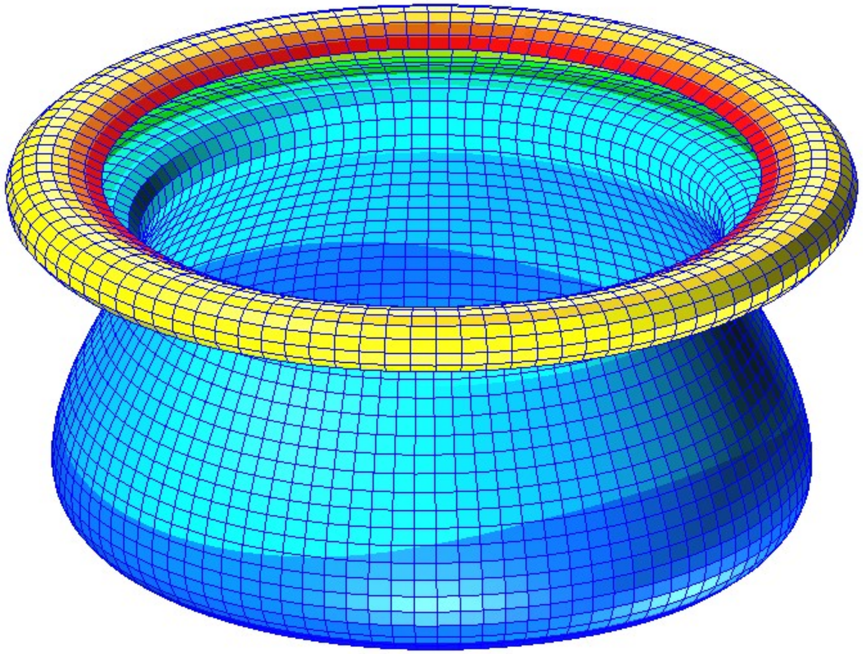
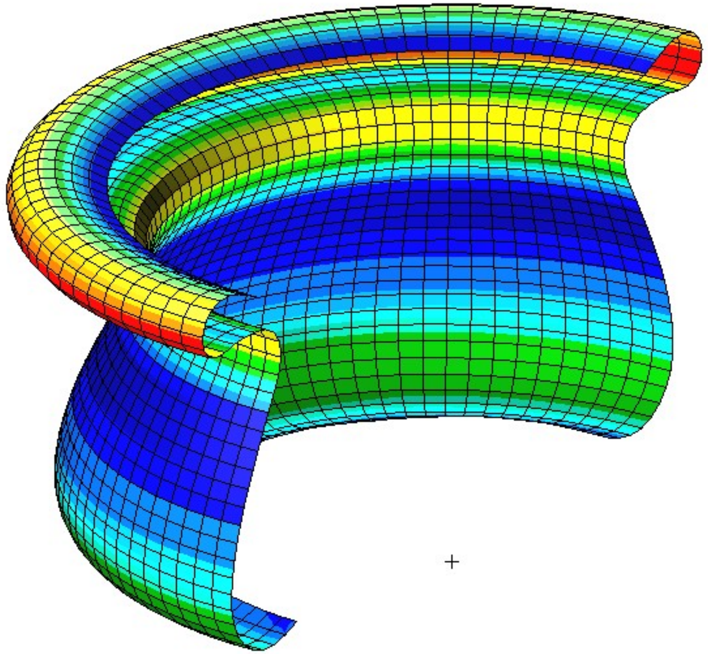
NONLINEAR MECHANICS OF STRUCTURES

EXERCISE 5

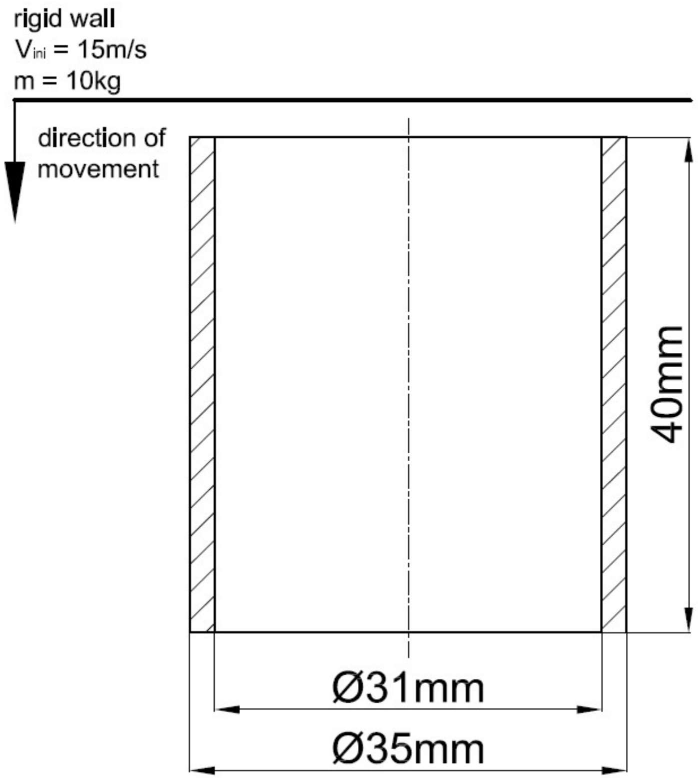




HUMAN CAPITAL
NATIONAL COHESION STRATEGY

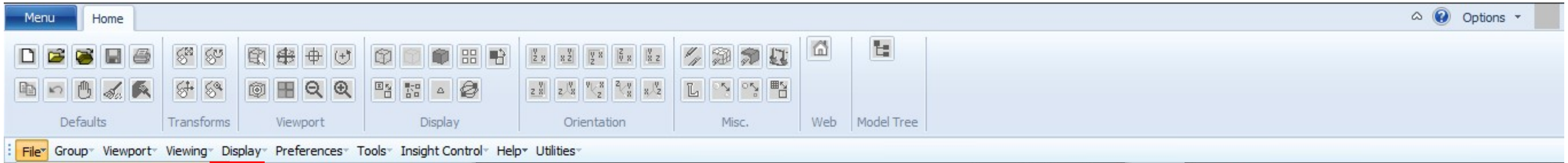
EUROPEAN UNION
EUROPEAN
SOCIAL FUND

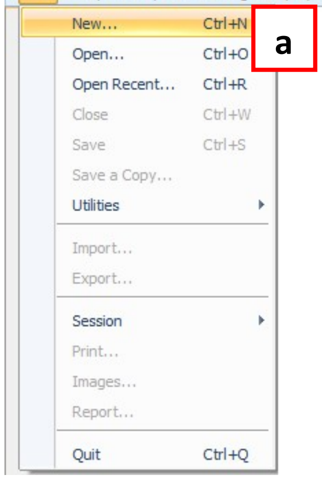
PROBLEM DESCRIPTION

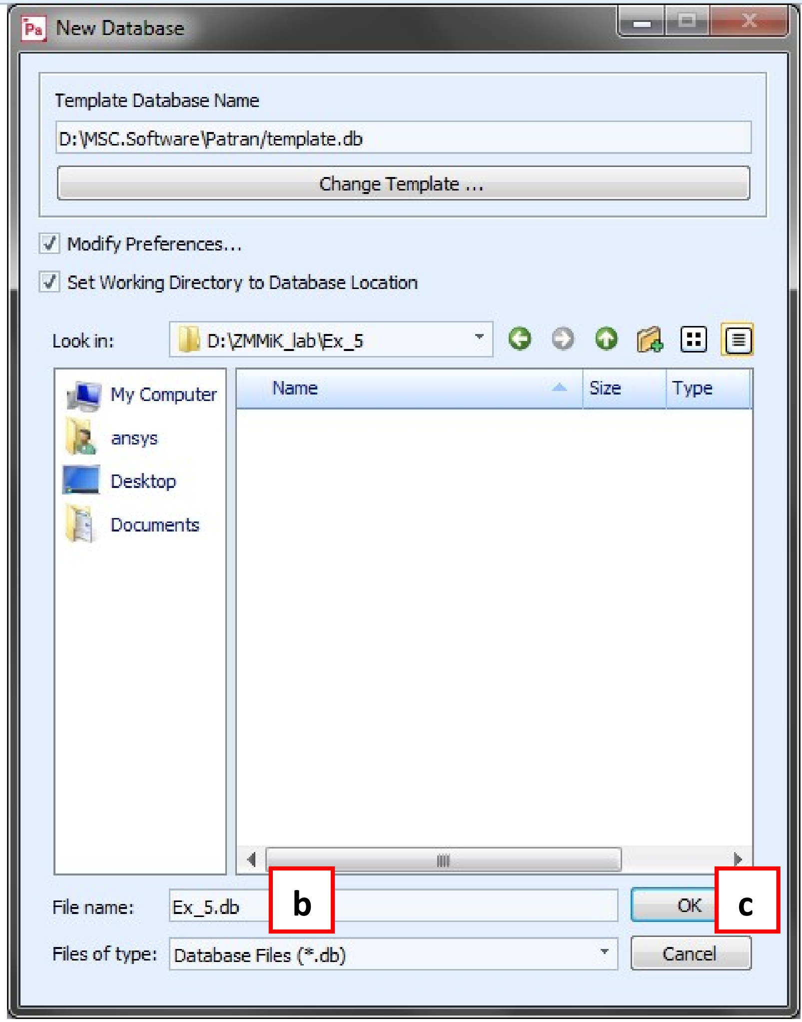


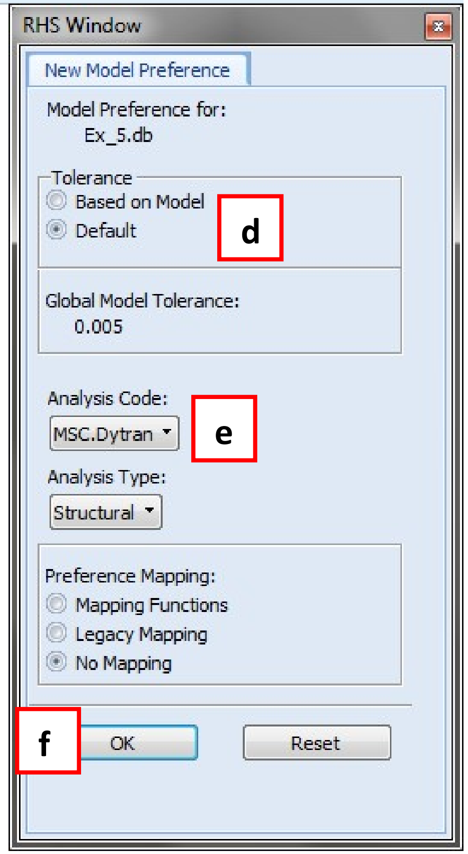
A thin-walled aluminum cylindrical element is struck by a rigid wall. The mass of the wall is 10kg and its initial velocity is equal to 15m/s. The goal of this exercise is to access the crashworthiness of the element (which can be used further e.g. in the crumple zone of a passenger car).

Units: mm, kg, ms, kN, GPa



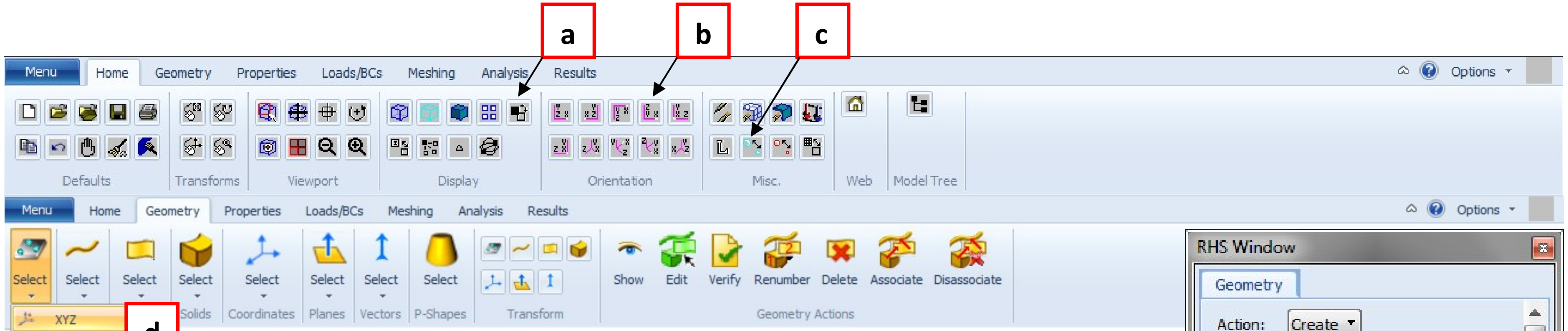


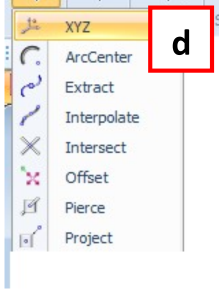




Create a new database:

a. **File / New...**
b. Enter **Ex_5.db** as the File name
c. Click **OK**
d. Select **Default**
e. Select **MSC.Dytran** as the Analysis Code
f. Click **OK**

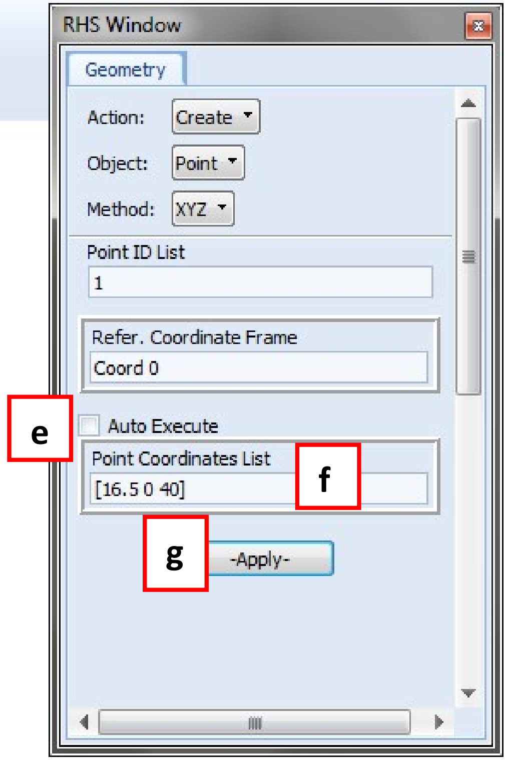


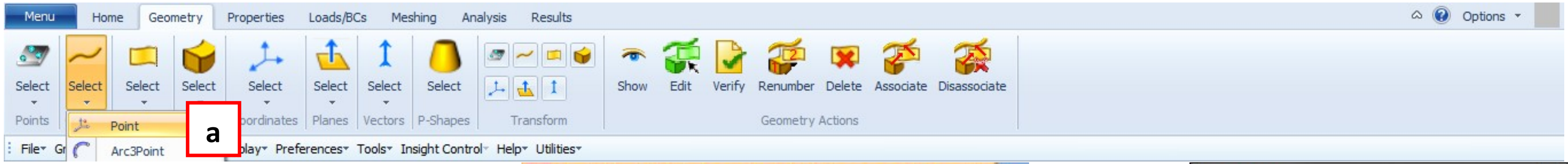


- a. Change *Background Color* to **Black** (click on the **Cycle Background** icon)
- b. Click on the **Bottom view** icon
- c. Click on the **Point size** icon

Create geometry points:

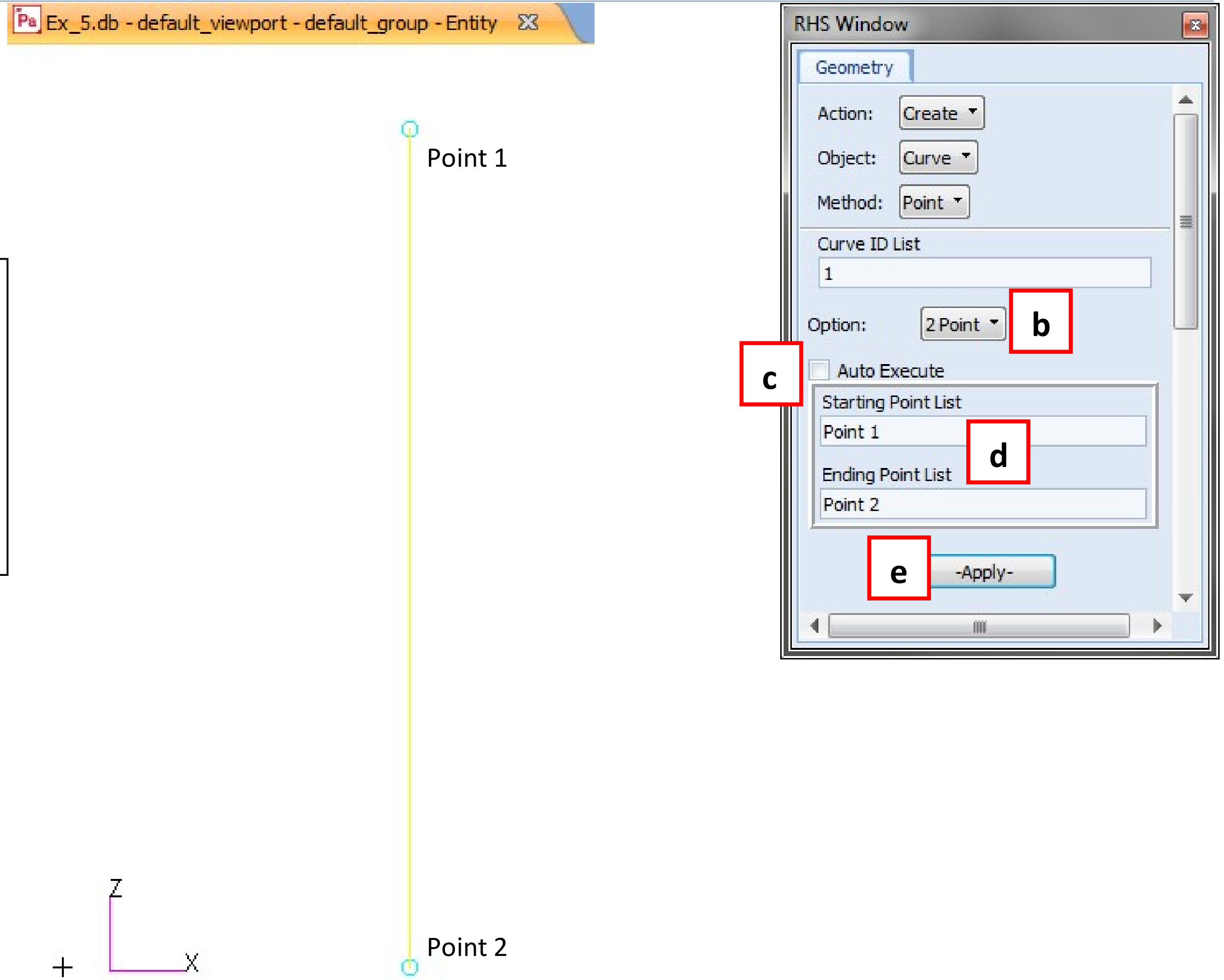
- d. Click on the **Geometry** icon**/Points** icon**/Select/XYZ**
- e. Uncheck **Auto Execute**
- f. Enter **[16.5 0 40]** as the Point Coordinates List
- g. Click **Apply**
- h. Create one more point using coordinates: **[16.5 0 0]**

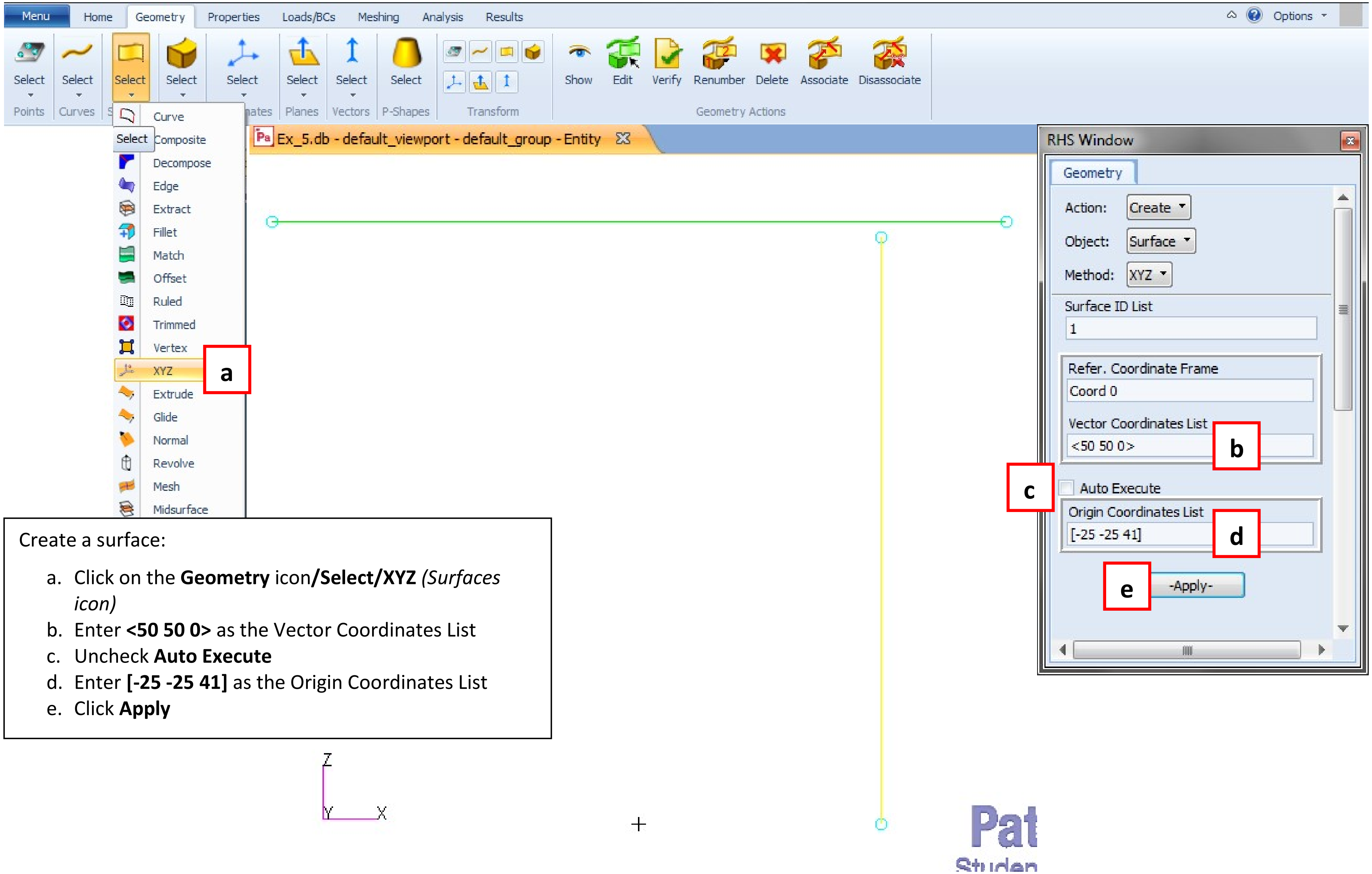




Create a curve:

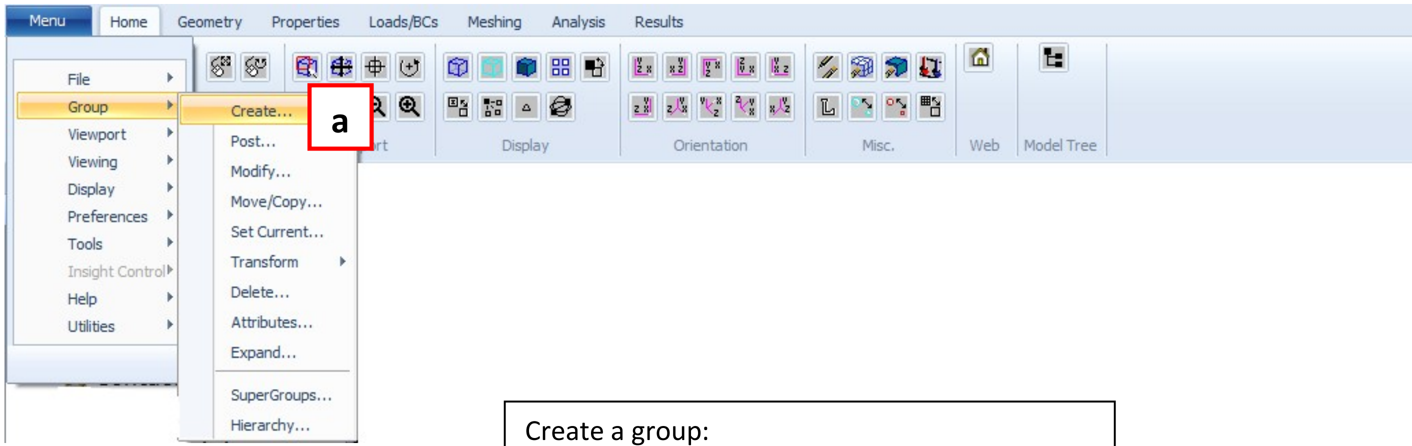
a. Click on the **Geometry** icon: **Curves** icon/**Select/Point**
b. Option: **2 Point**
c. Uncheck **Auto Execute**
d. Select **Point 1** as the starting point and **Point 2** as the ending point
e. Click **Apply**





Create a surface:

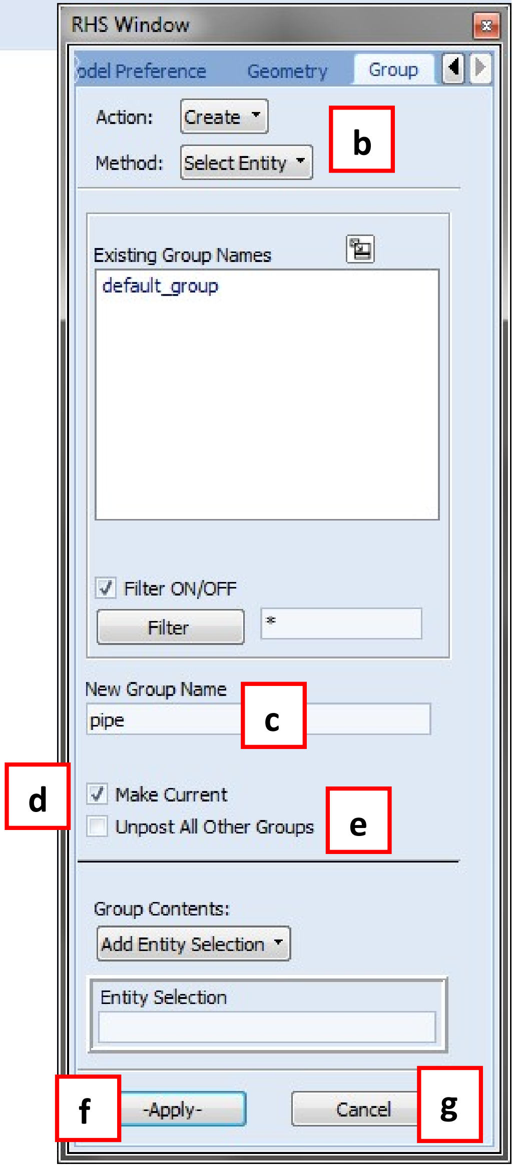
a. Click on the **Geometry** icon/**Select/XYZ** *(Surfaces icon)*
b. Enter **<50 50 0>** as the Vector Coordinates List
c. Uncheck **Auto Execute**
d. Enter **[-25 -25 41]** as the Origin Coordinates List
e. Click **Apply**

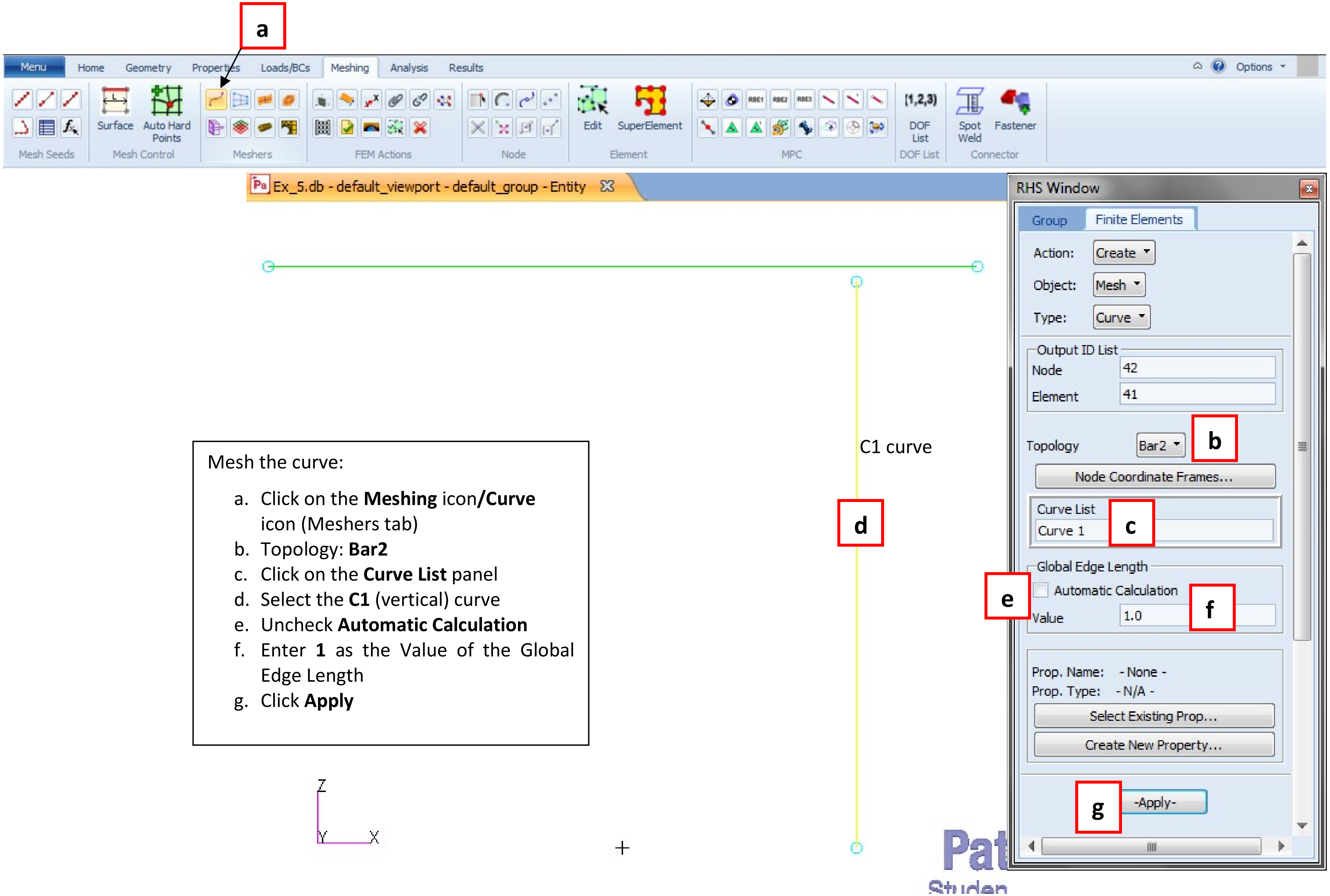


Create a group:

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**

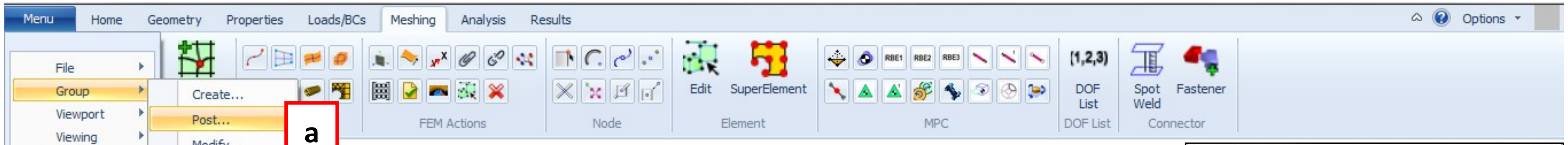
<u>Remark</u>: Mesh of the cylindrical element will be assigned to this group.

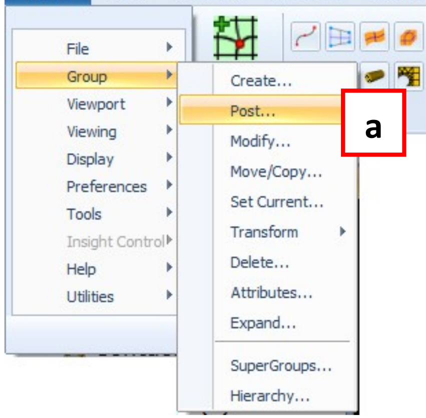




Mesh the curve:

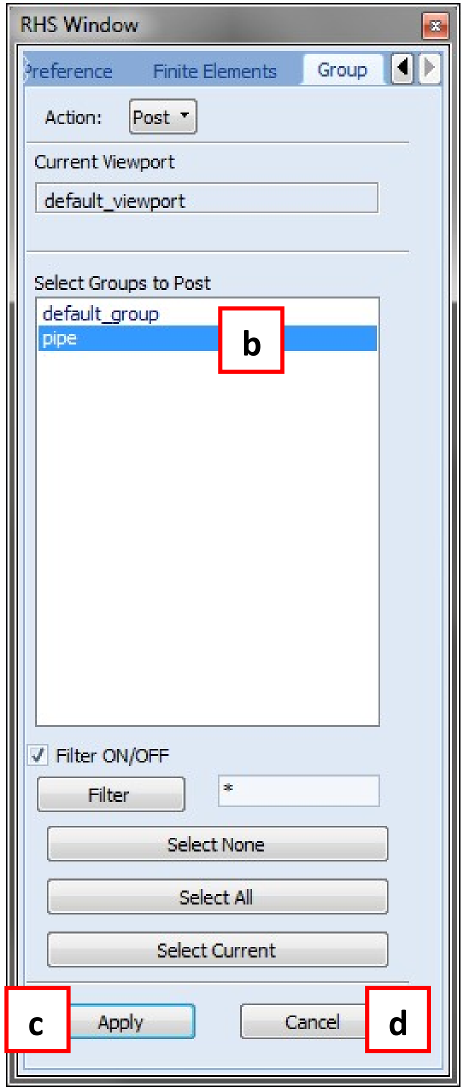
a. Click on the **Meshing** icon/**Curve** icon (Meshers tab)
b. Topology: **Bar2**
c. Click on the **Curve List** panel
d. Select the **C1** (vertical) curve
e. Uncheck **Automatic Calculation**
f. Enter **1** as the Value of the Global Edge Length
g. Click **Apply**





Post only the **pipe** group:

a. **Group / Post...**
b. Select **pipe**
c. Click **Apply**
d. Click **Cancel**

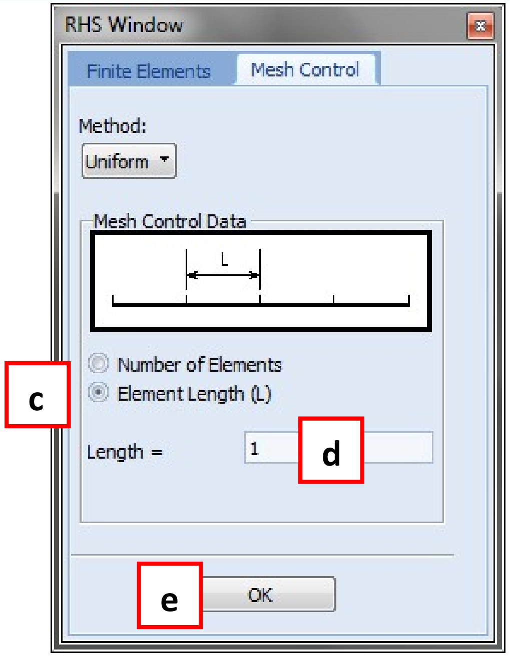


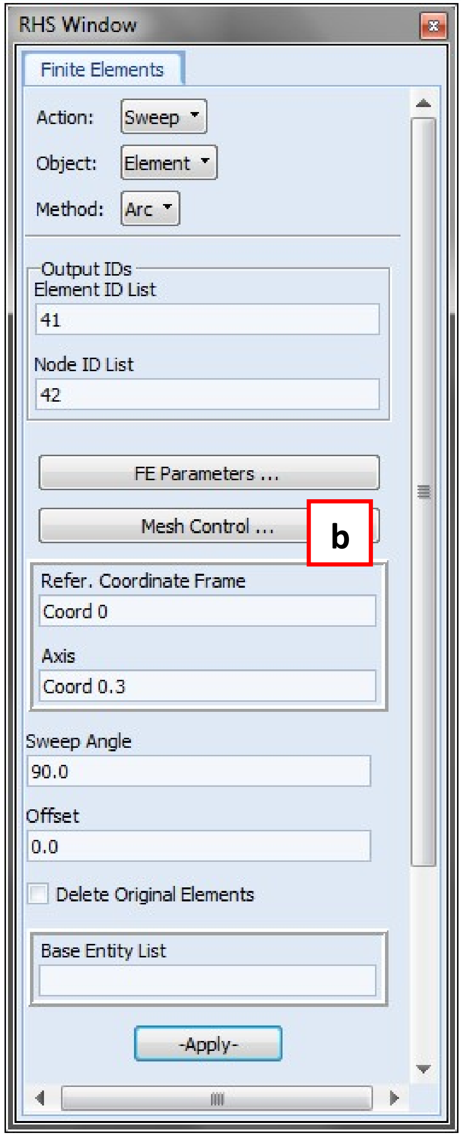
HUMAN CAPITAL
NATIONAL COHESION STRATEGY

EUROPEAN UNION
EUROPEAN
SOCIAL FUND

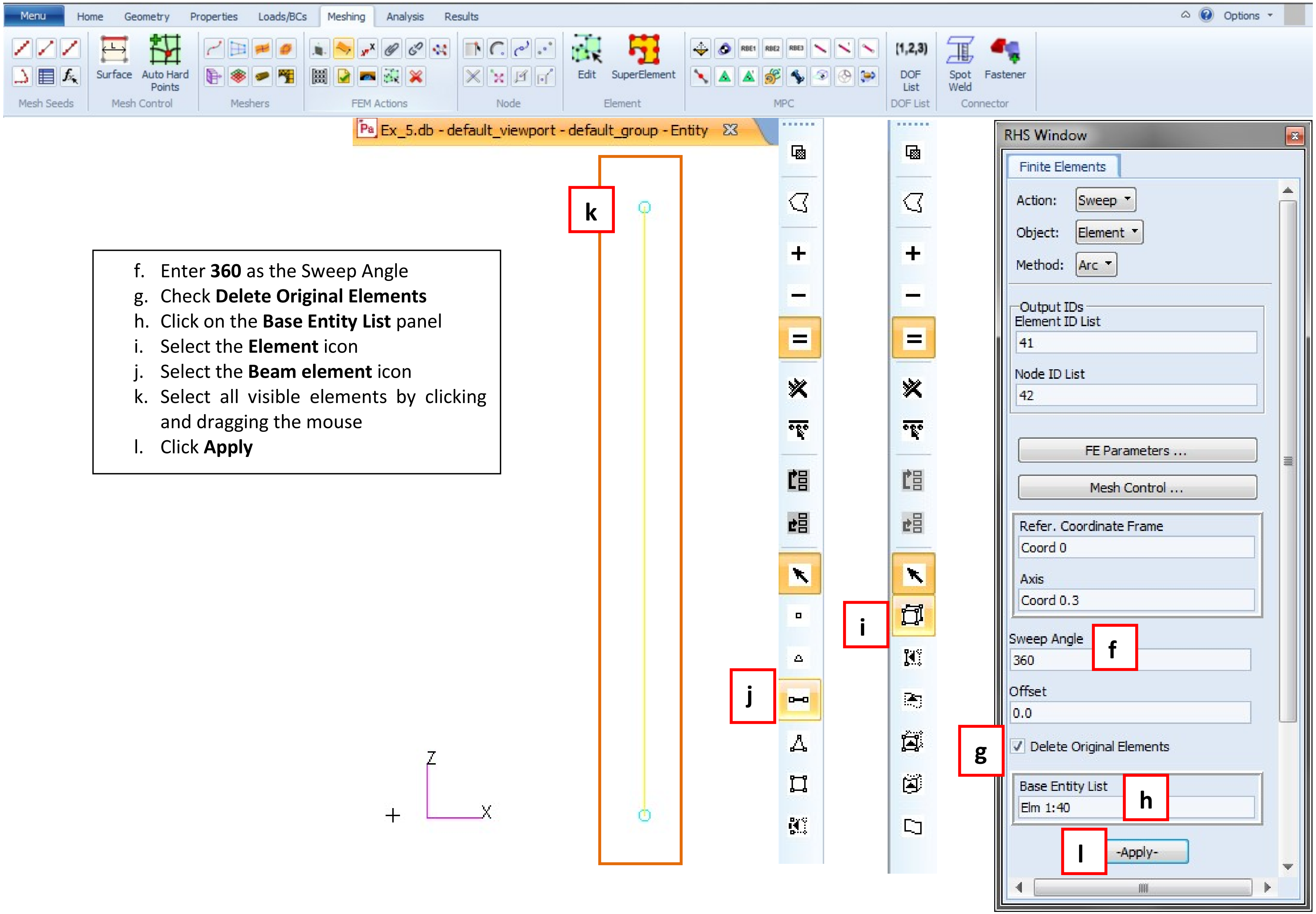
Sweep the elements:

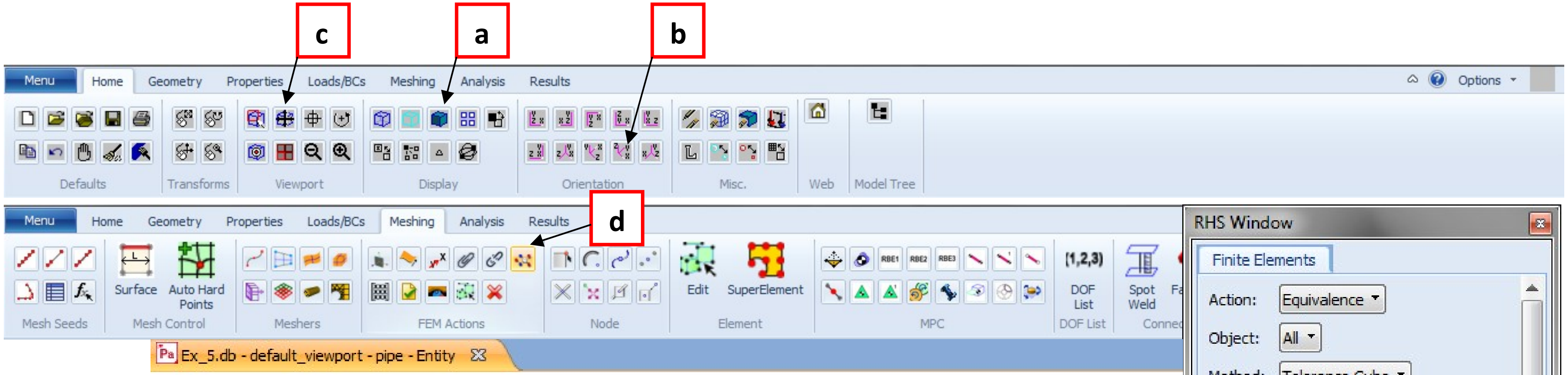
a. Meshing: **Sweep** icon (FEM Actions tab)
b. Click **Mesh Control…**
c. Select **Element Length (L)**
d. Enter **1** as the Length
e. Click **OK**





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

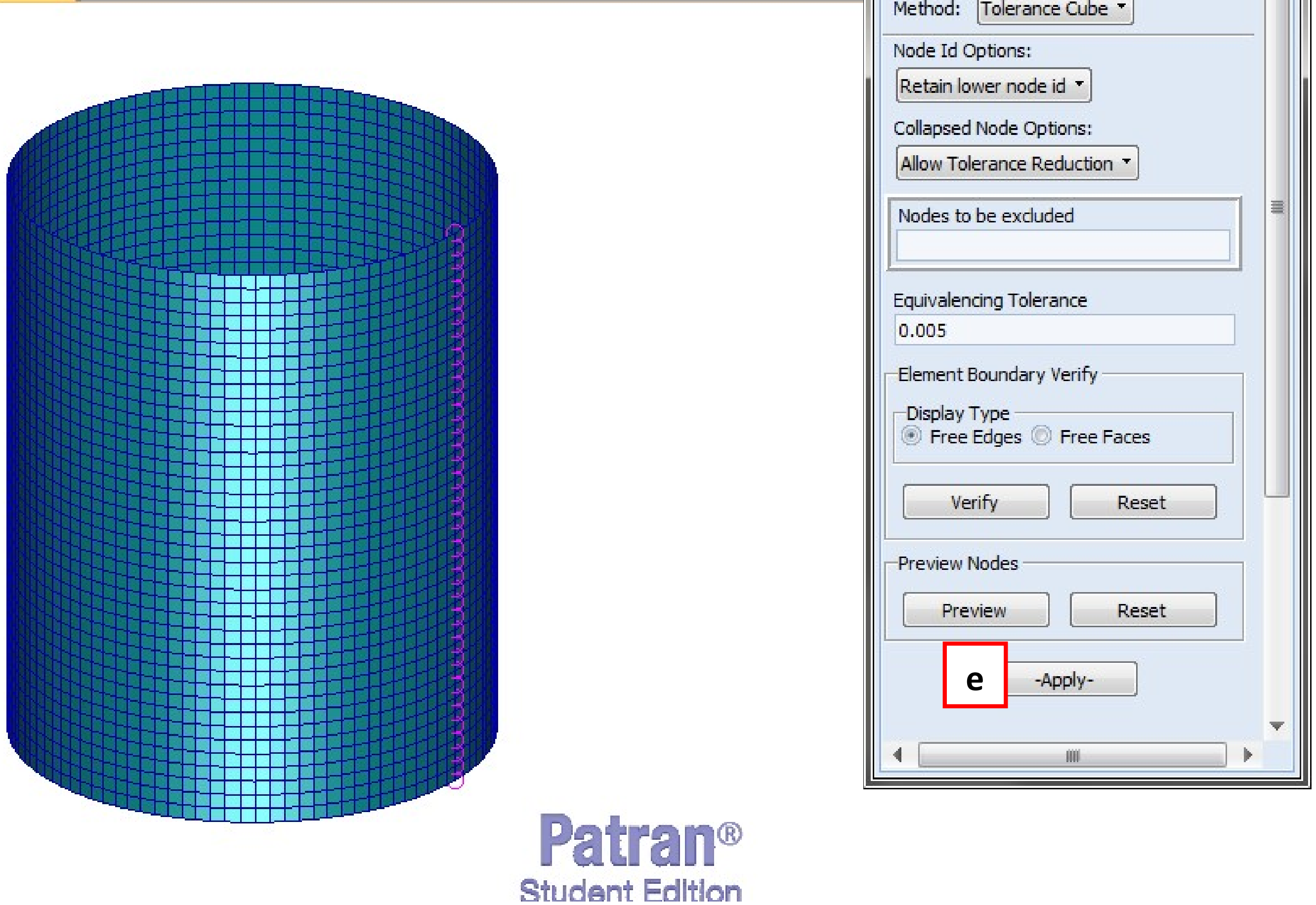


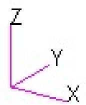


a. Click on the **Smooth shaded** icon
b. Click on the **Iso 3 View** icon
c. Click on the **Fit view** icon

Delete duplicate nodes:

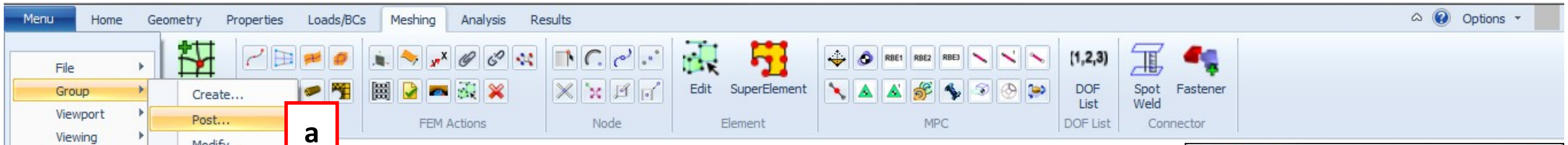
d. Meshing: **Equivalence** icon *(FEM Actions tab)*
e. Click **Apply**

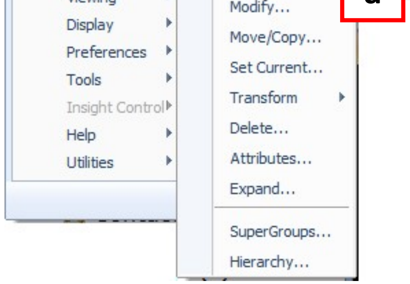




HUMAN CAPITAL
NATIONAL COHESION STRATEGY

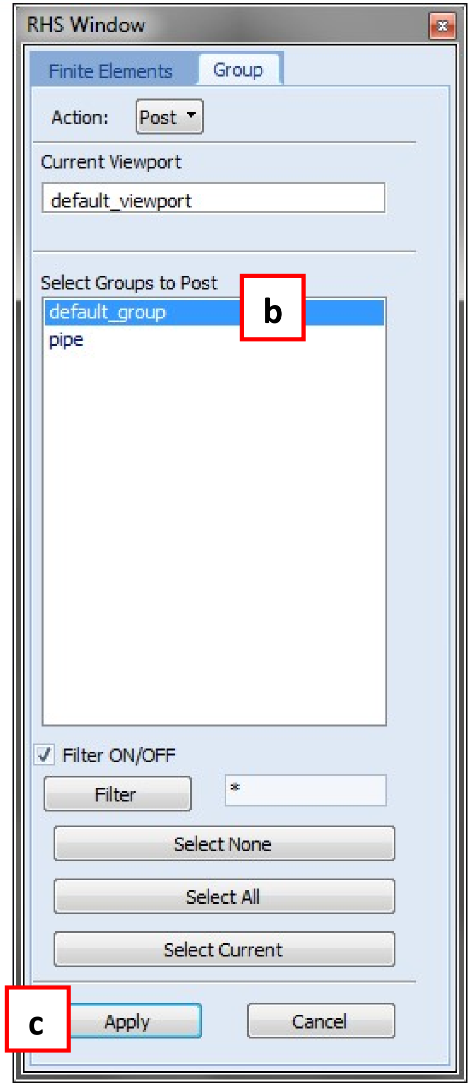
EUROPEAN UNION
EUROPEAN
SOCIAL FUND

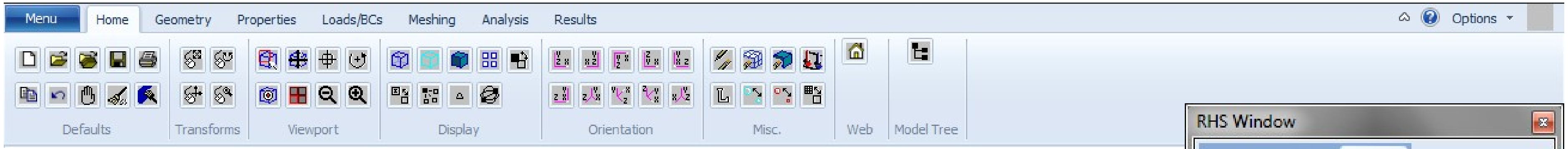




Post only the **default_group**:

a. **Group / Post…**
b. Select **default_group**
c. Click **Apply**

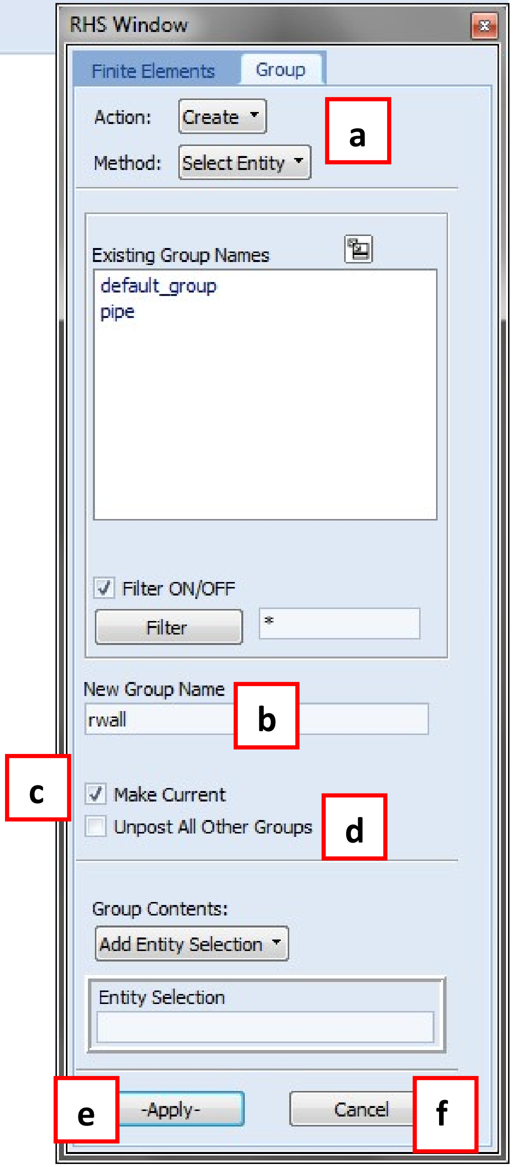


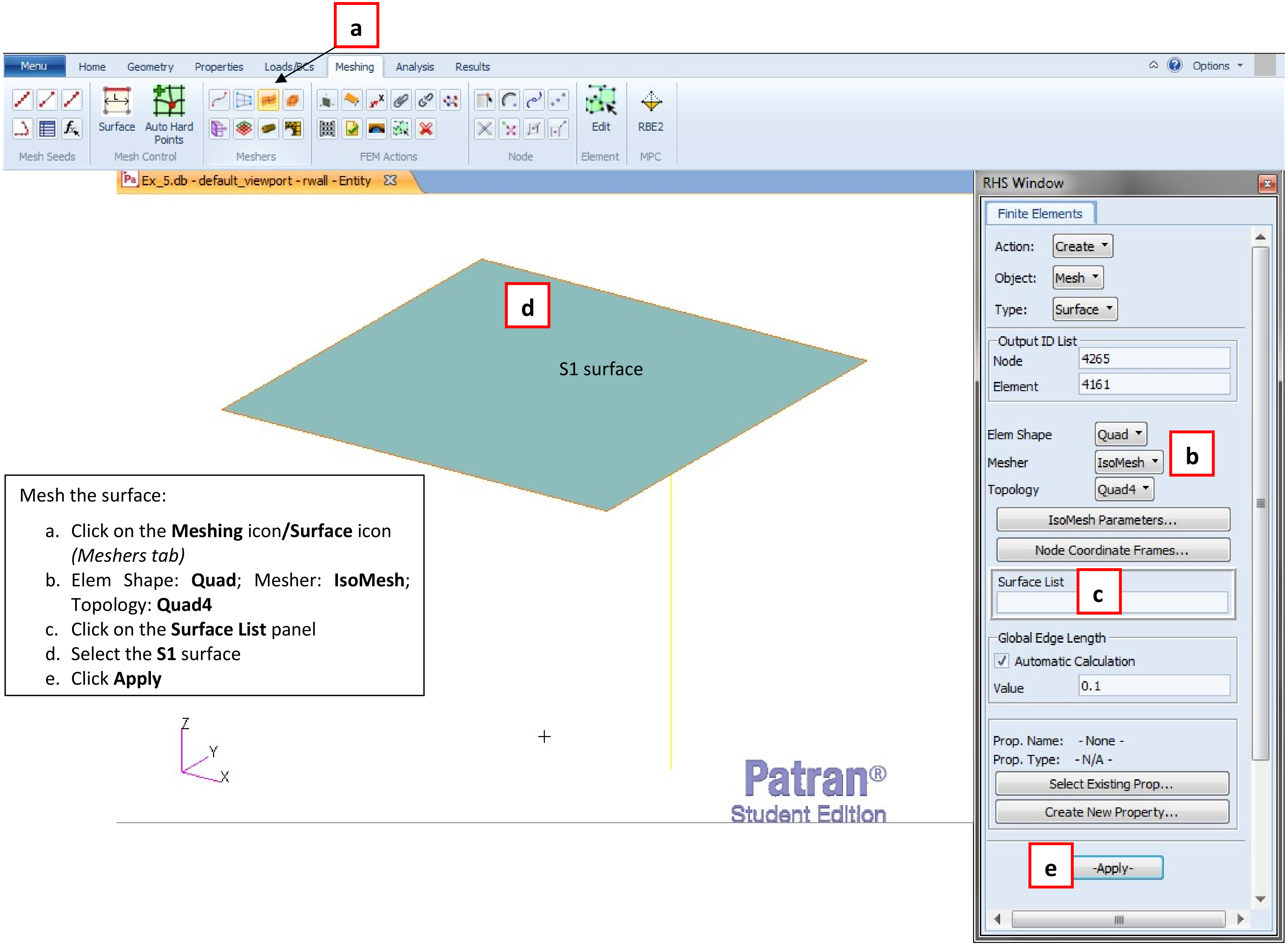


Create one more group:

a. Change *Action* to **Create**
b. Enter **rwall** as the New Group Name
c. Check **Make Current**
d. Uncheck **Unpost All Other Groups**
e. Click **Apply**
f. Click **Cancel**

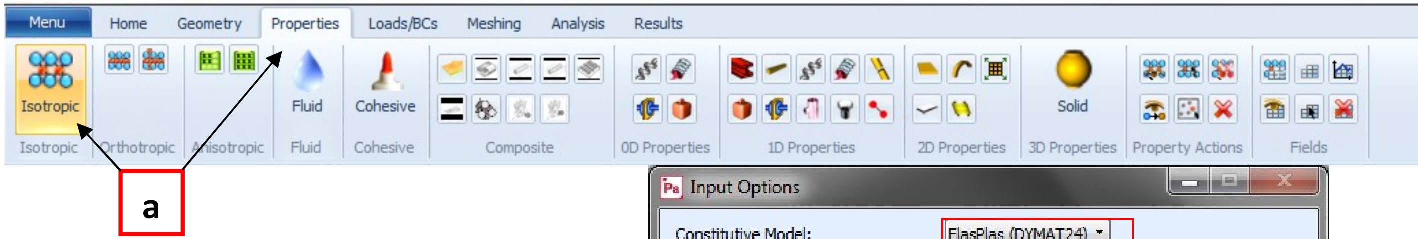
<u>Remark</u>: Mesh of the moving rigid wall will be assigned to this group.





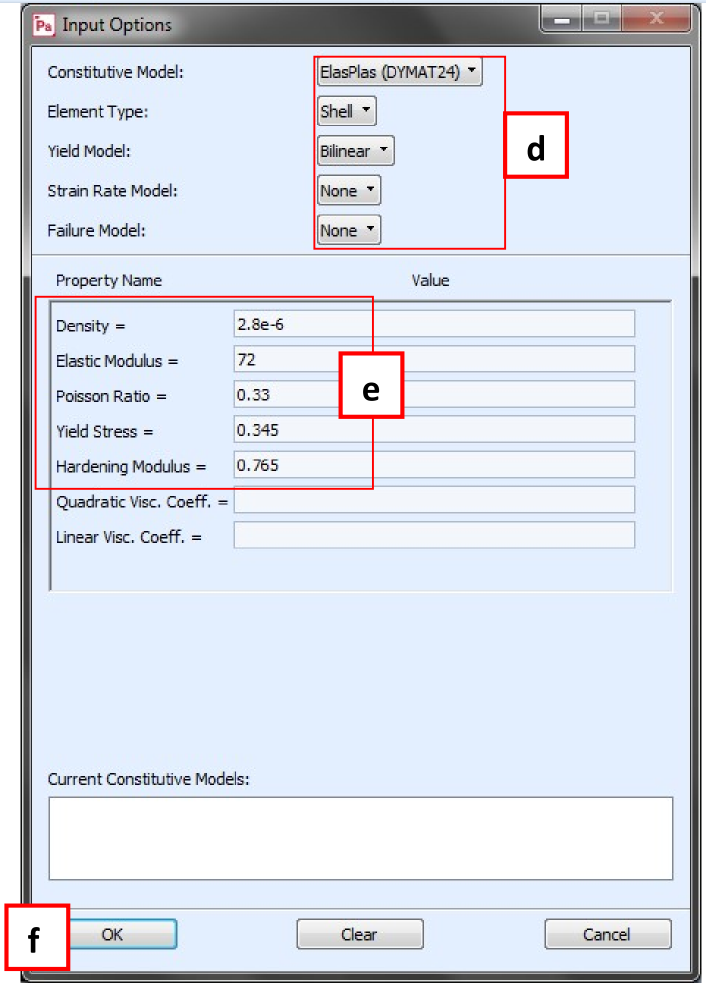
Mesh the surface:

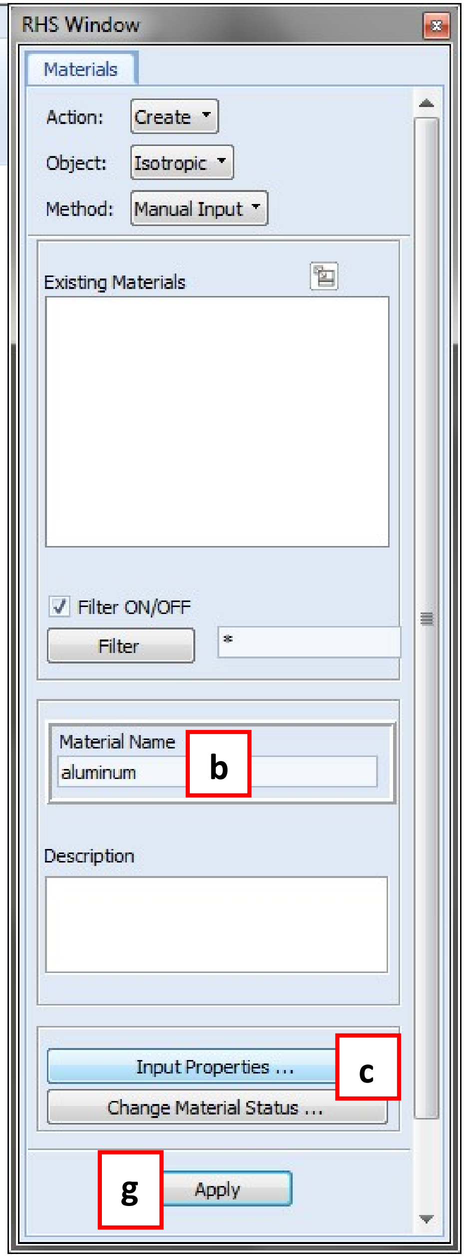
a. Click on the **Meshing** icon/**Surface** icon *(Meshers tab)*
b. Elem Shape: **Quad**; Mesher: **IsoMesh**; Topology: **Quad4**
c. Click on the **Surface List** panel
d. Select the **S1** surface
e. Click **Apply**



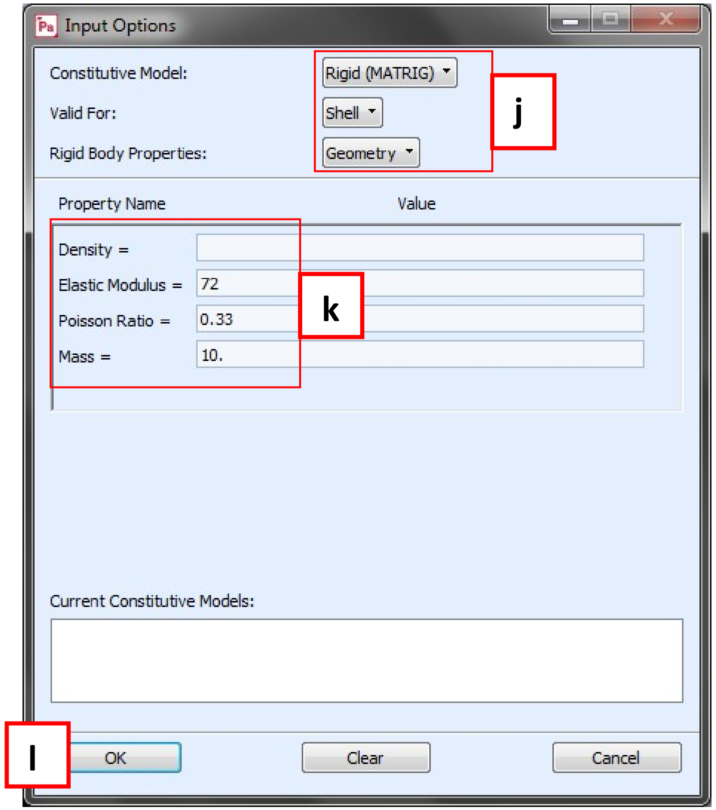
Define materials:

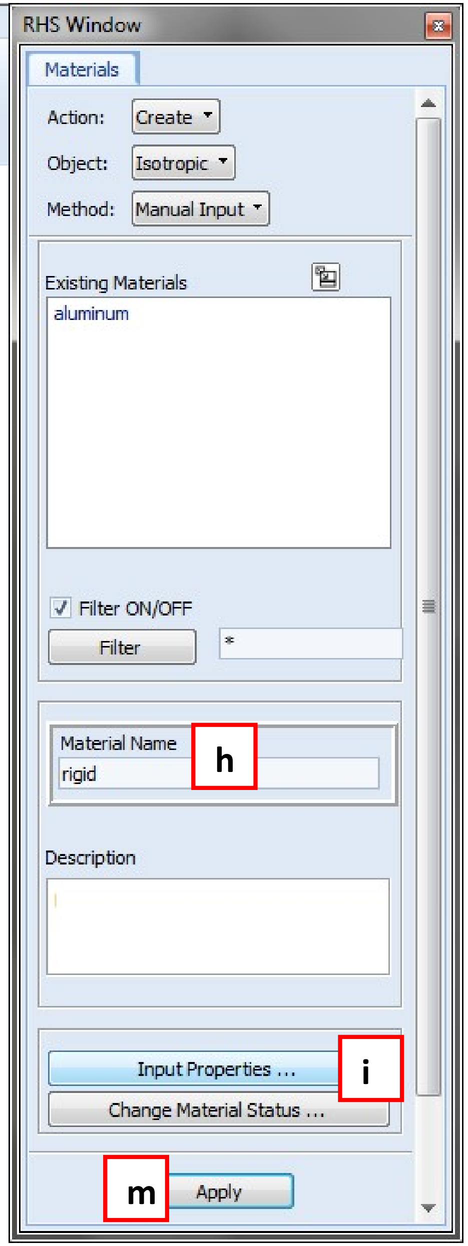
a. Click on the Click on the **Properties** icon/**Isotropic** icon
b. Enter **aluminum** as the Material Name
c. Click **Input Properties…**
d. Constitutive Model: **ElasPlas (DYMAT24)**; Element Type: **Shell**; Yield Model: **Bilinear**; Strain Rate Model & Failure Model: **None**
e. Enter **2.8e-6** as Density, **72** as Elastic Modulus, **0.33** as Poisson Ratio, **0.345** as Yield Stress and **0.765** as Hardening Modulus
f. Click **OK**
g. Click **Apply**

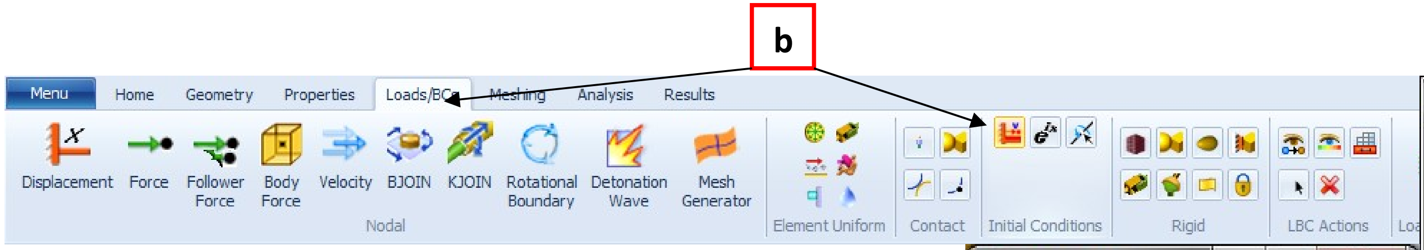




h. Enter **rigid** as the Material Name
i. Click **Input Properties…**
j. Constitutive Model: **Rigid (MATRIG)**; Valid For: **Shell**; Rigid Body Properties: **Geometry**
k. Enter **72** as Elastic Modulus, **0.33** as Poisson Ratio and **10** as Mass (of the wall)
l. Click **OK**
m. Click **Apply**



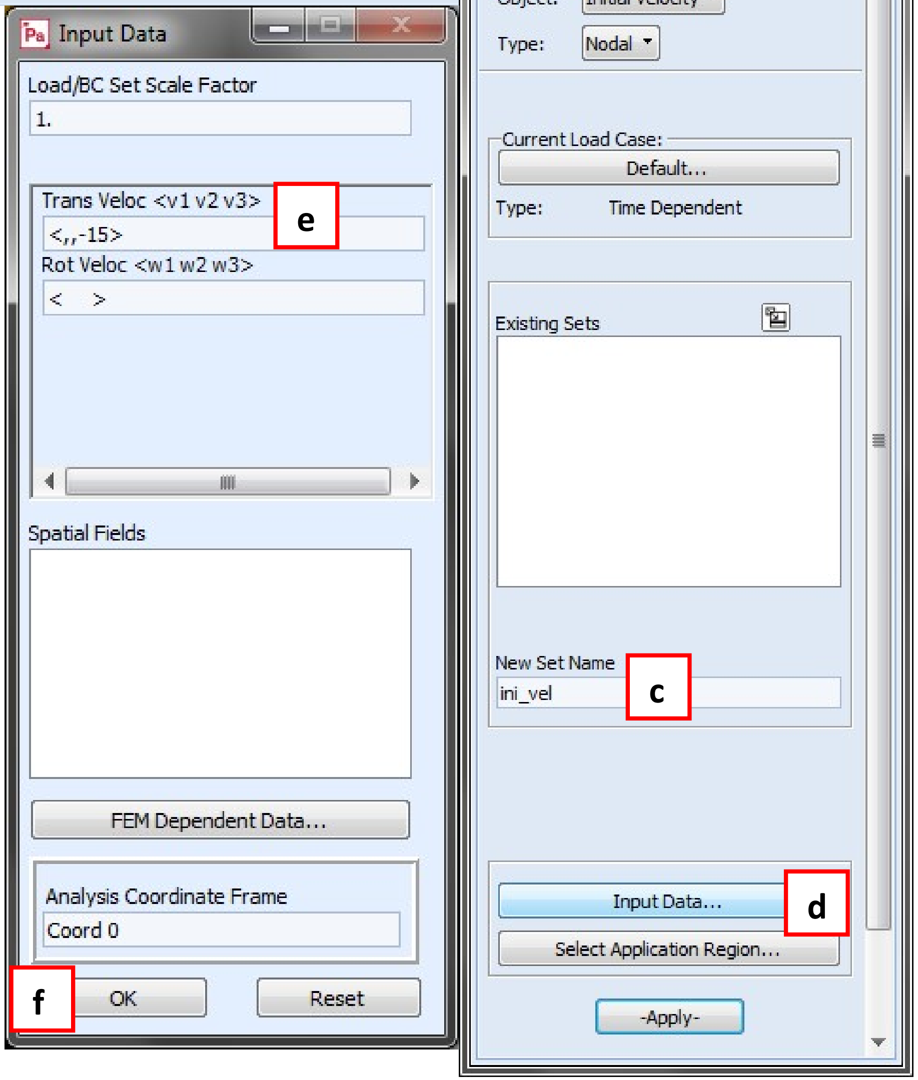


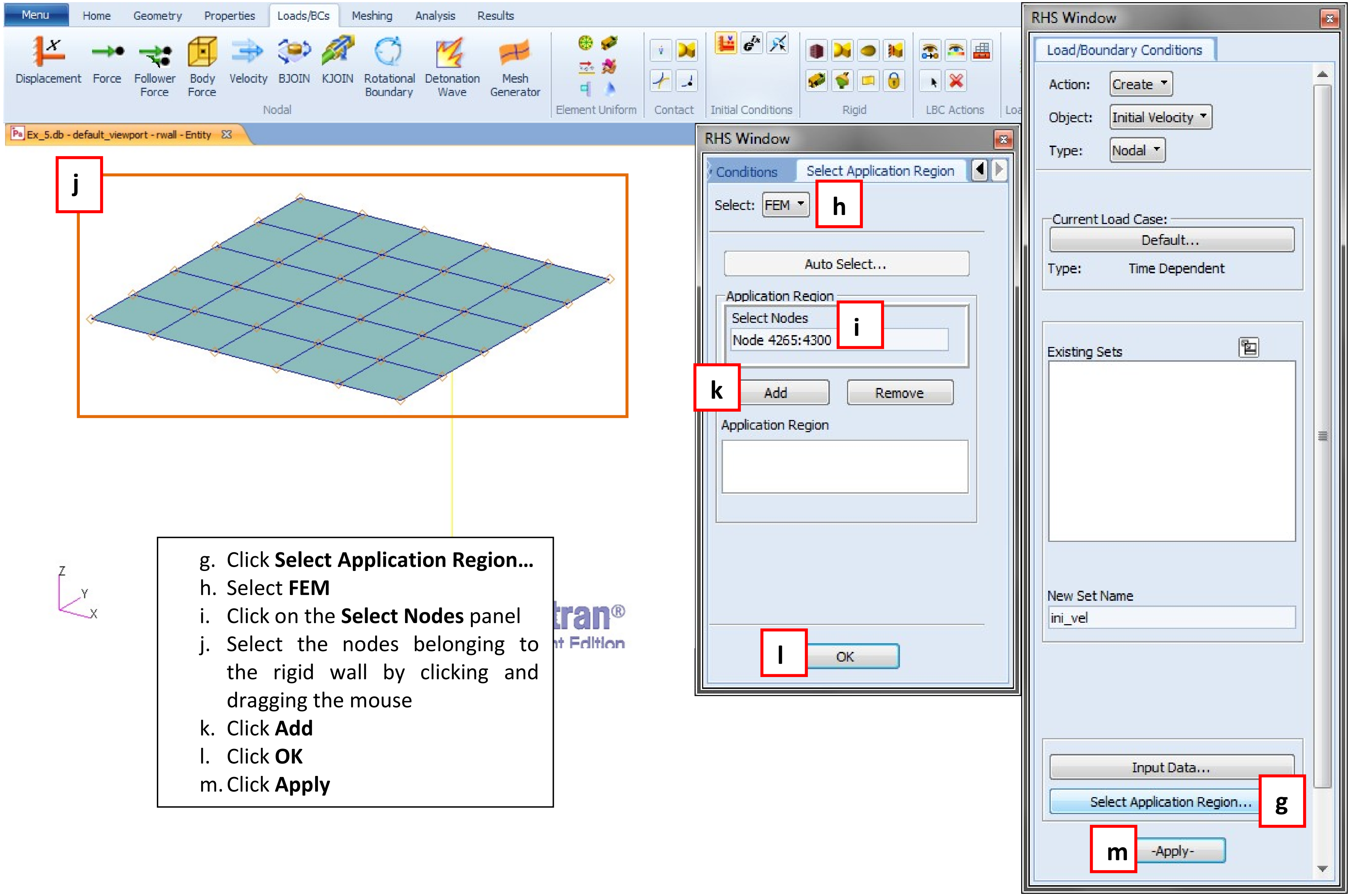


a. Post only the **rwall** group: **Menu/Group/Post.../**select only **rwall**

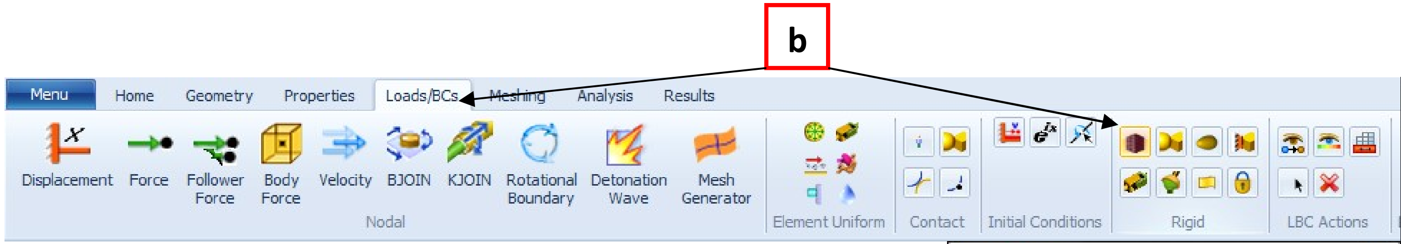
Apply initial velocity to the rigid wall:

b. Click on the **Loads/BCs** icon**/Initial Velocity** icon *(Initial Conditions tab)*
c. Enter **ini_vel** as the New Set Name
d. Click **Input Data...**
e. Enter **<,,-15>** for the Trans Veloc
f. Click **OK**





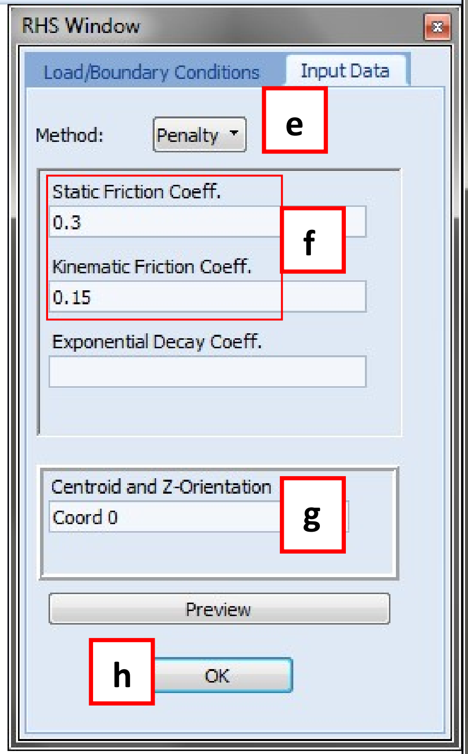
g. Click **Select Application Region…**
h. Select **FEM**
i. Click on the **Select Nodes** panel
j. Select the nodes belonging to the rigid wall by clicking and dragging the mouse
k. Click **Add**
l. Click **OK**
m. Click **Apply**

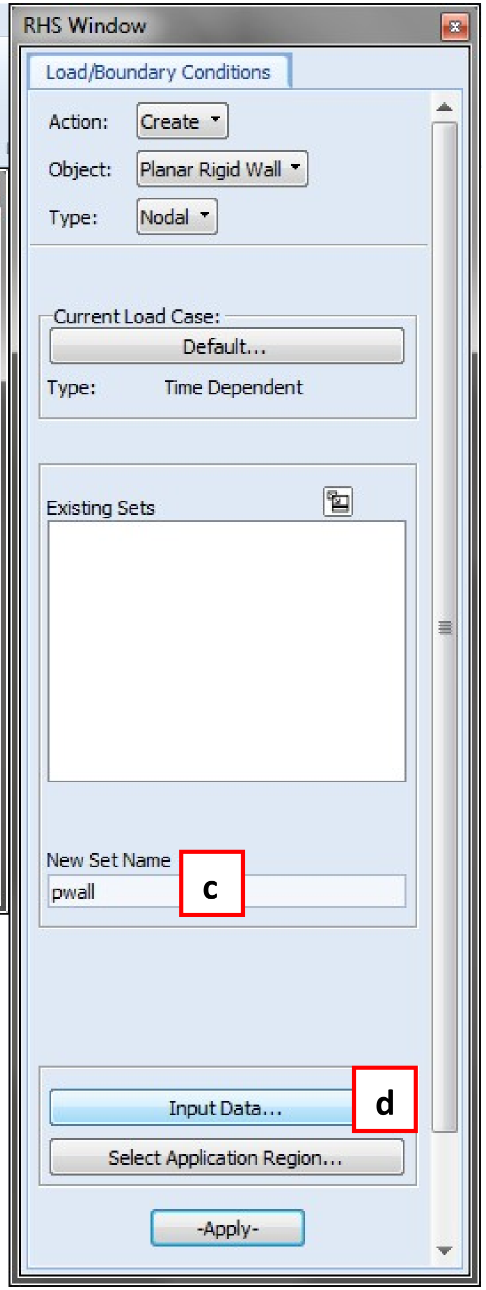


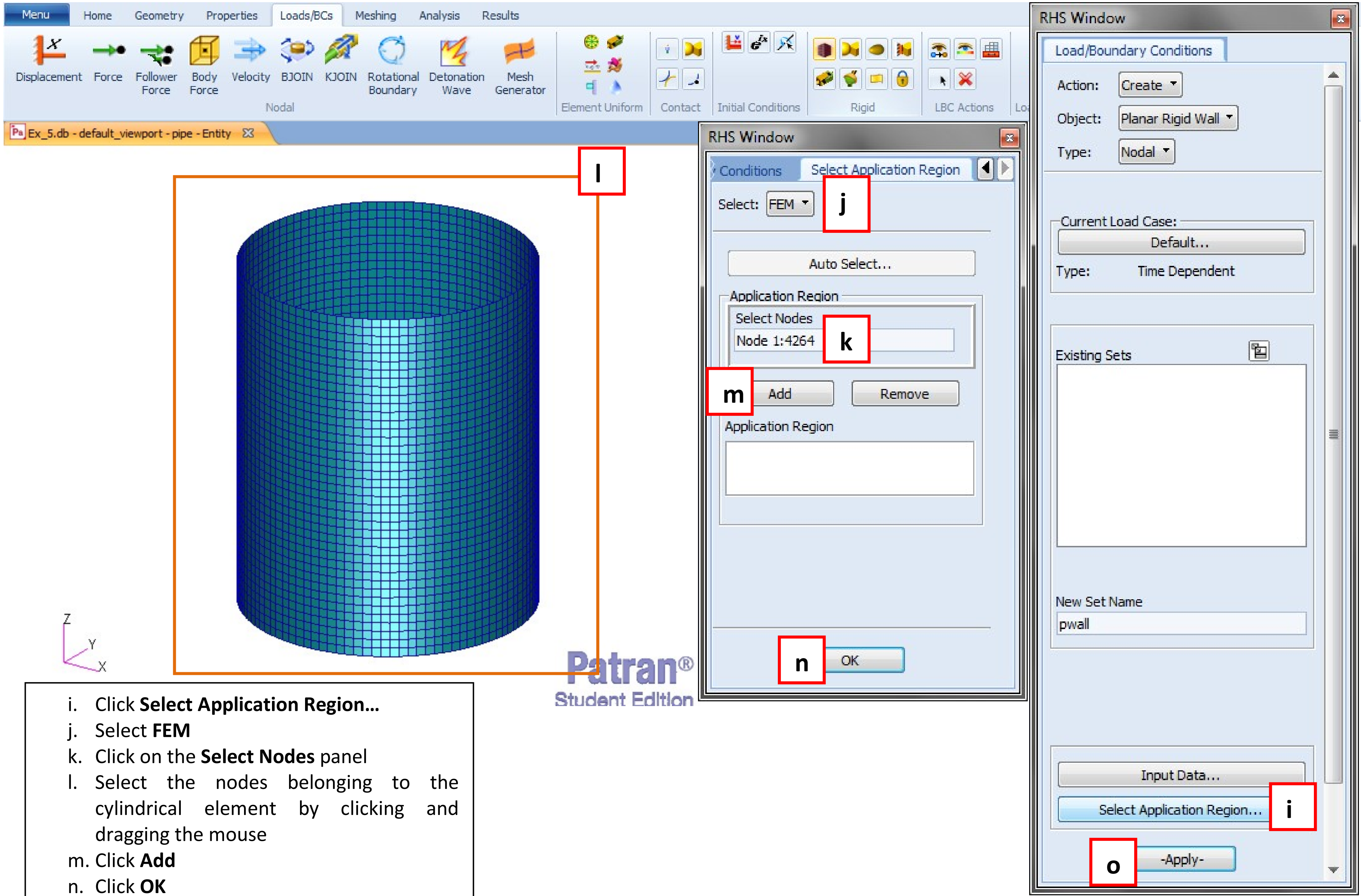
a. Post only the **pipe** group
   Menu/Group/Post.../select only **pipe**

Define a planar rigid wall which will be used as the bottom boundary/barrier for the cylindrical element:

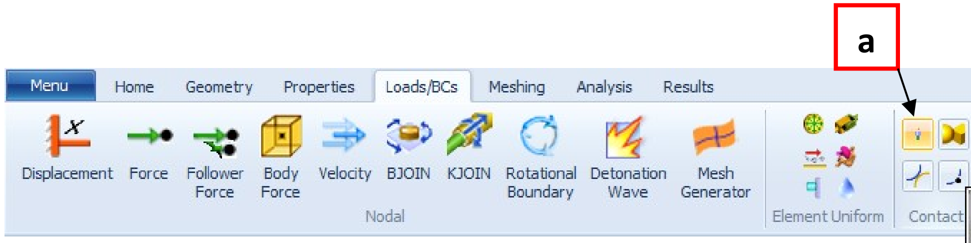
b. Click on the **Loads/BCs** icon**/Planar Rigid Wall** icon *(Rigid tab)*
c. Enter **pwall** as the New Set Name
d. Click **Input Data…**
e. Select **Penalty**
f. Enter **0.3** as the Static Friction Coefficient and **0.15** as the Kinematic Friction Coefficient
g. Centroid and Z-Orientation: **Coord 0**
h. Click **OK**





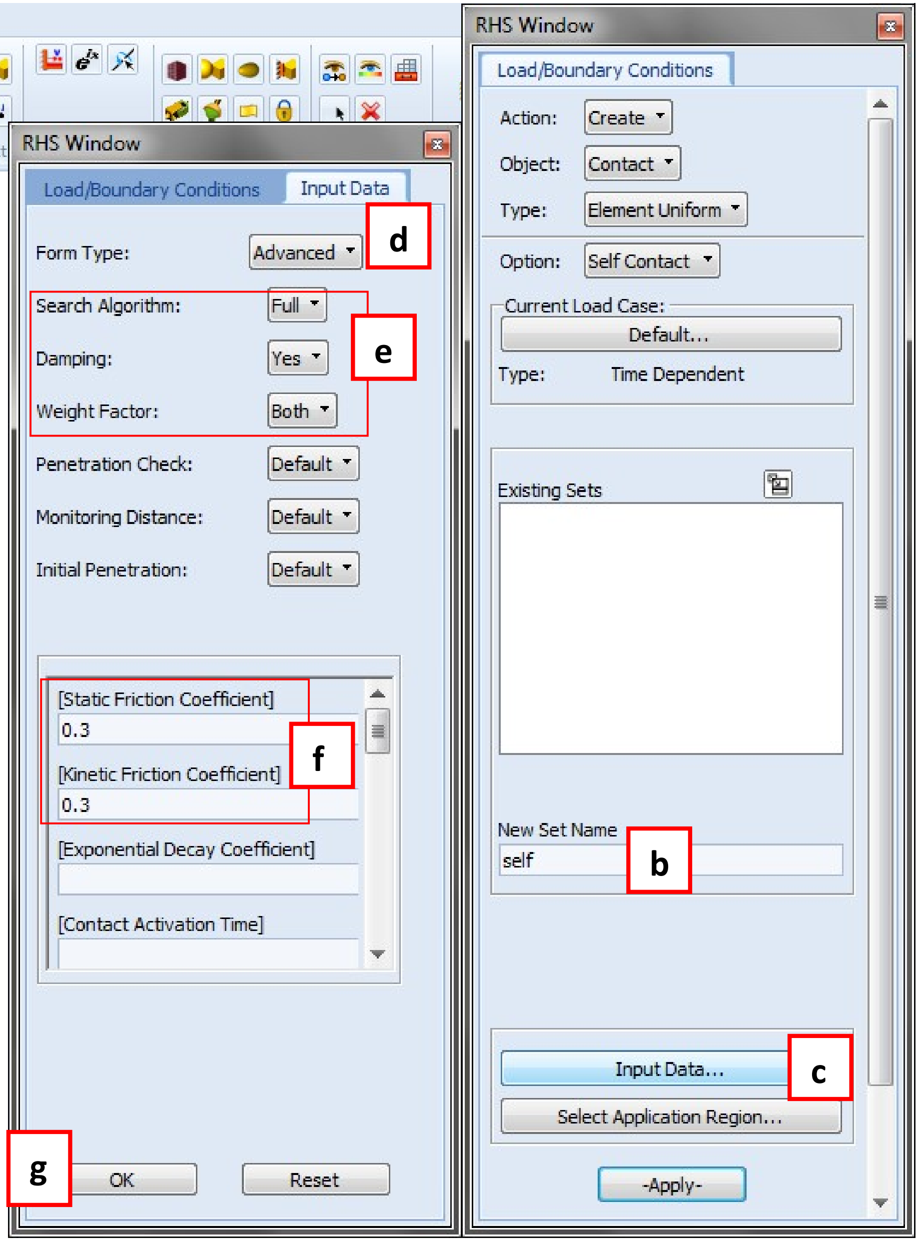


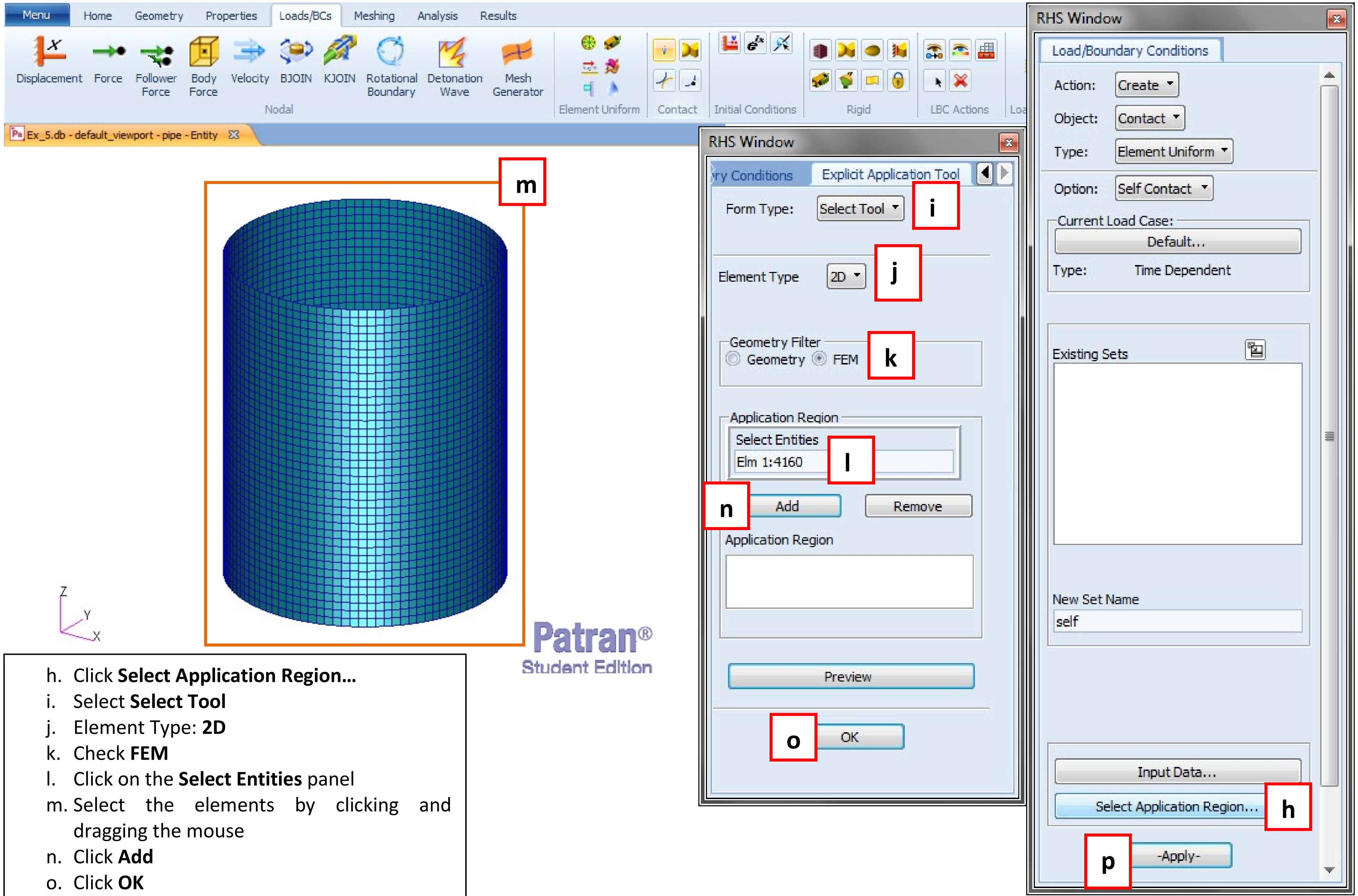
i. Click **Select Application Region…**
j. Select **FEM**
k. Click on the **Select Nodes** panel
l. Select the nodes belonging to the cylindrical element by clicking and dragging the mouse
m. Click **Add**
n. Click **OK**
o. Click **Apply**



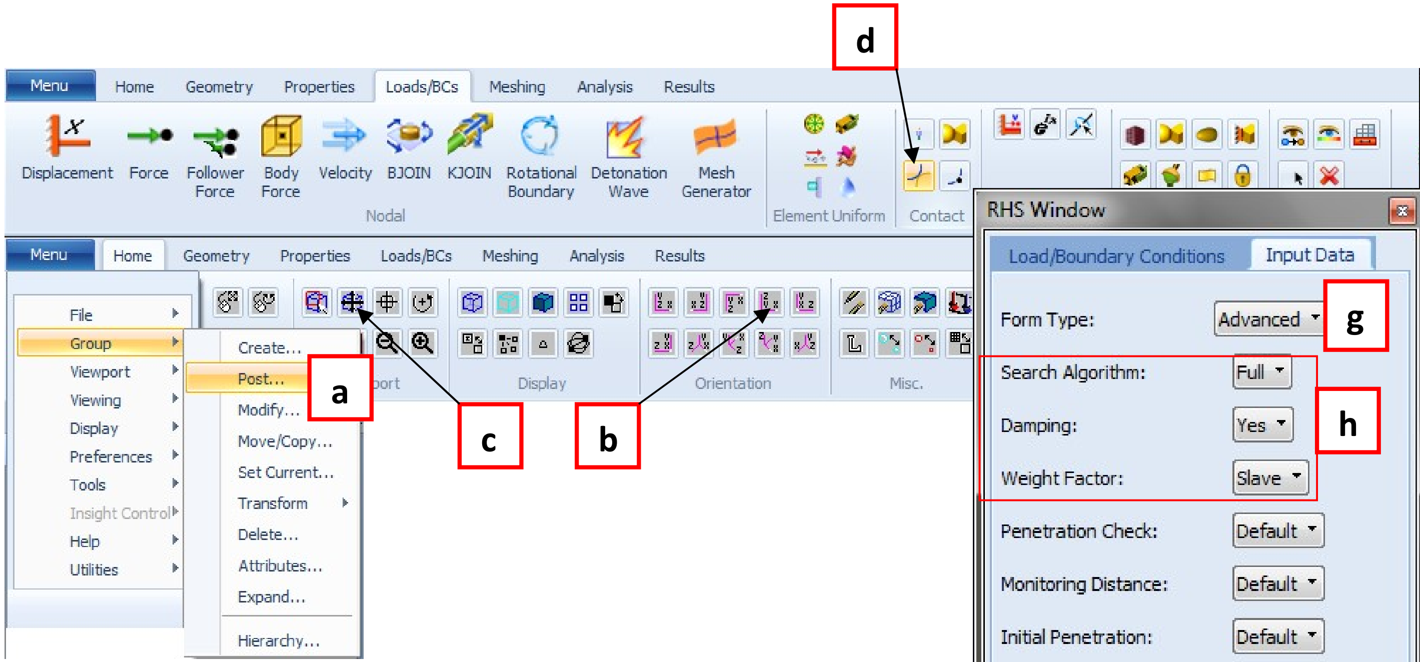
Define a self contact for the cylindrical element:

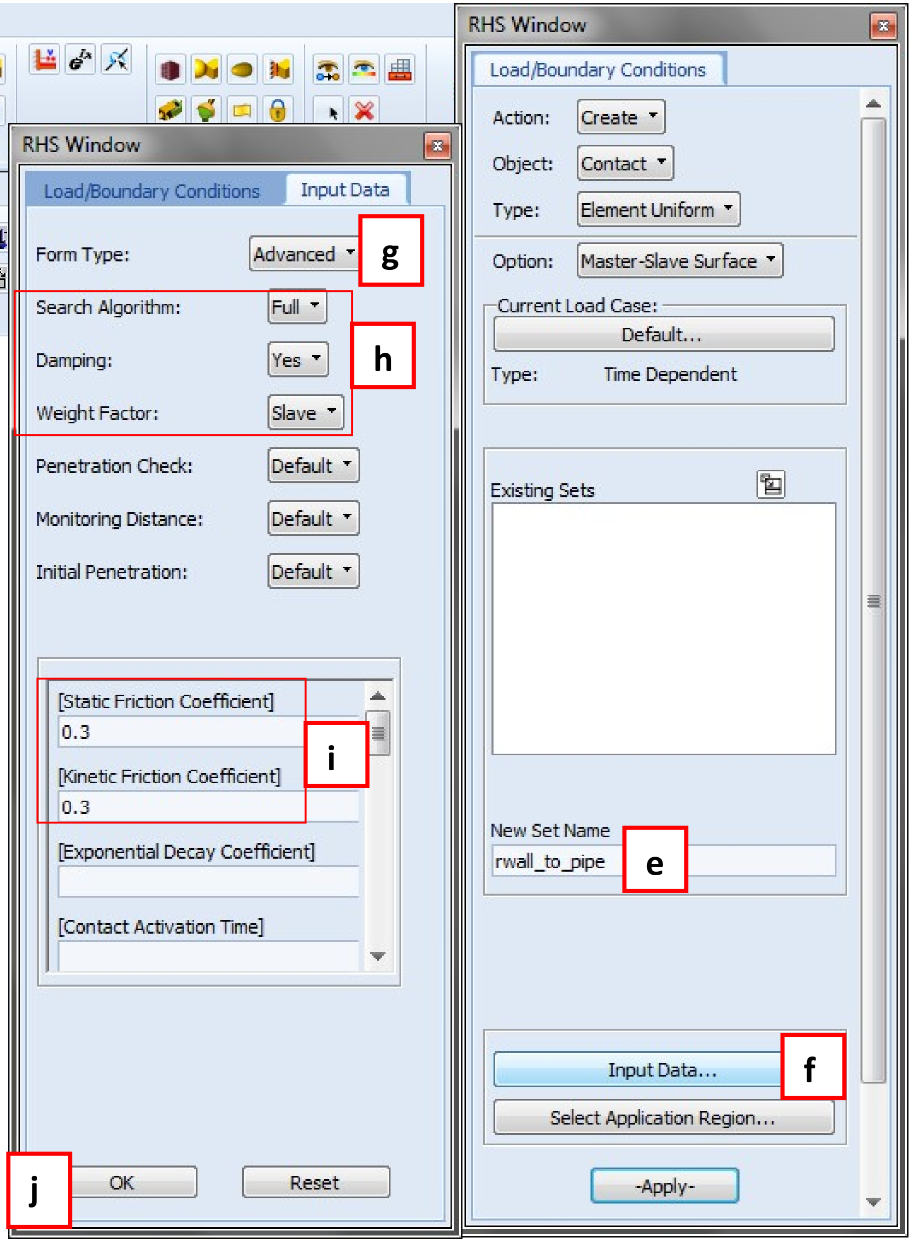
a. Loads/BCs: **Self Contact** icon *(Contact tab)*
b. Enter **self** as the New Set Name
c. Click **Input Data…**
d. Select **Advanced** as the Form Type
e. Search Algorithm: **Full**;
   Damping: **Yes**;
   Weight Factor: **Both**
f. Enter **0.3** as the Static and Kinematic Friction Coefficient
g. Click **OK**





h. Click **Select Application Region…**
i. Select **Select Tool**
j. Element Type: **2D**
k. Check **FEM**
l. Click on the **Select Entities** panel
m. Select the elements by clicking and dragging the mouse
n. Click **Add**
o. Click **OK**
p. Click **Apply**

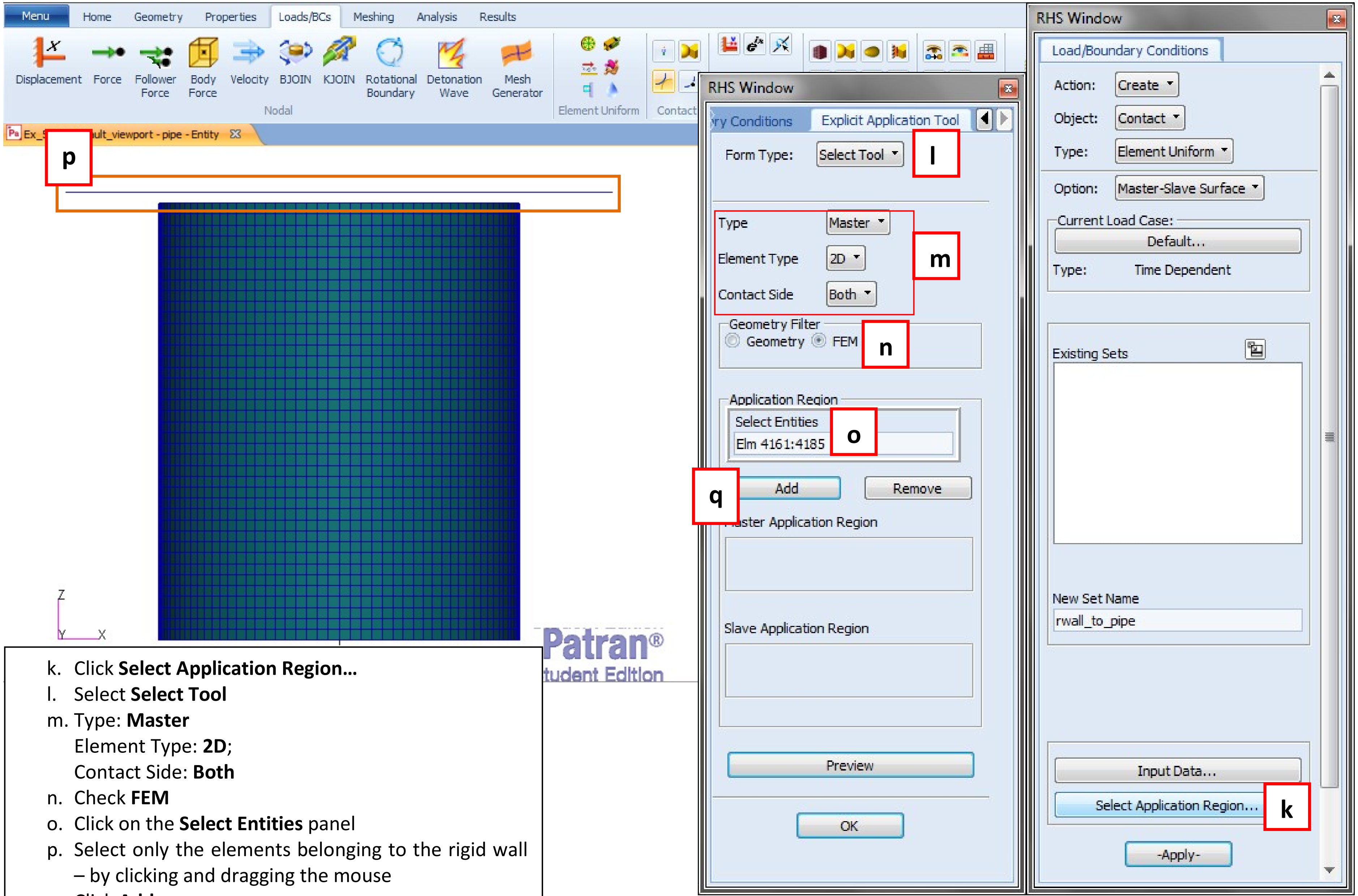




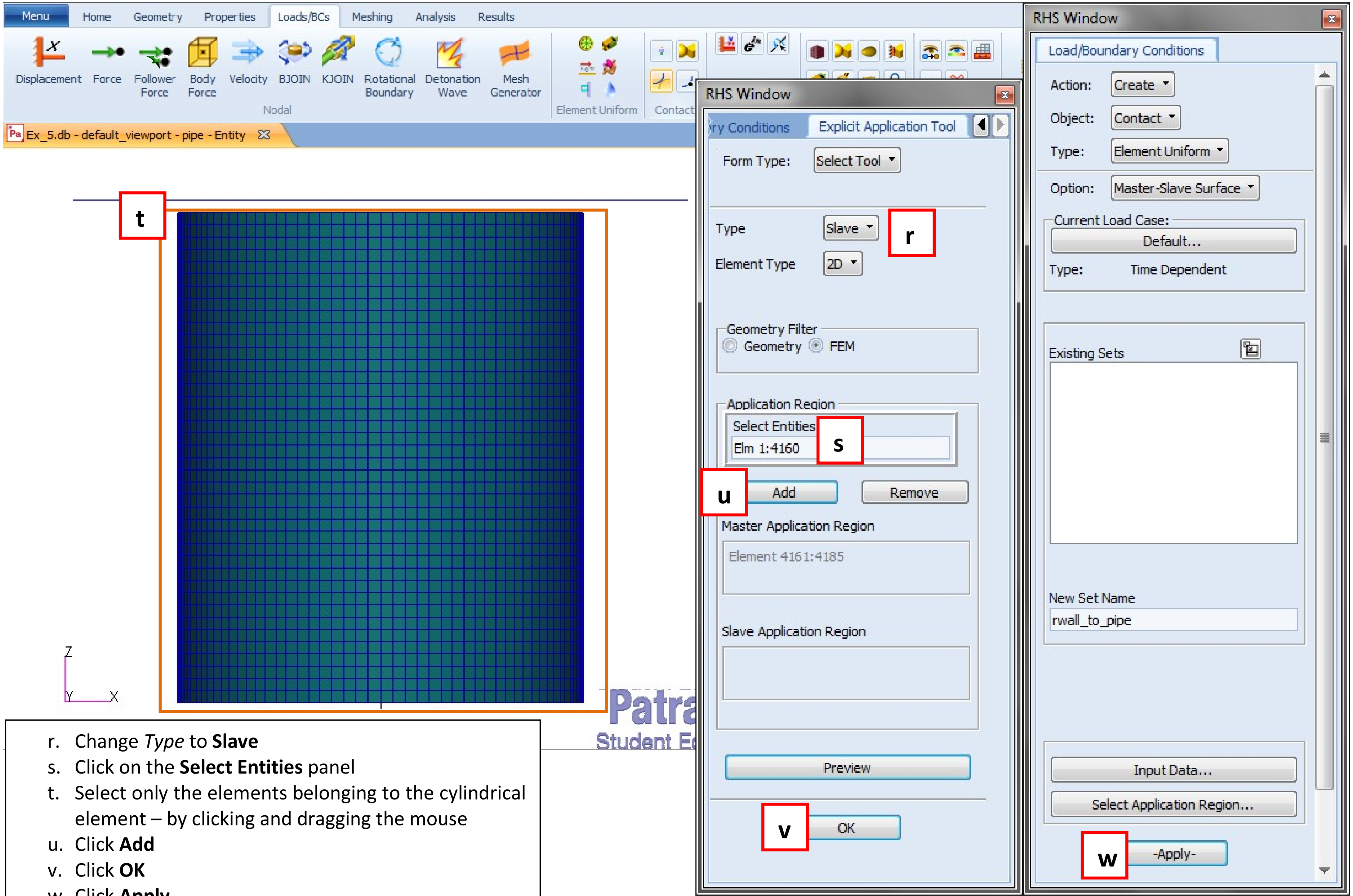
a. Post two groups: **rwall** and **pipe**:
   Menu/Group/Post.../Select **rwall** and **pipe**
b. Click on the **Bottom view** icon
c. Click on the **Fit view** icon

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

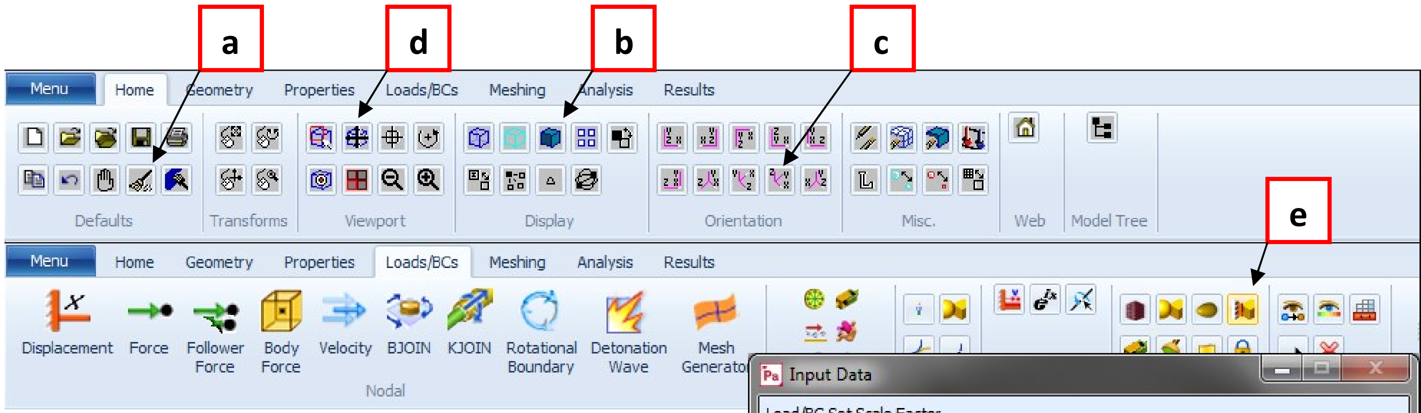
d. Loads/BCs: **Master-Slave Surface** icon *(Contact tab)*
e. Enter **rwall_to_pipe** as the New Set Name
f. Click **Input Data...**
g. Select **Advanced** as the Form Type
h. Search Algorithm: **Full**;
   Damping: **Yes**;
   Weight Factor: **Slave**
i. Enter **0.3** as the Static and Kinematic Friction Coefficient
j. Click **OK**



k. Click **Select Application Region...**
l. Select **Select Tool**
m. Type: **Master**
   Element Type: **2D**;
   Contact Side: **Both**
n. Check **FEM**
o. Click on the **Select Entities** panel
p. Select only the elements belonging to the rigid wall – by clicking and dragging the mouse
q. Click **Add**



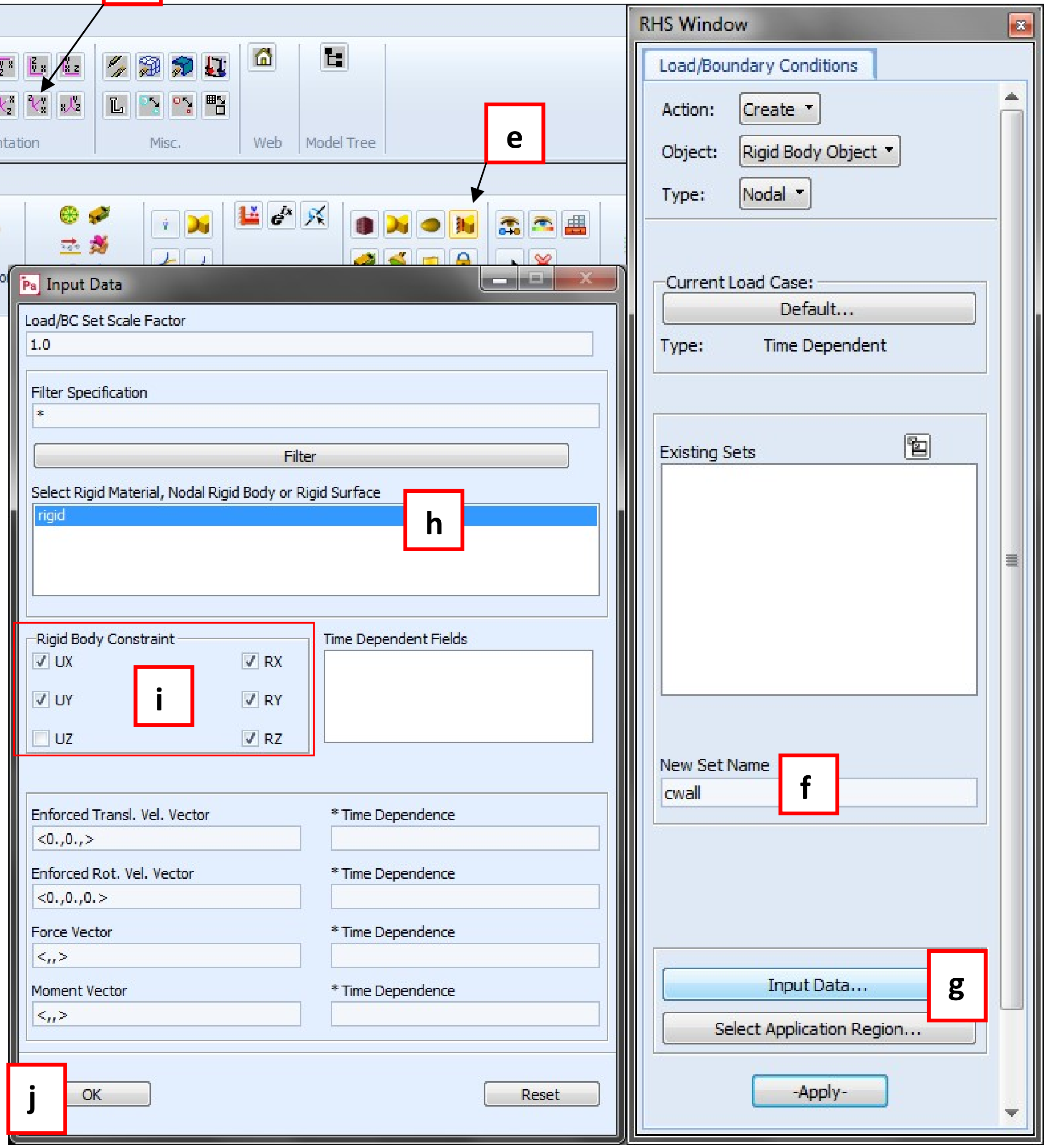
r. Change *Type* to **Slave**
s. Click on the **Select Entities** panel
t. Select only the elements belonging to the cylindrical element – by clicking and dragging the mouse
u. Click **Add**
v. Click **OK**
w. Click **Apply**

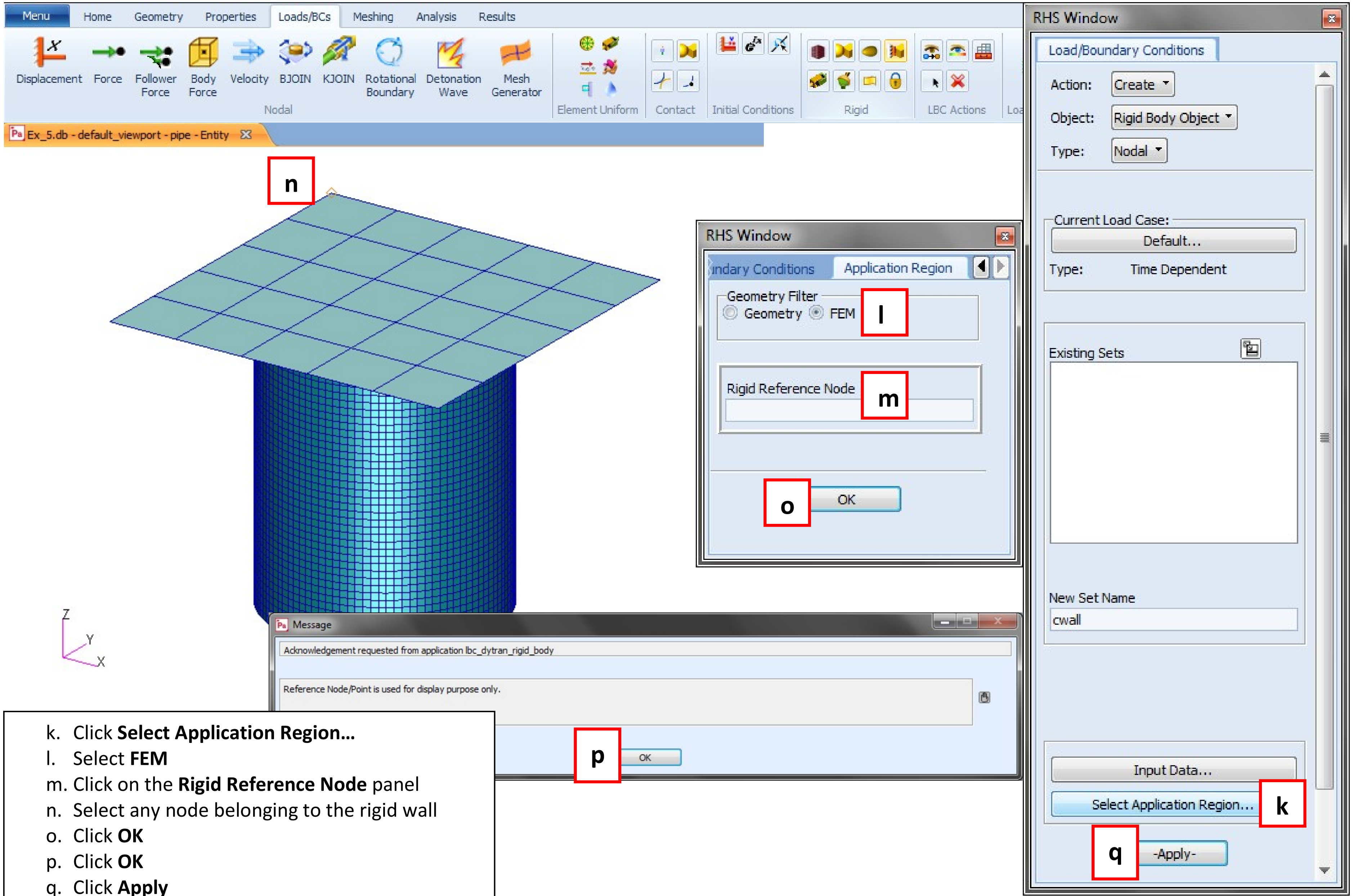


- a. Click on the **Reset graphics** icon
- b. Click on the **Smooth shaded** icon
- c. Click on the **Iso 3 View** icon
- d. Click on the **Fit view** icon

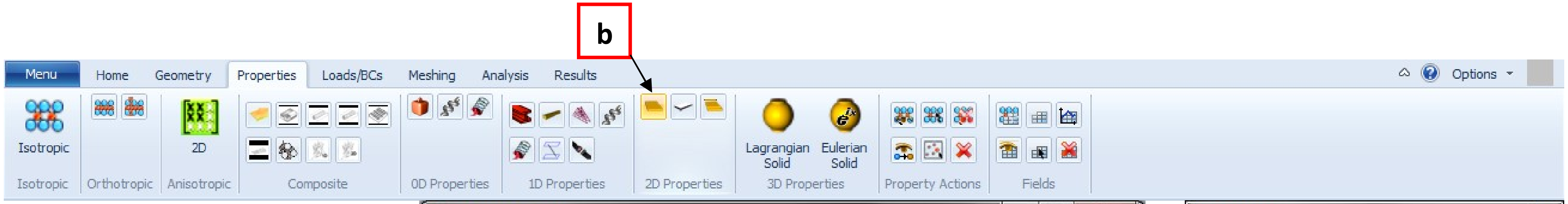
Apply the boundary conditions to the moving rigid wall:

- e. Loads/BCs: **Body Object** icon *(Rigid tab)*
- f. Enter **cwall** as the New Set Name
- g. Click **Input Data...**
- h. Select **rigid**
- i. By checking **UX**, **UY**, **RX**, **RY**, **RZ** select DOF's of the rigid wall which will be constrained during the analysis
- j. Click **OK**





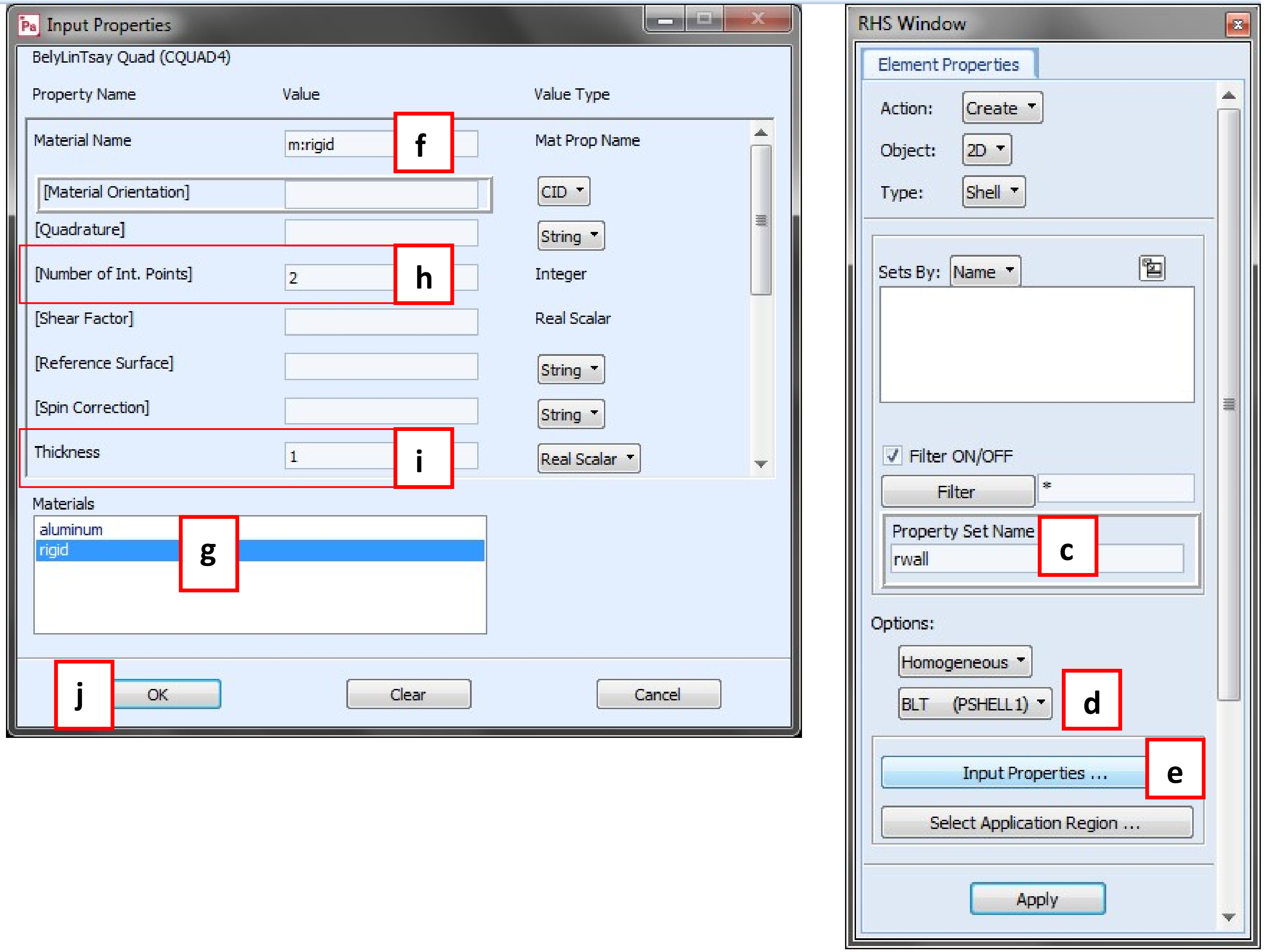
k. Click **Select Application Region…**
l. Select **FEM**
m. Click on the **Rigid Reference Node** panel
n. Select any node belonging to the rigid wall
o. Click **OK**
p. Click **OK**
q. Click **Apply**



a. Post only the **rwall** group

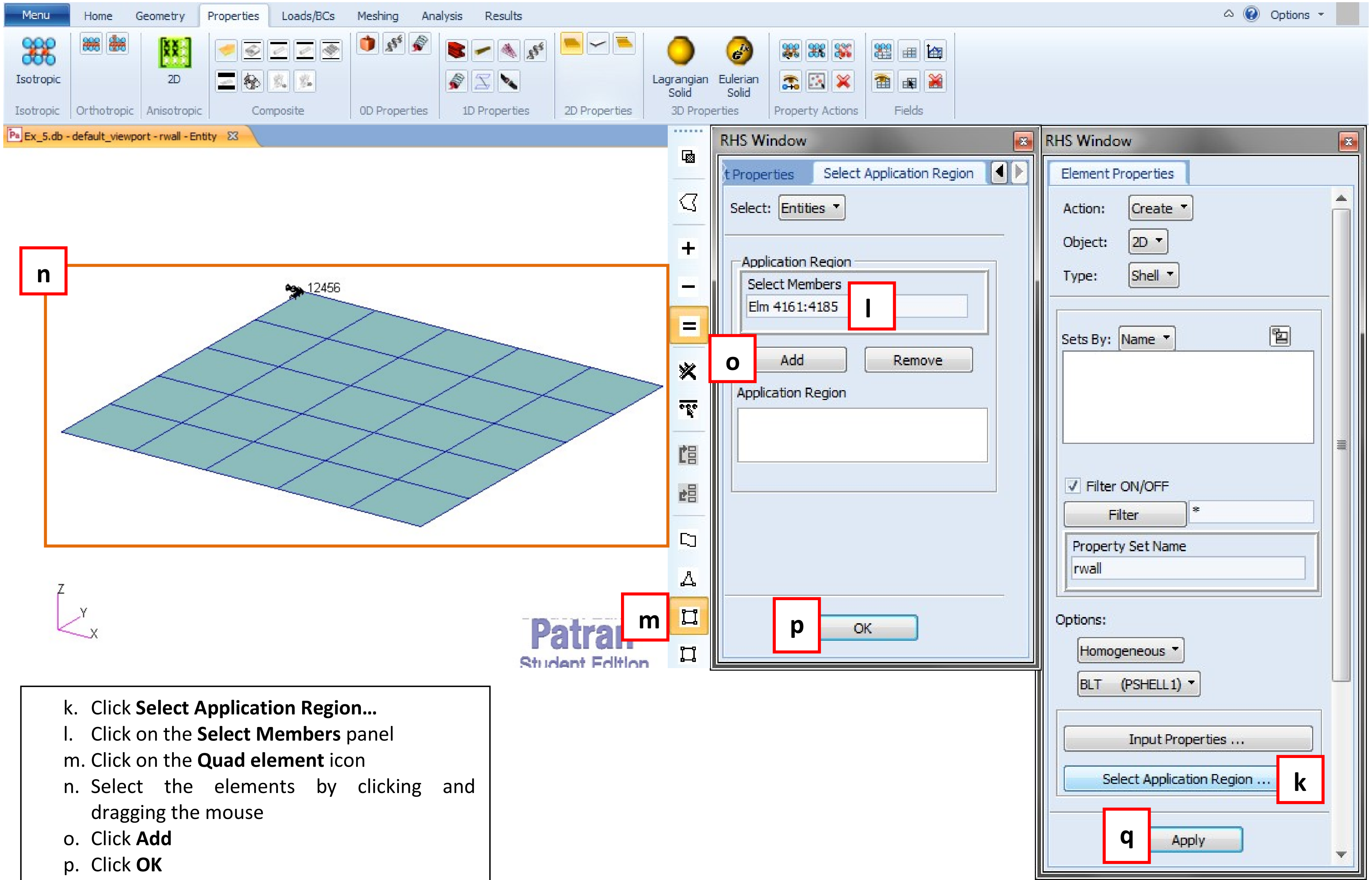
Assign the properties

b. Properties: **Shell** icon *(2D Properties tab)*
c. Enter **rwall** as the New Set Name
d. Select **BLT (PSHELL1)**
e. Click **Input Properties…**
f. Click on the **Material Name** panel
g. Select **rigid**
h. Enter **2** as the Number of Points
i. Enter **1** as the Thickness
j. Click **OK**



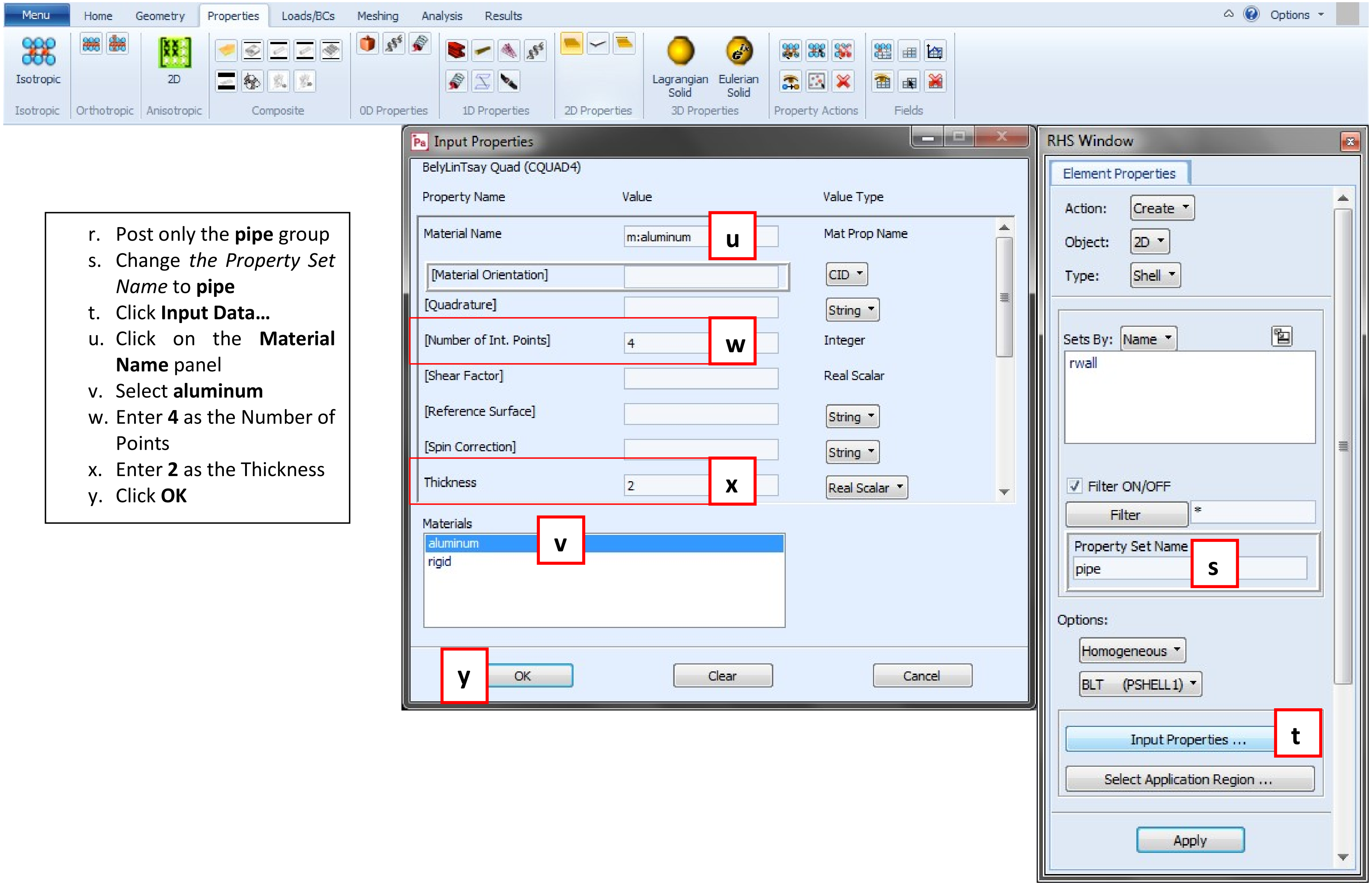
HUMAN CAPITAL
NATIONAL COHESION STRATEGY

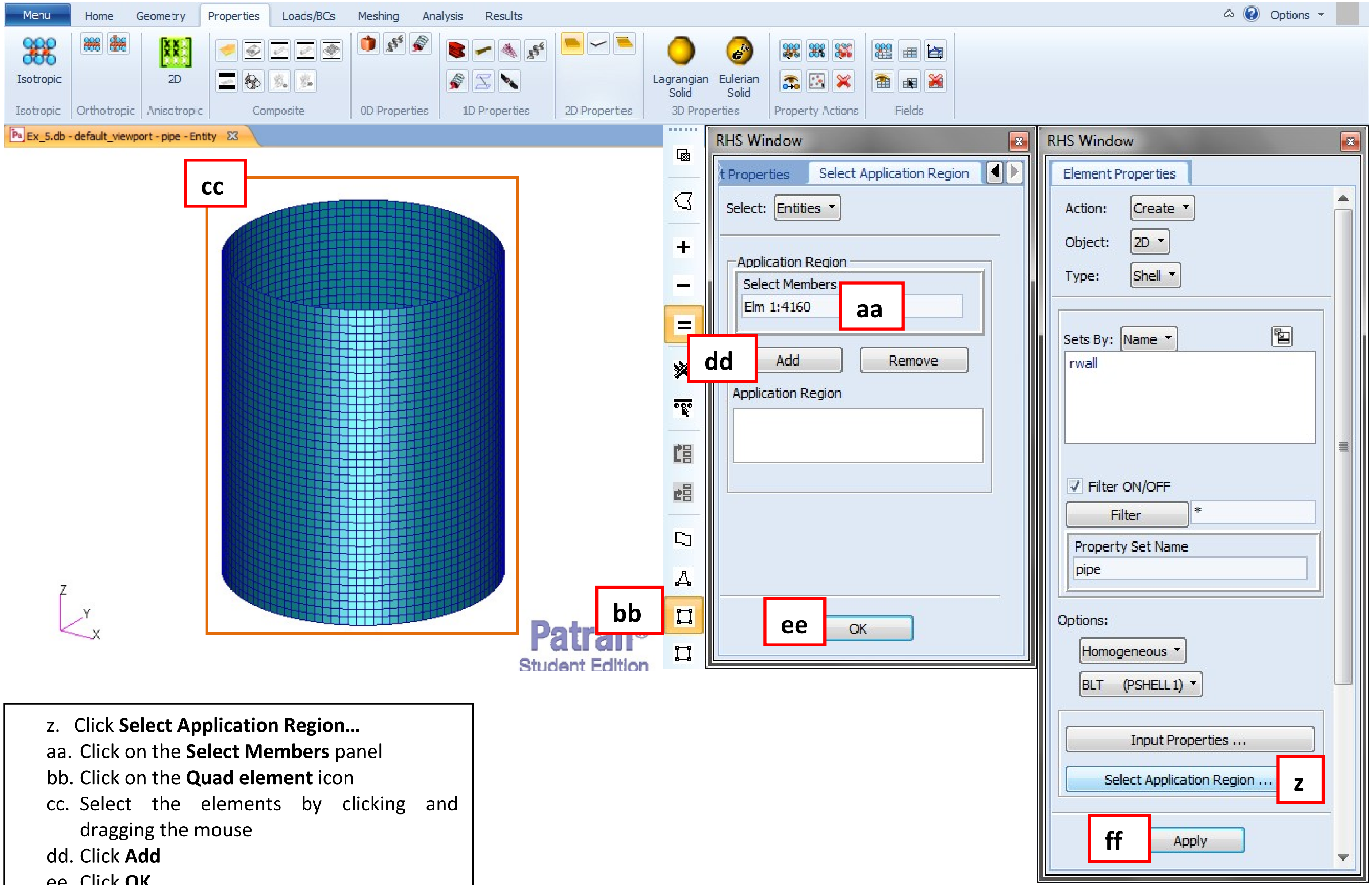
EUROPEAN UNION
EUROPEAN
SOCIAL FUND



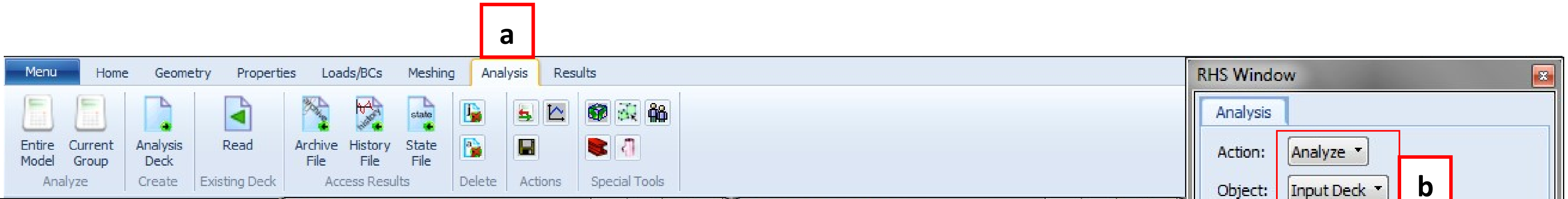
k. Click **Select Application Region...**
l. Click on the **Select Members** panel
m. Click on the **Quad element** icon
n. Select the elements by clicking and dragging the mouse
o. Click **Add**
p. Click **OK**
q. Click **Apply**

r. Post only the **pipe** group
s. Change *the Property Set Name* to **pipe**
t. Click **Input Data...**
u. Click on the **Material Name** panel
v. Select **aluminum**
w. Enter **4** as the Number of Points
x. Enter **2** as the Thickness
y. Click **OK**



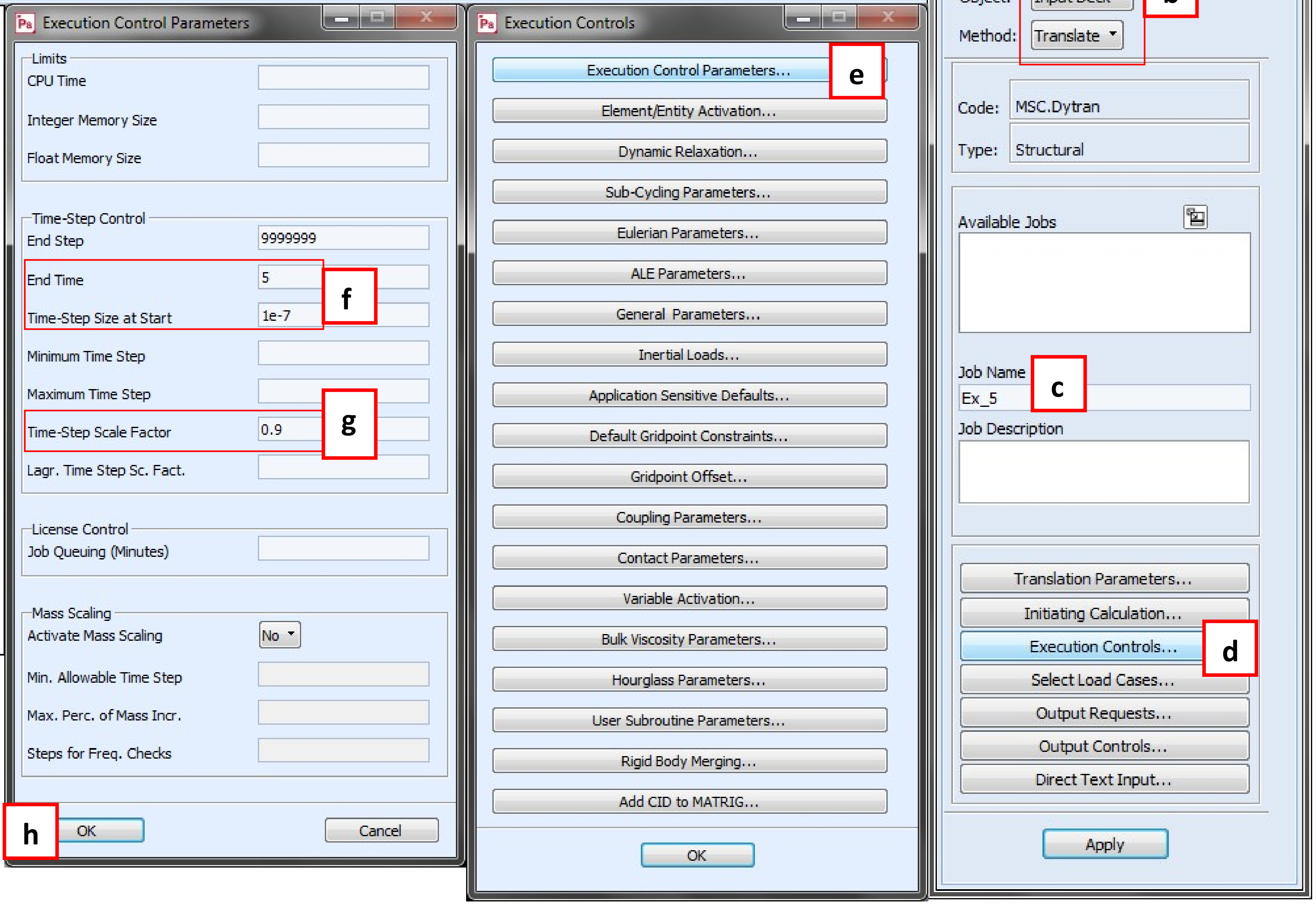


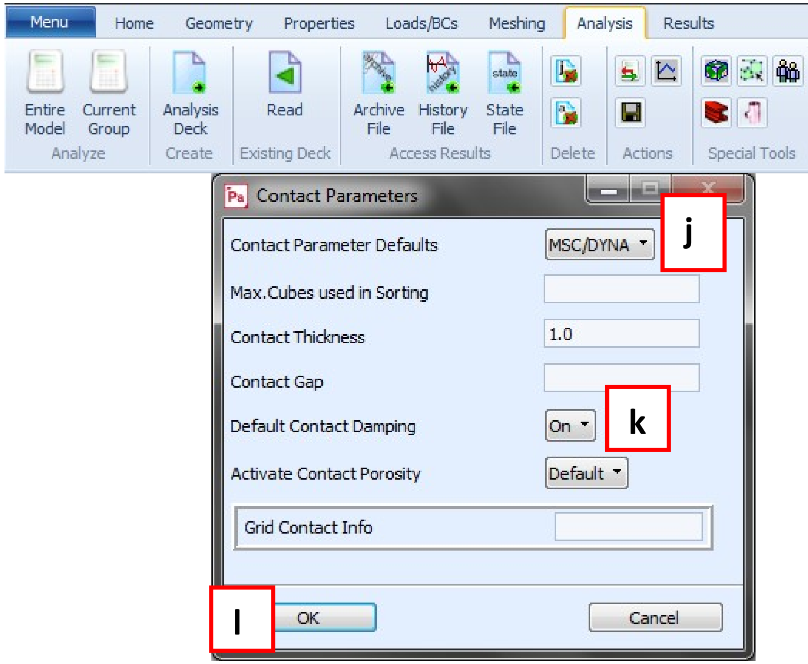
z. Click **Select Application Region…**
aa. Click on the **Select Members** panel
bb. Click on the **Quad element** icon
cc. Select the elements by clicking and dragging the mouse
dd. Click **Add**
ee. Click **OK**
ff. Click **Apply**

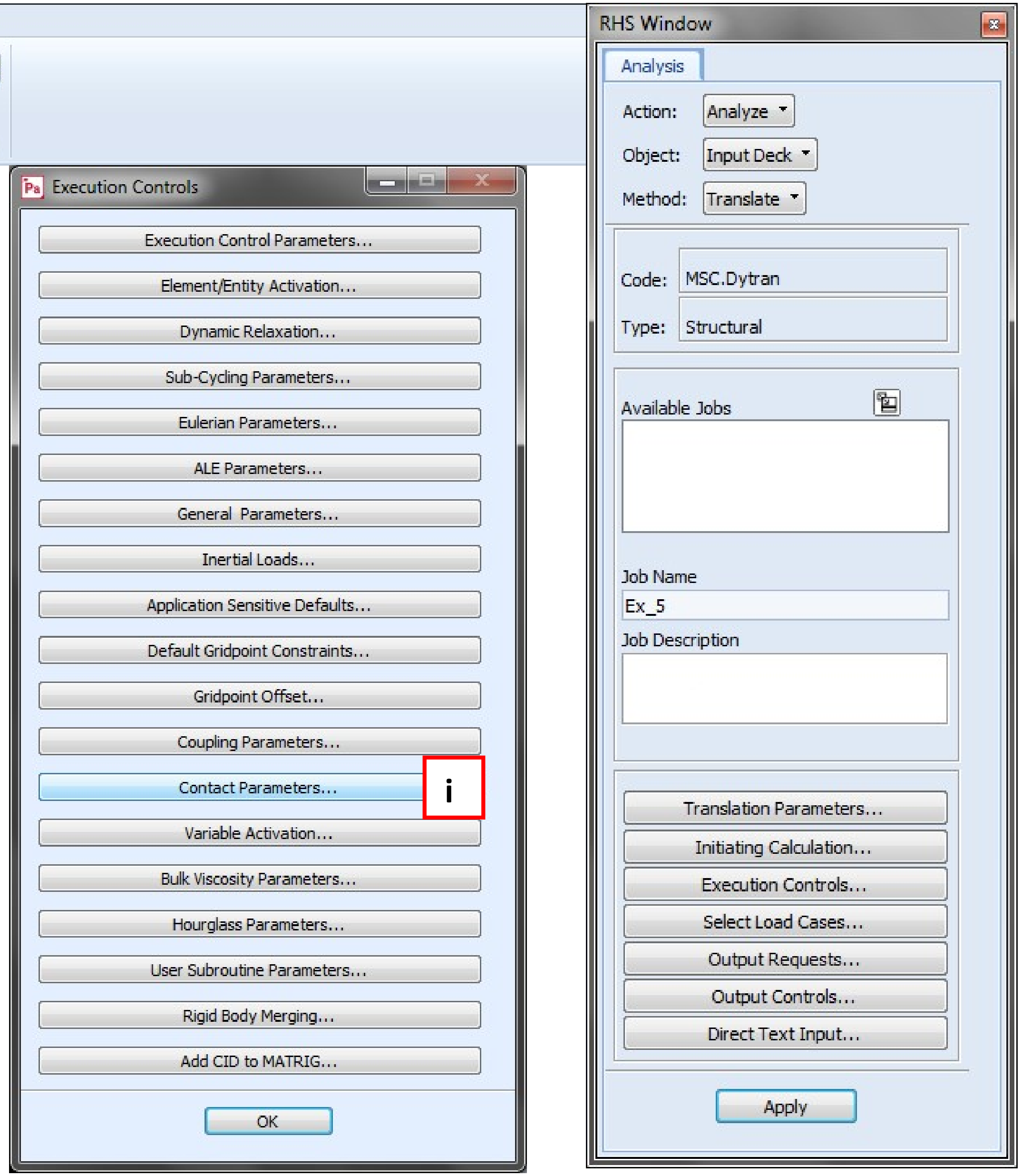


Set analysis parameters:

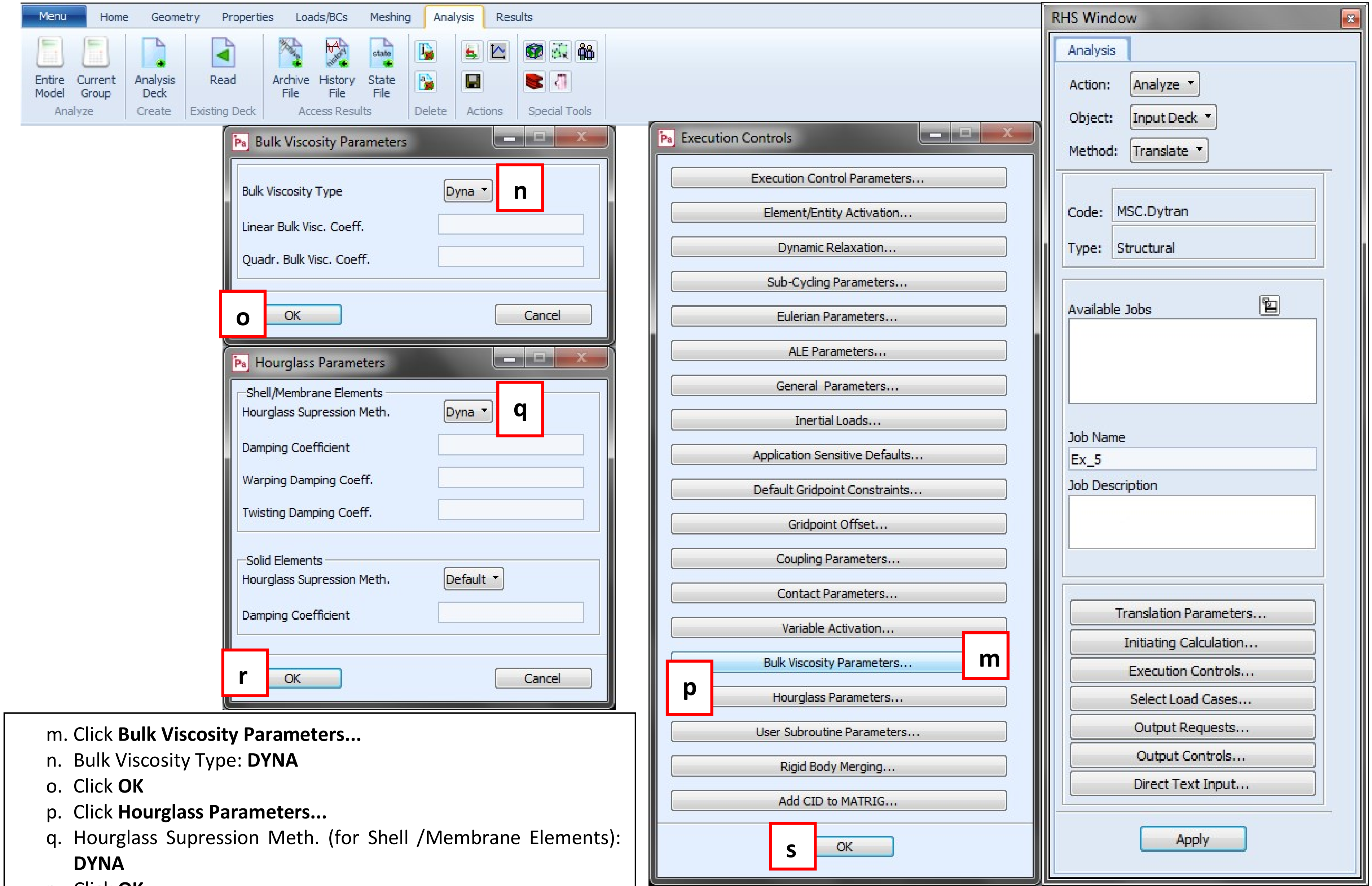
a. Click on the **Analysis** icon
b. Analysis: **Analyze/Input Deck/Translate**
c. Enter **ex_5** as the Job Name
d. Click **Execution Controls..**
e. Click **Execution Control Parameters**
f. Enter **5** as the End Time and **1e-7** as the Time-Step Size at Start
g. Enter **0.9** as the Time-Step Scale factor
h. Click **OK**







i. Click **Contact Parameters...**
j. Contact Parameter Defaults: **MSC/DYNA**
k. Default Contact Damping: **On**
l. Click **OK**



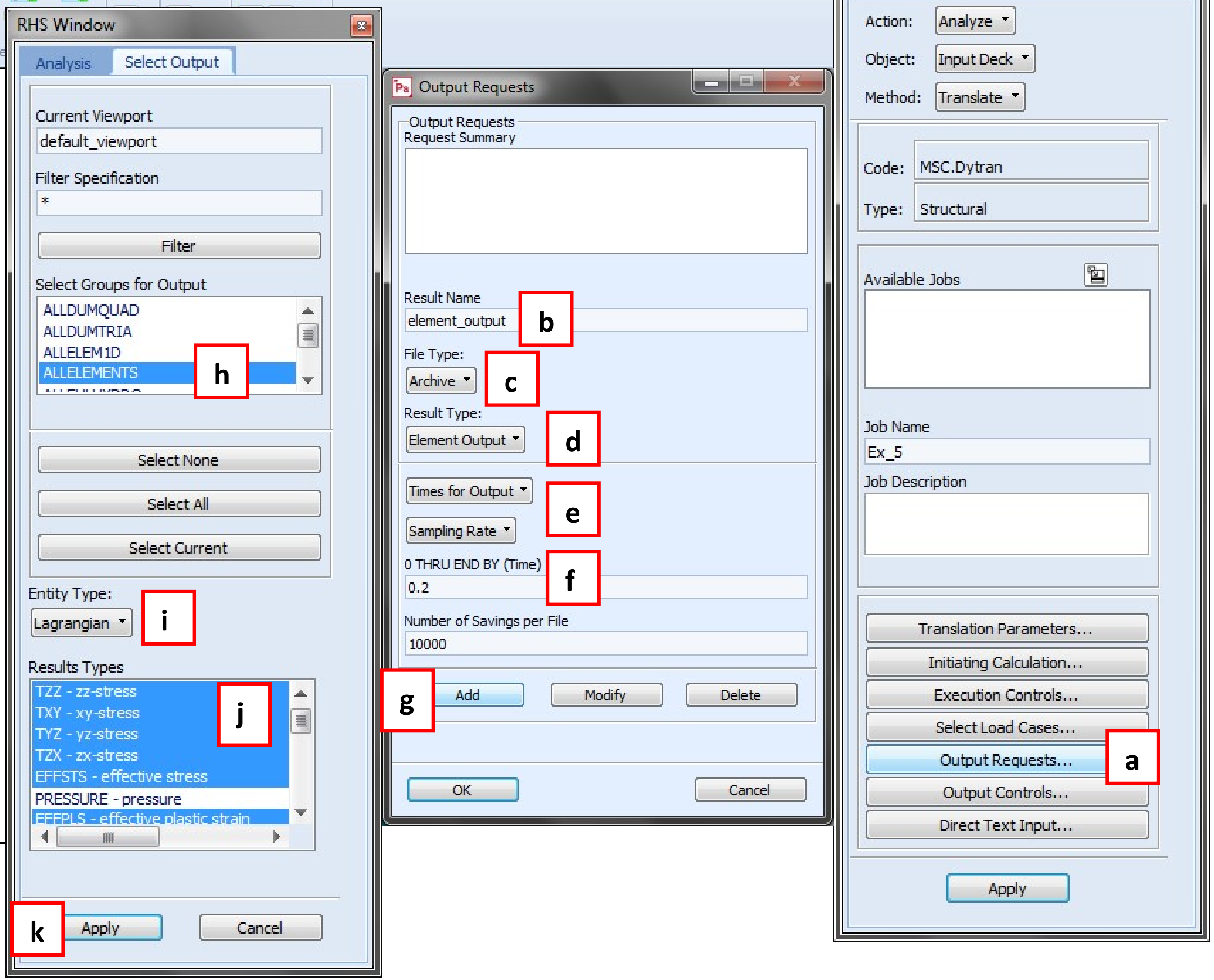
m. Click **Bulk Viscosity Parameters...**
n. Bulk Viscosity Type: **DYNA**
o. Click **OK**
p. Click **Hourglass Parameters...**
q. Hourglass Supression Meth. (for Shell /Membrane Elements): **DYNA**
r. Click **OK**
s. Click **OK**

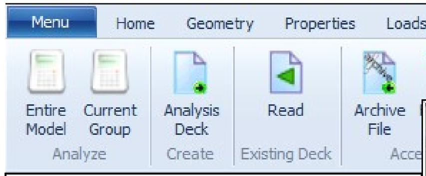
HUMAN CAPITAL
NATIONAL COHESION STRATEGY

EUROPEAN UNION
EUROPEAN
SOCIAL FUND

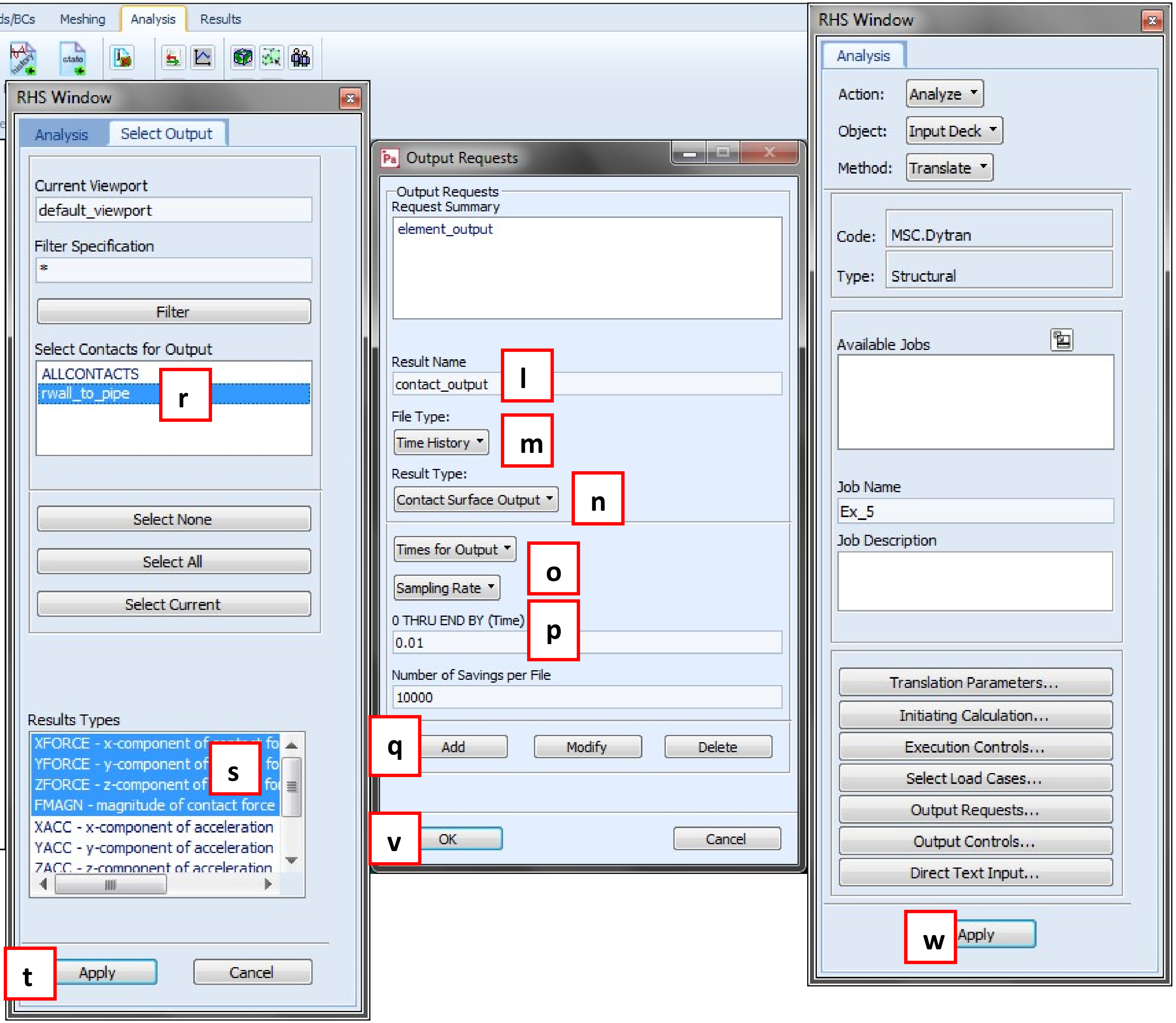
Define the output:

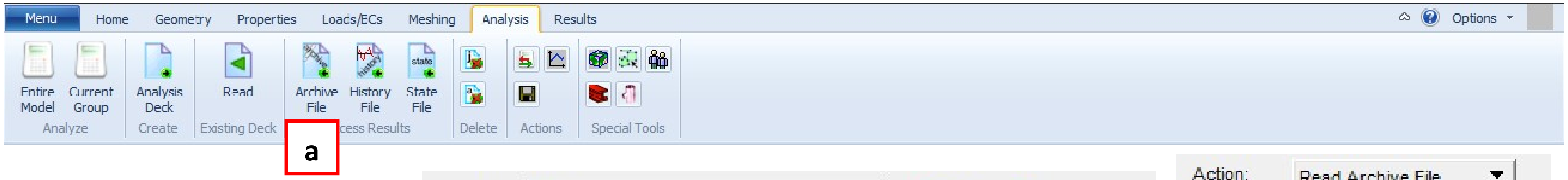
a. Click **Output Requests...**
b. Enter **element_output** as the Result Name
c. File Type: **Archive**
d. Result Type: **Element Output**
e. Select **Times for Output** and **Sampling Rate**
f. Enter **0.2** as the archive time step
g. Click **Add**
h. Select **ALLELEMENTS**
i. Change *Entity Type* to **Lagrangian**
j. Select Results Types:
    - Stress: **TXX**, **TYY**, **TZZ**, **TXY**, **TYZ**, **TZX**
    - **EFFSTS** (effective stress)
    - **EFFPLS** (effective plastic strain)
    - Centroidal Strain: **EPSXX**, **EPSYY**, **EPSZZ**
k. Click **Apply**





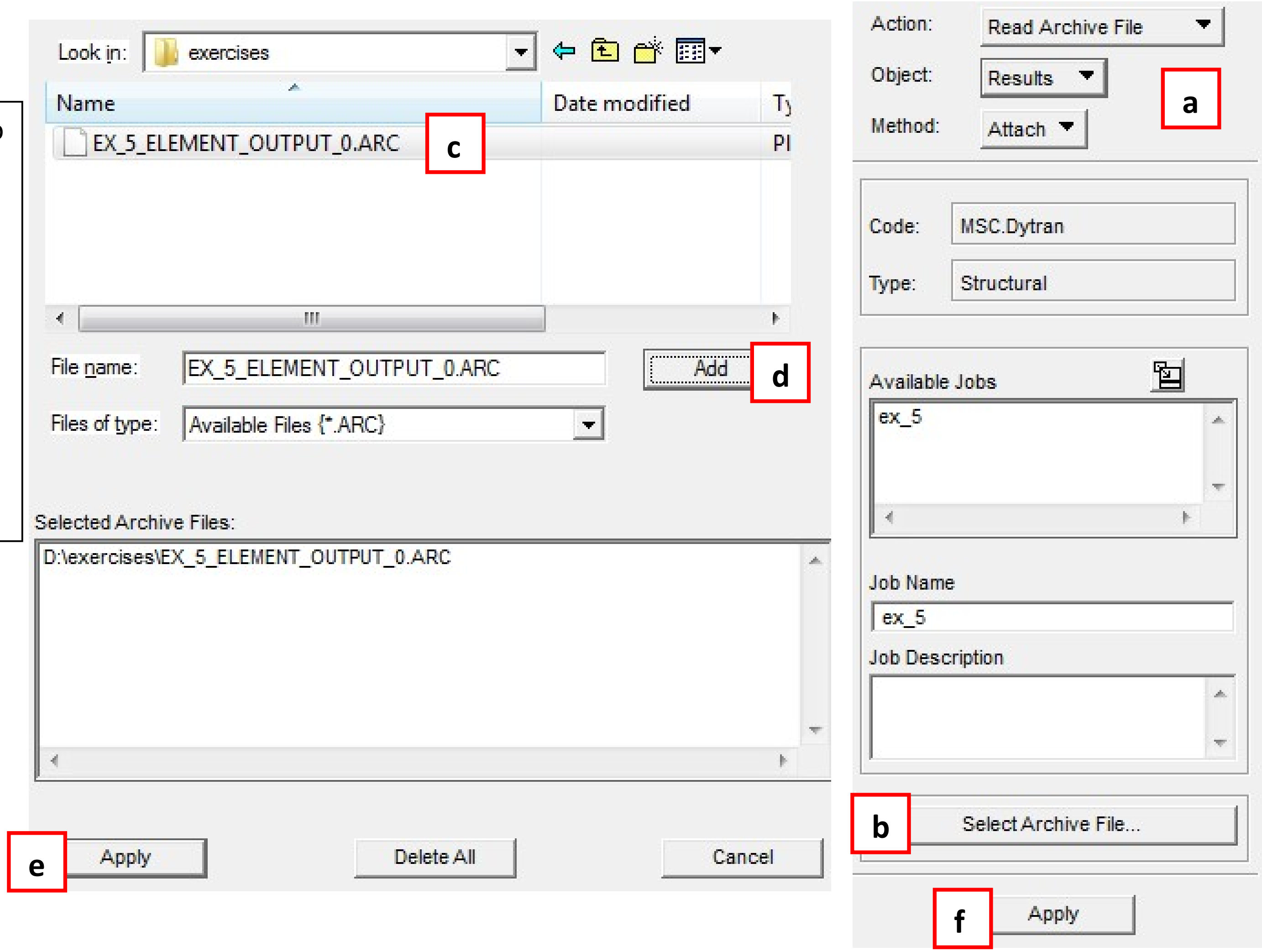
l. Enter **contact_output** as the Result Name
m. File Type: **Time History**
n. Result Type: **Contact Surface Output**
o. Select **Times for Output** and **Sampling Rate**
p. Enter **0.01** as the archive time step
q. Click **Add**
r. Select **rwall_to_pipe**
s. Select Results Types: **XFORCE**, **YFORCE**, **ZFORCE**, **FMAGN**
t. Click **Apply**
u. <u>**Additional task:**</u> Using the **Time History** and **Grid Point Output** options, try to define an output for the rigid wall
v. Click **OK**
w. Click **Apply**
x. Run **Dytran** analysis using **Ex_5.dat** file

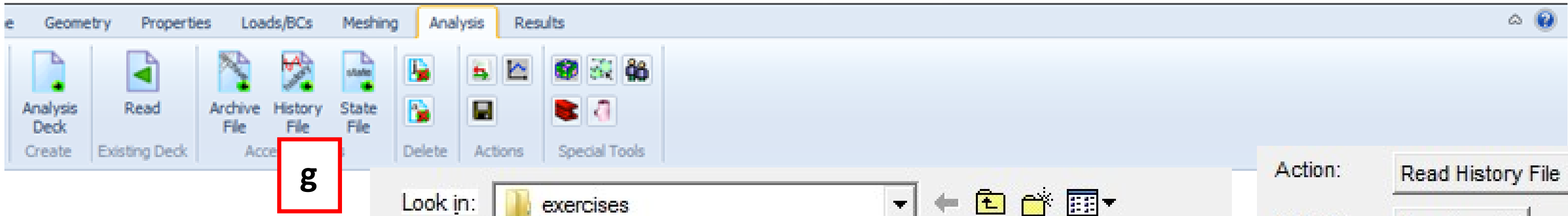




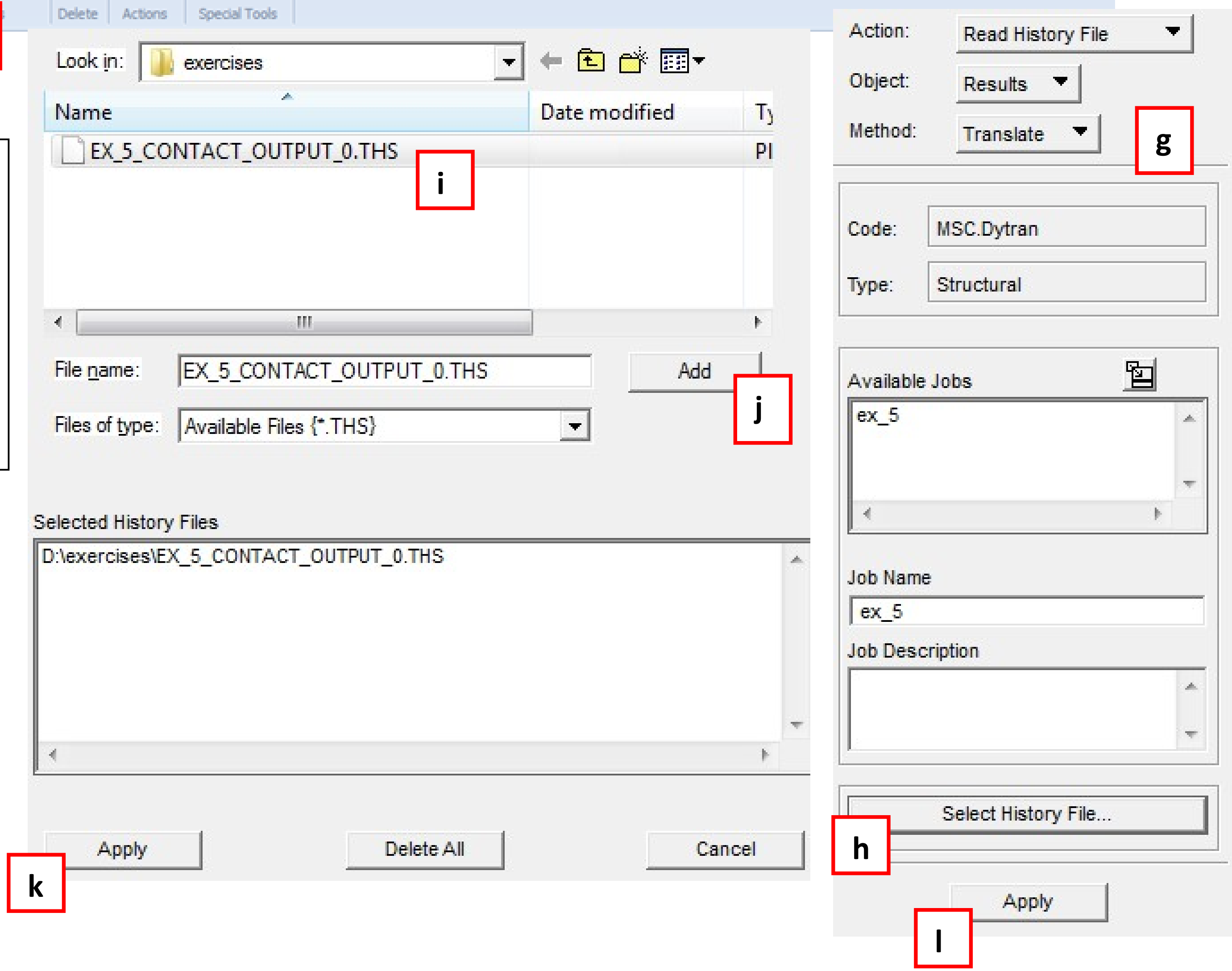
Attach the results files, when the analysis job is completed:

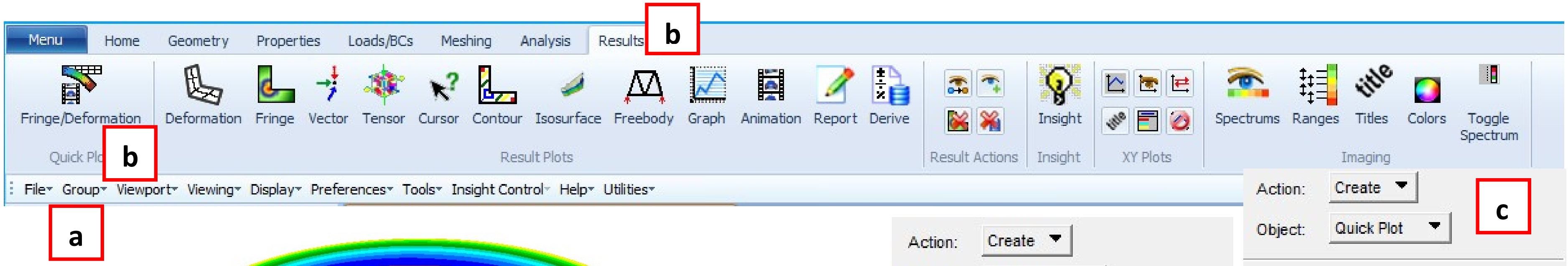
a. Analysis:
   Read Archive File/Results/Attach
b. Click **Select Archive File…**
c. Select the **EX_5_ELEMENT_OUTPUT_0.ARC** file
d. Click **Add**
e. Click **Apply**
f. Click **Apply**

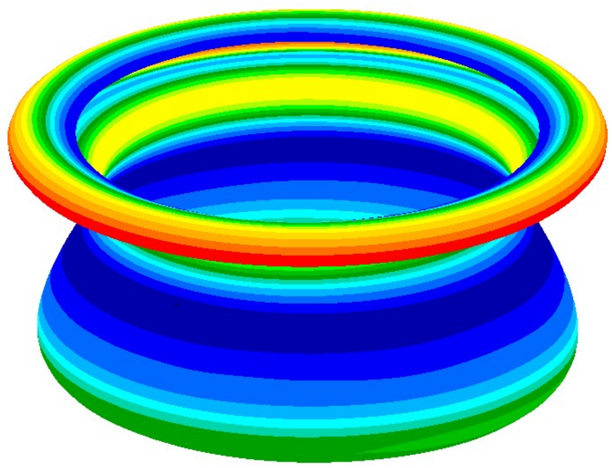




g. Analysis: **Read History File /Results/Attach**
h. Click **Select History File...**
i. Select the **EX_5_CONTACT_OUTPUT_0.THS** file
j. Click **Add**
k. Click **Apply**
l. Click **Apply**





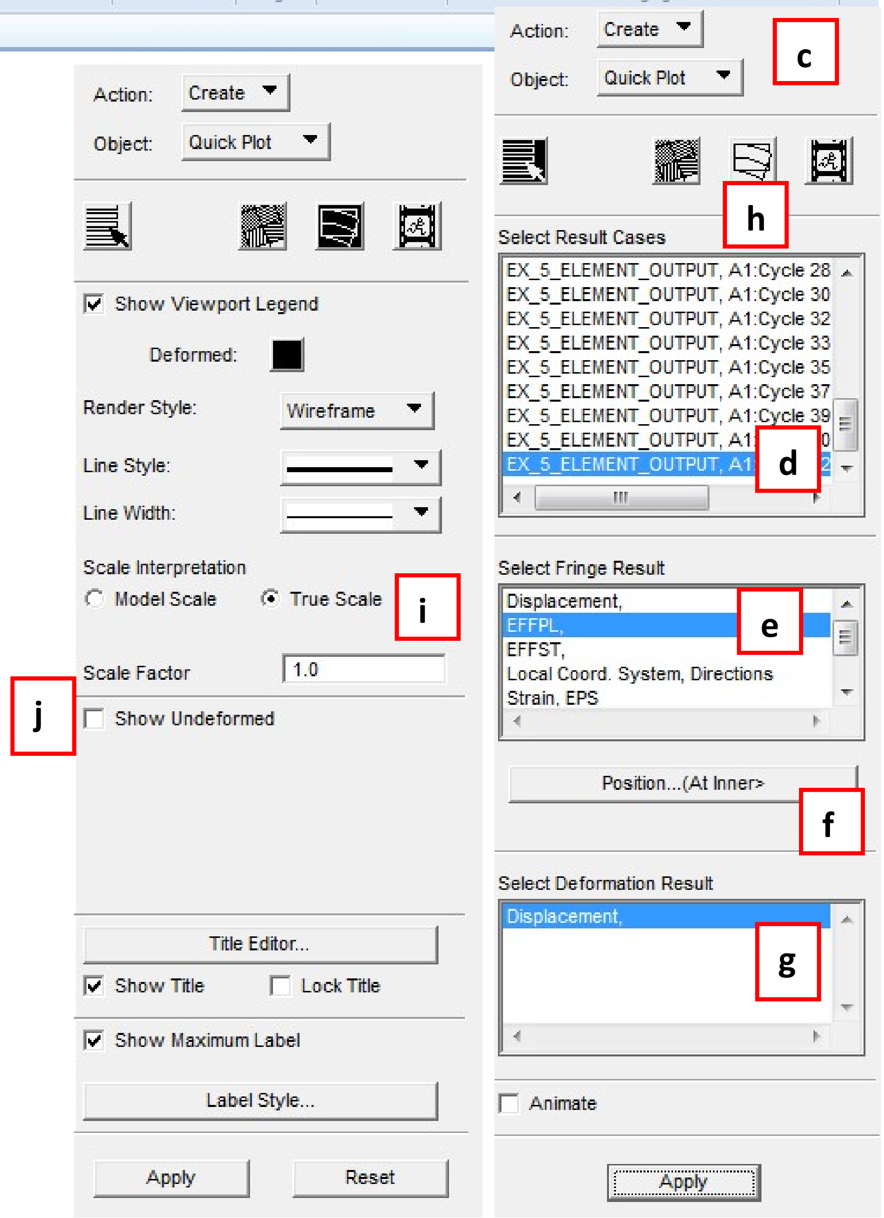


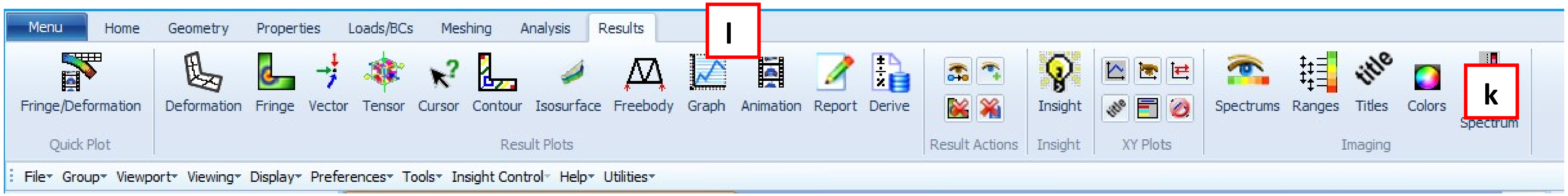
- a. Post only the **pipe** group (from menu: **Group/Post**....)

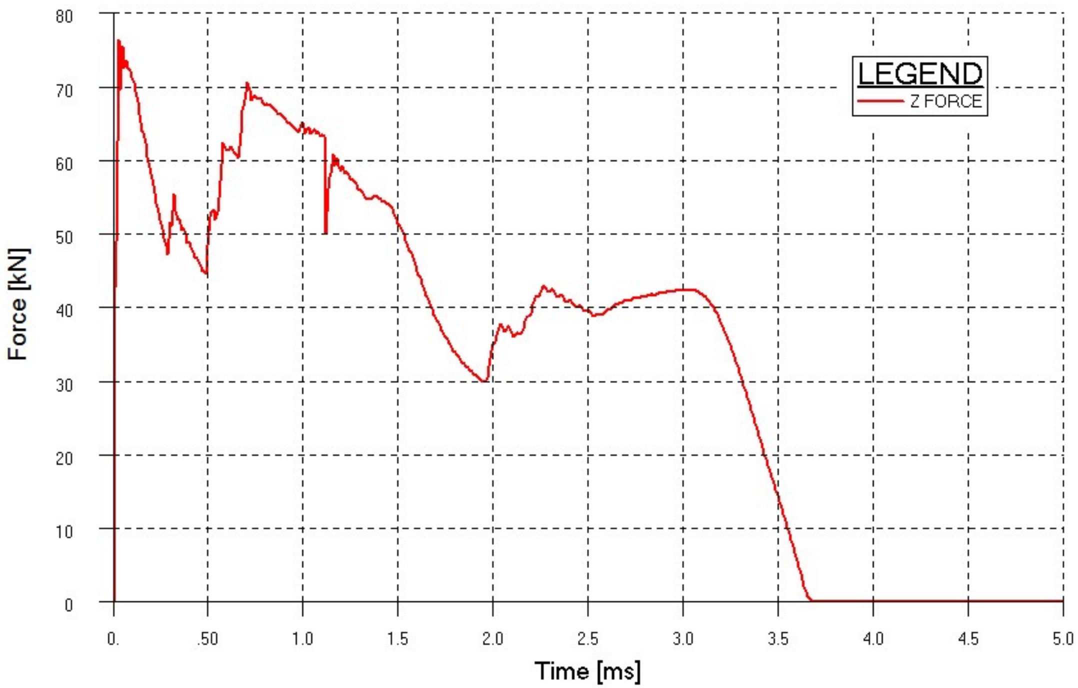
Post-process the results:

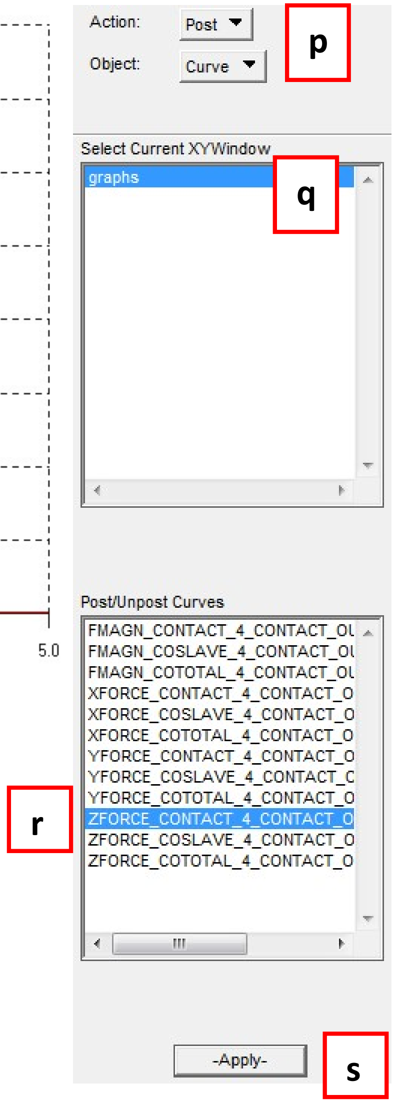
- b. Click on the **Results** tab in the Ribbon (***Fringe/Deformation*** icon)
- c. Results: **Create/Quick Plot**
- d. Select ***the last step***
- e. Select Fringe Result: **EFFPL** (EFFective PLastic strain)
- f. Position: select **At Inner**, **At Middle**, **At Outer**
- g. Select Deformation Result: **Displacement**
- h. Click on the **Deform Attributes** icon
- i. Select **True Scale** with the Scale Factor equals to **1.0**
- j. Uncheck **Show Undeformed**
- k. Click **Apply**

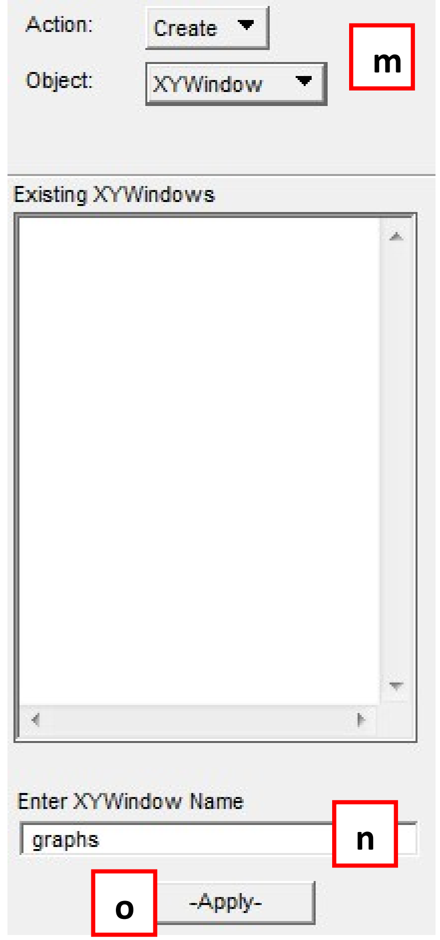
Remark: To capture the plot use **File / Images…**











l. Click on the **Graph** icon
m. XY Plot: **Create/XYWindow**
n. Enter **graphs** as the XYWindow Name
o. Click **Apply**
p. Change *Action* to **Post** (Object: **Curve**)
q. Select **graphs**
r. Select **ZFORCE_CONTACT**
s. Click **Apply**

HUMAN CAPI
NATIONAL COHESION STI

EUROPEAN UNION
EUROPEAN
SOCIAL FUND