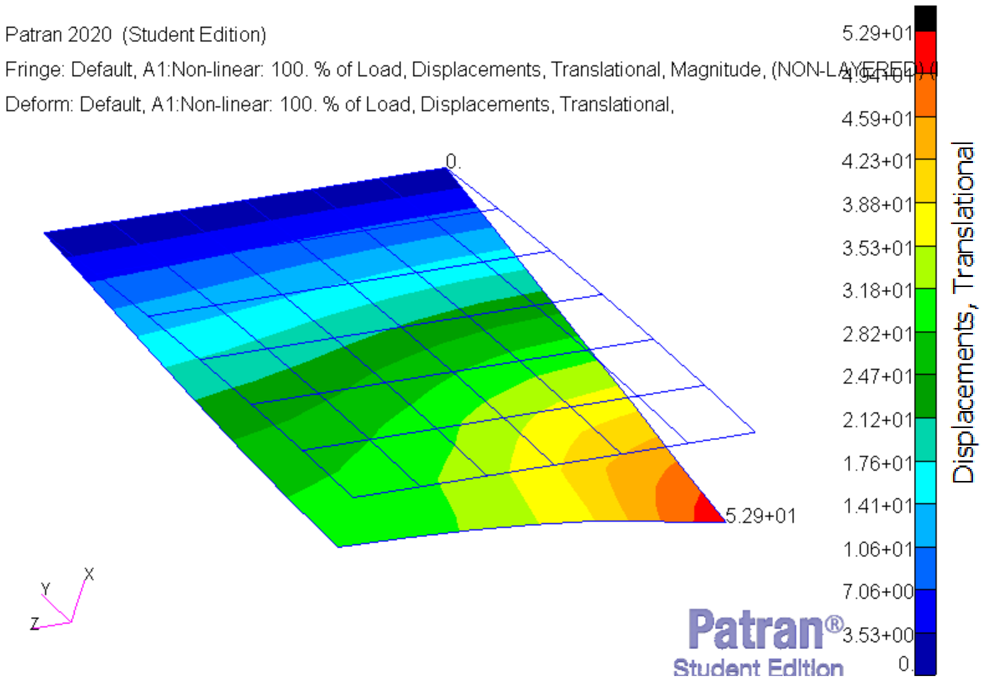




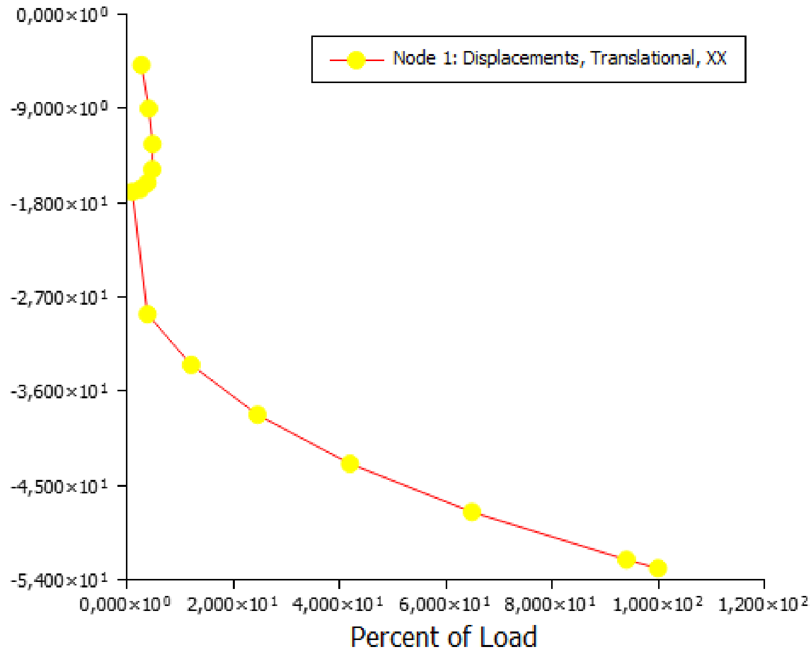
t. Click **Output Requests...**
u. Form type: **Advanced**
v. Intermediate Output Option: **Yes**
w. Click **OK**
x. Click **Apply**
y. Click **Cancel**
z. Click **Apply**
aa. Run **Nastran** analysis using **Ex_3b.bdf** file



a. Attach the results file
b. Post-process the results



100% of Load; magnitude displacements



Equilibrium path