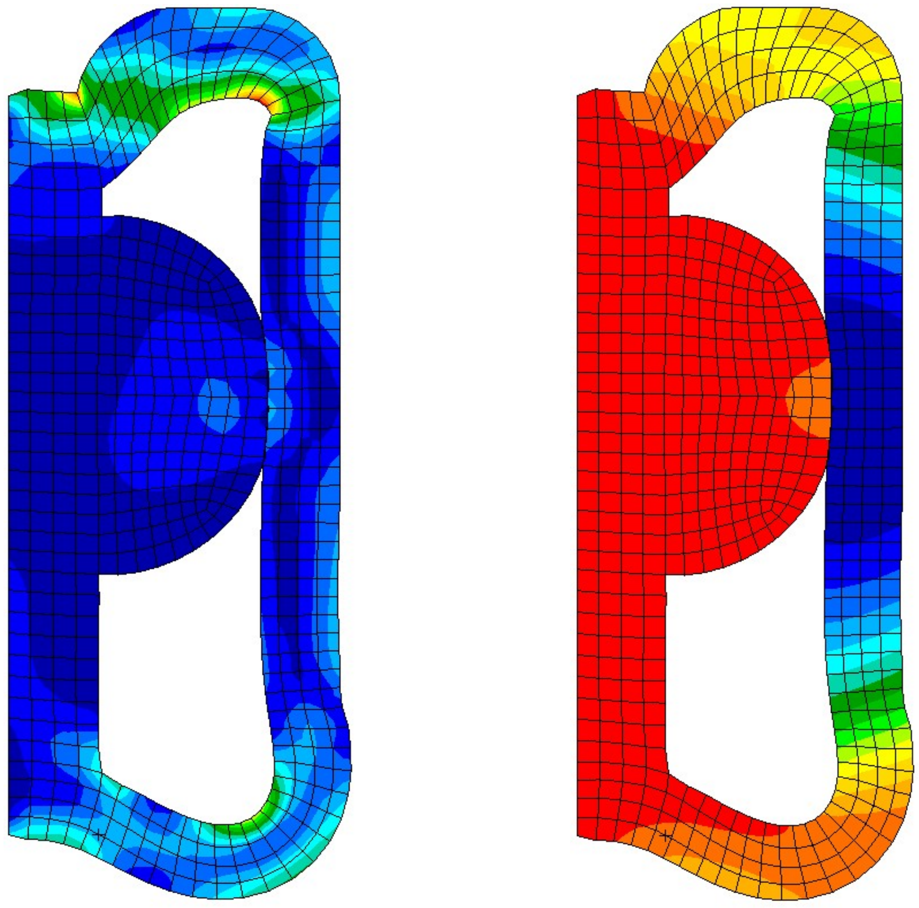
NONLINEAR MECHANICS OF STRUCTURES

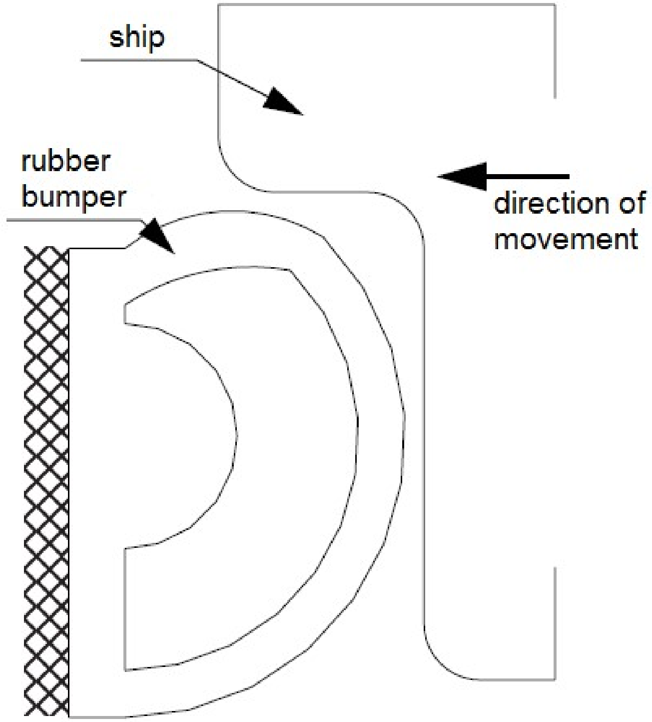
EXERCISE 1



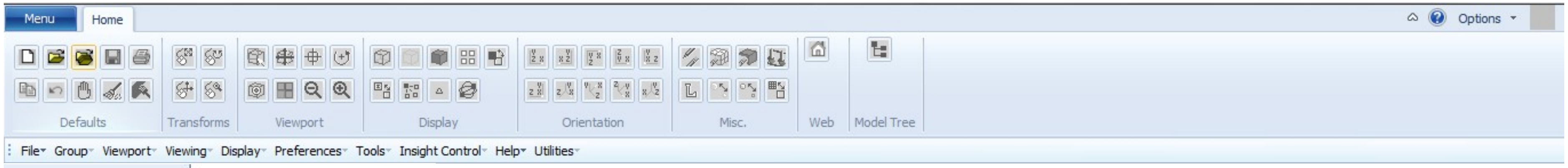
HUMAN CAPITAL
NATIONAL COHESION STRATEGY

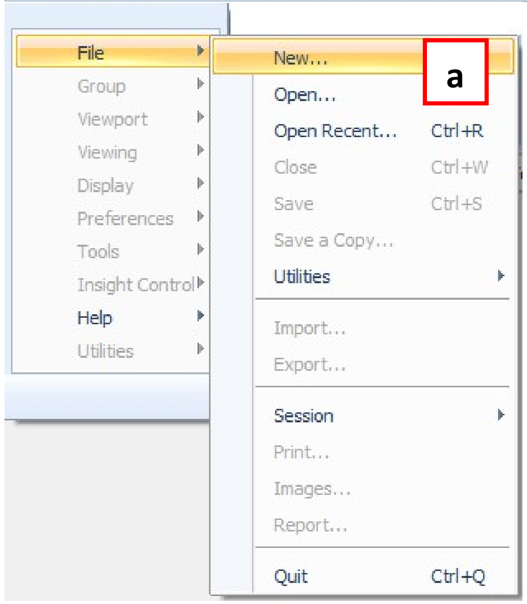
EUROPEAN UNION
EUROPEAN
SOCIAL FUND

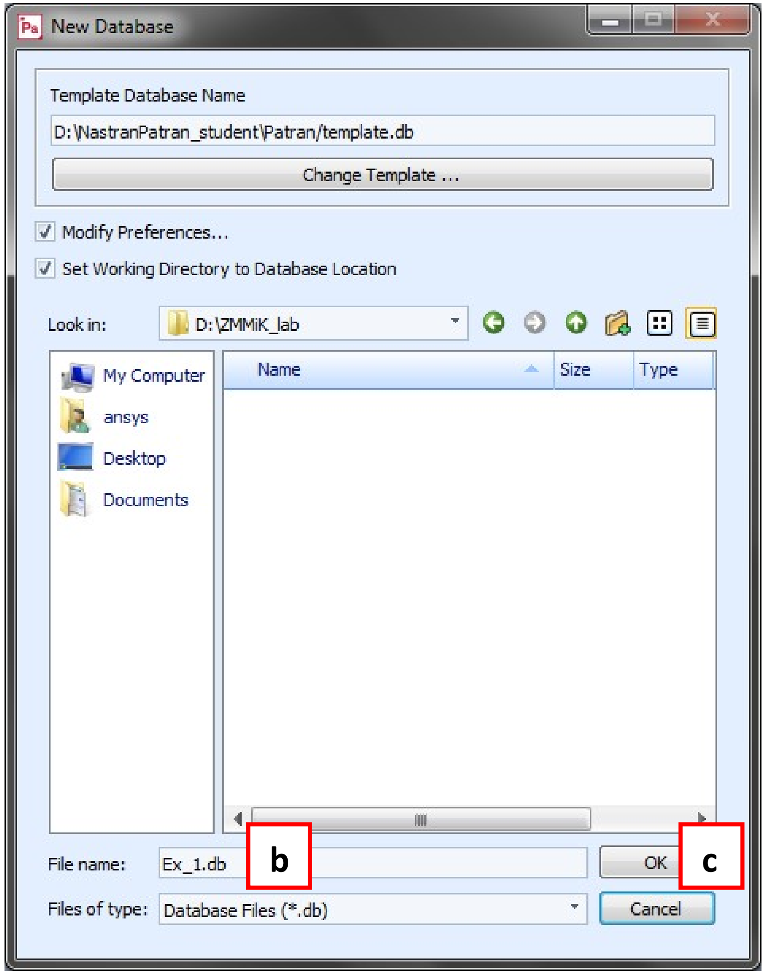
PROBLEM DESCRIPTION

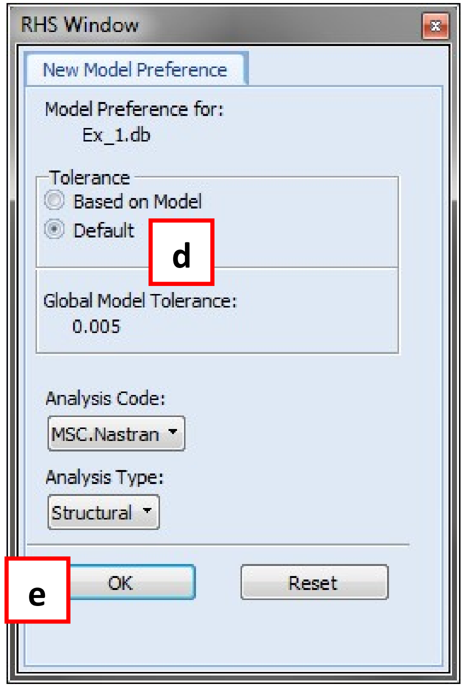


A rubber bumper comes into a contact with a ship. The geometry of the parts is described in an iges CAD file that is imported to Patran for mesh generation. The bumper is assumed to be in a state of plane strain. Contact between the bumper and the ship includes friction.



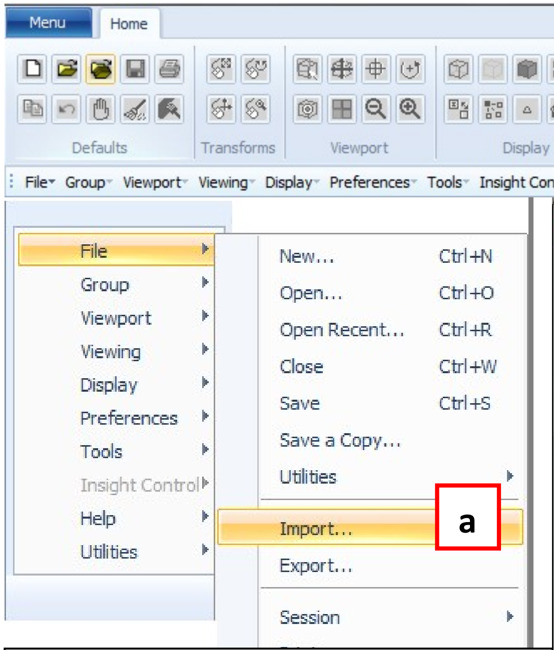


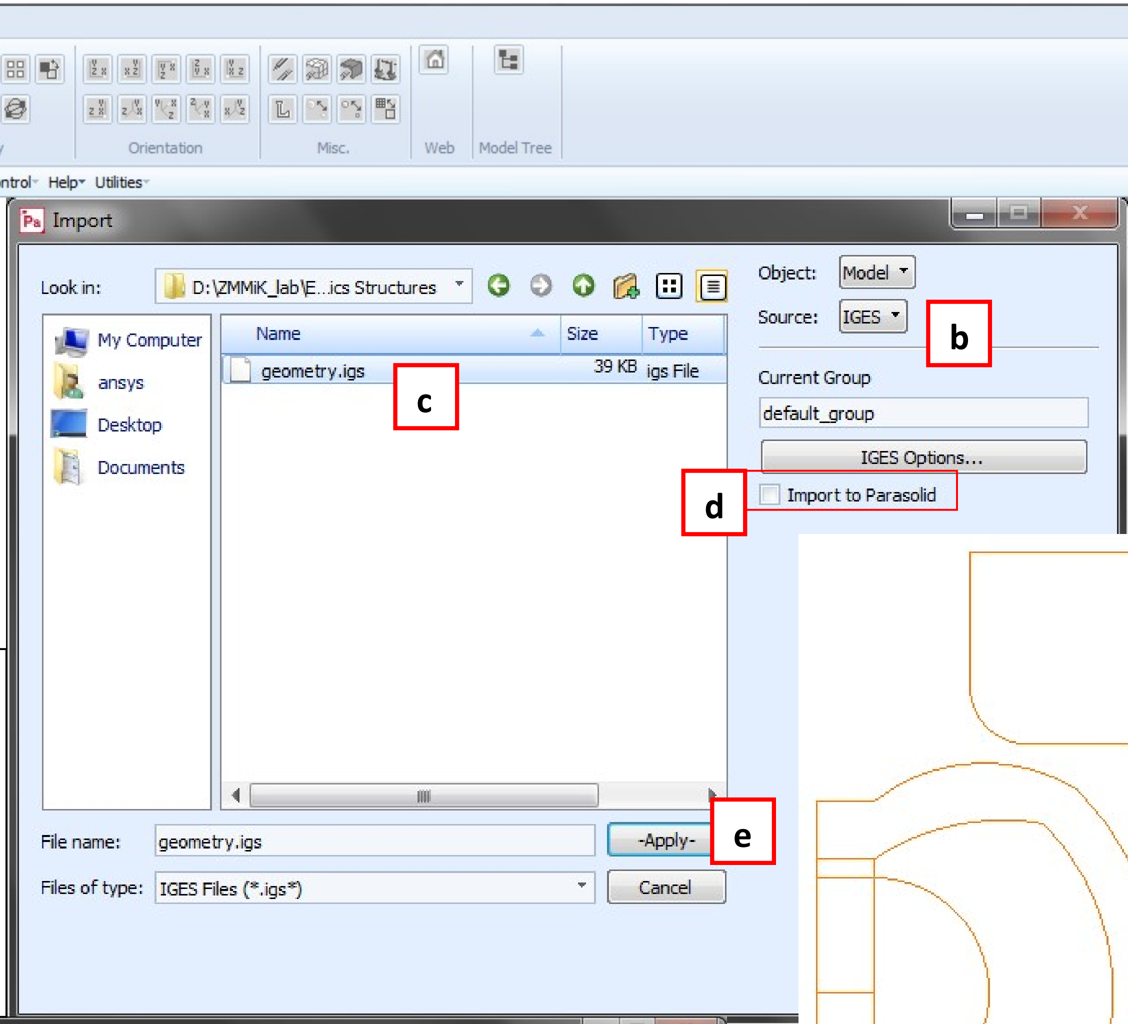




Create a new database:

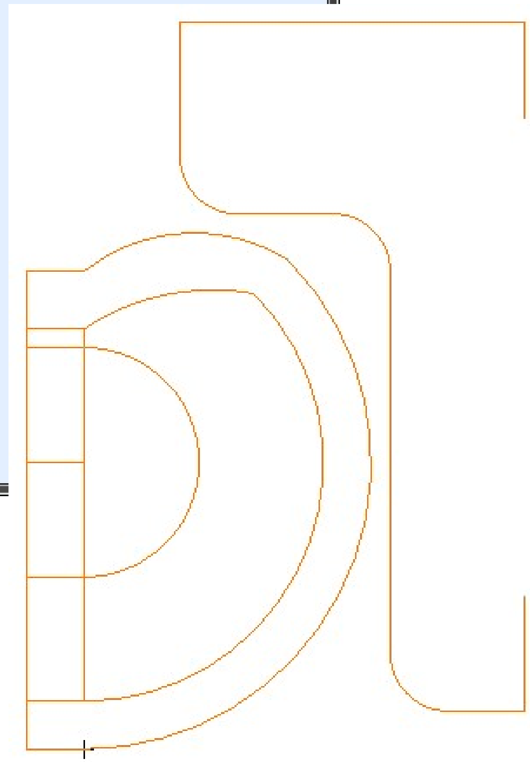
a. **File / New...**
b. Enter **ex_1.db** as the File name
c. Click **OK**
d. Select **Default**
e. Click **OK**

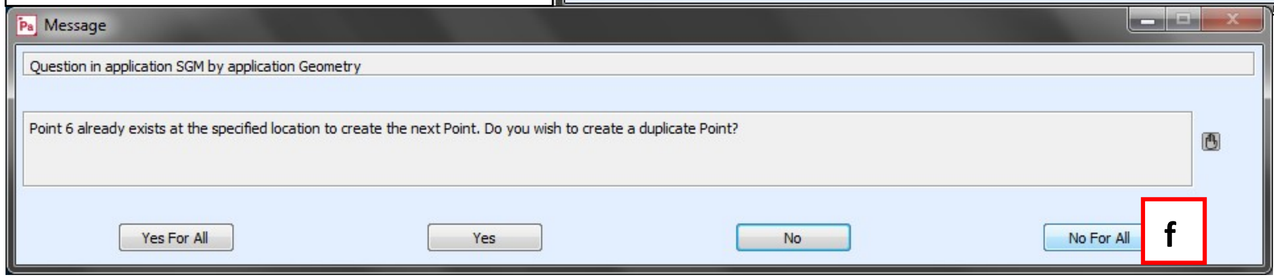


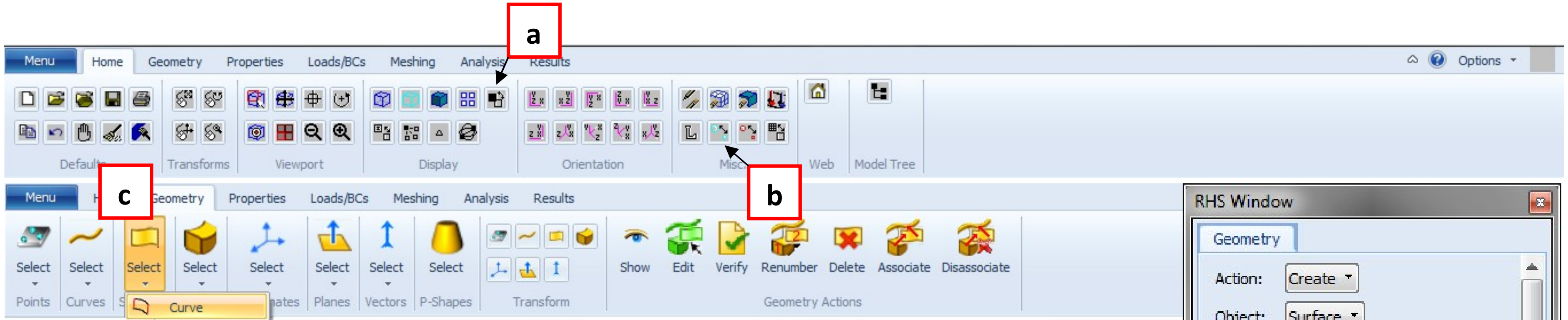


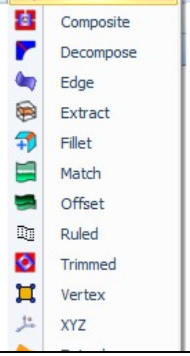
Import the geometry:

a. **File / Import…**
b. Source: **IGES**
c. Find and select the **geometry.igs** file
d. Uncheck **Import to Parasolid**
e. Click **Apply**
f. Click **No For All**
g. IGES Import Summary: click **OK**







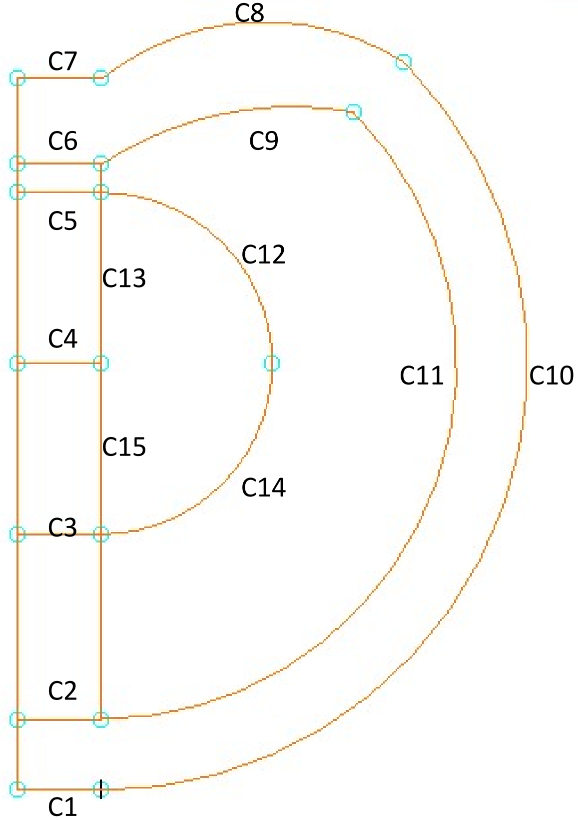


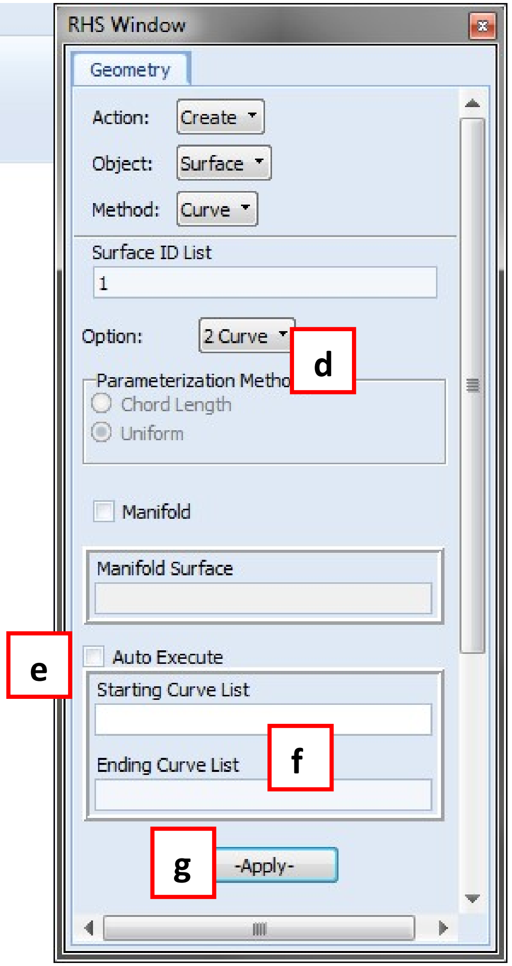
- a. Change Background Color to **Black** (click on the **Cycle Background** icon)
- b. Click on the **Point size** icon

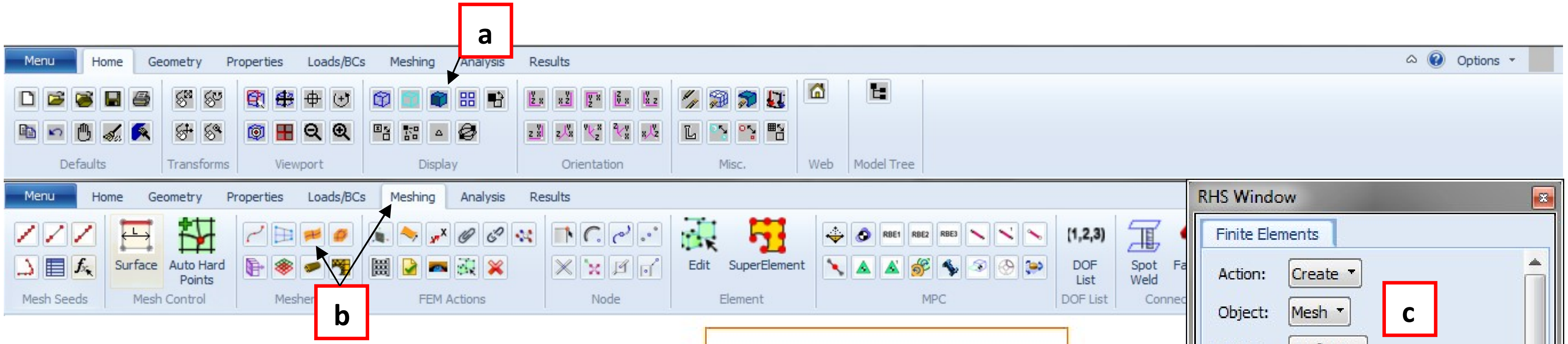
Create surfaces (defining the bumper):

- c. Click on the **Geometry** icon**/Select/Curve**
- d. Option: **2 Curve**
- e. Uncheck **Auto Execute**
- f. Select the **C1** curve as the starting curve and the **C2** curve as the ending curve
- g. Click **Apply**
- h. Create <u>nine</u> more surfaces using remaining curves – start from the pair: **C2** & **C3**

<u>Remark</u>: As the starting curve select <u>always</u> a curve with lower id. Ids are shown in figure on the right).



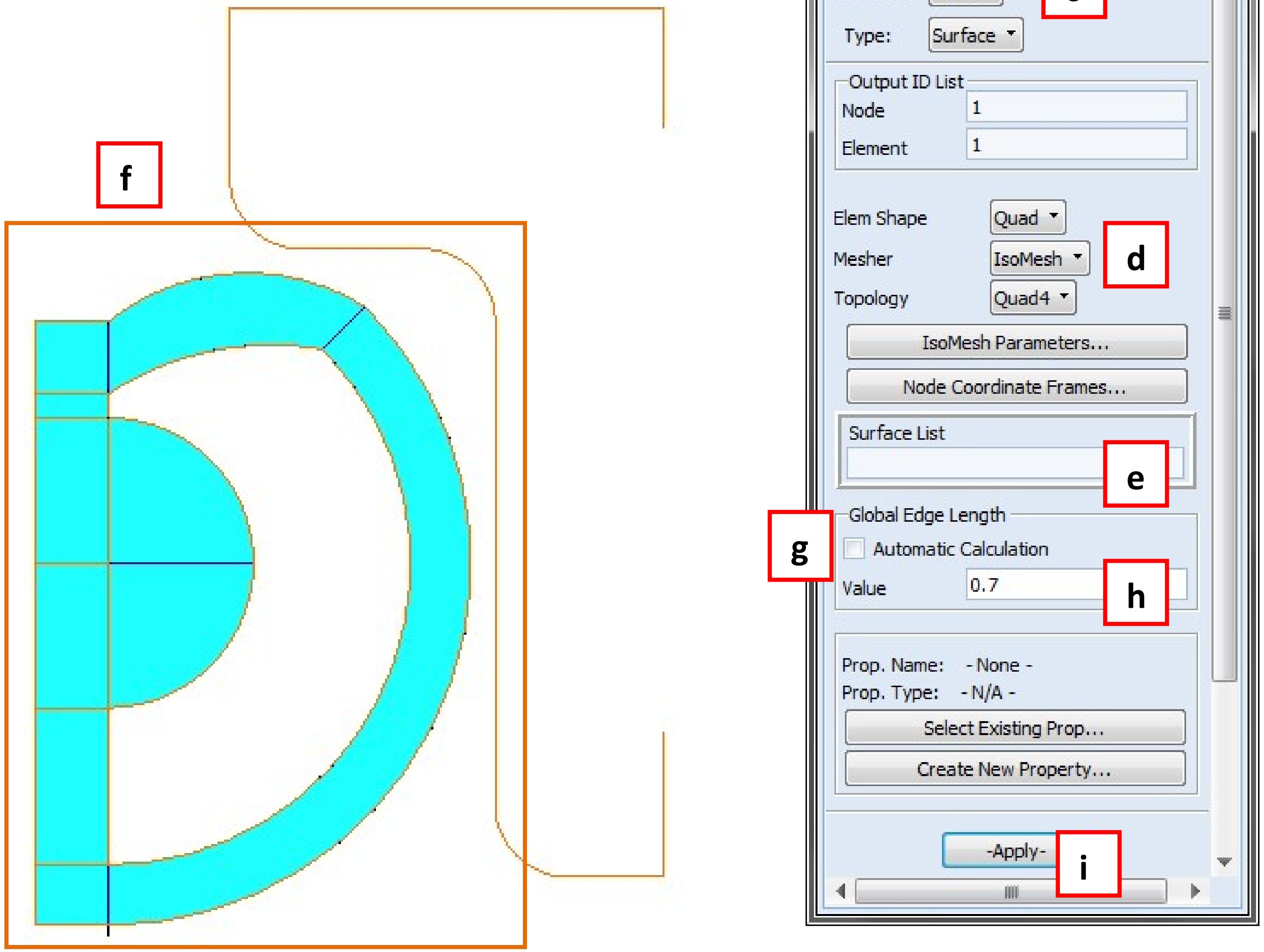




a. Click on the **Home/Smooth shaded** icon

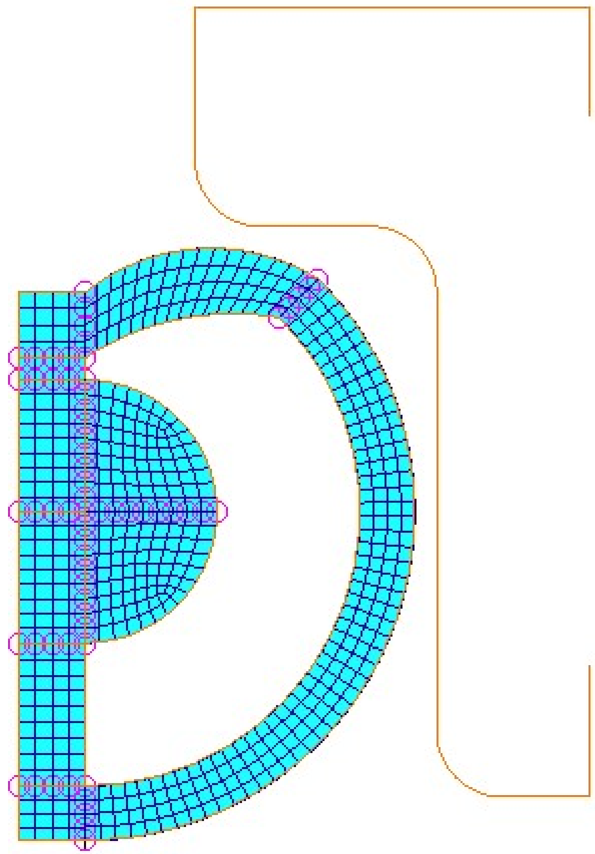
Mesh the surfaces:

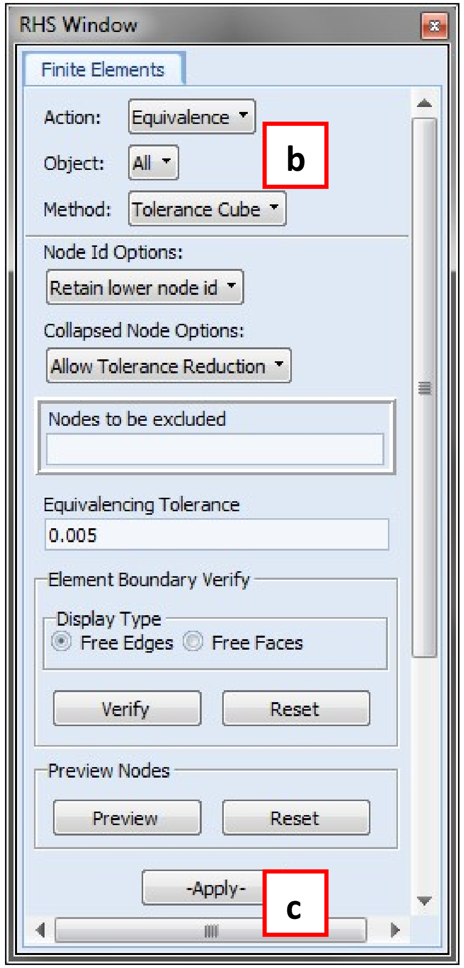
b. Click on the **Meshing/Surface** icon *(Meshers tab)*
c. Check out: **Create/Mesh/Surface**
d. Elem Shape: **Quad**; Mesher: **IsoMesh**; Topology: **Quad4**
e. Click on the **Surface List** panel
f. Select the surfaces by clicking and dragging the mouse
g. Uncheck **Automatic Calculation**
h. Enter **0.7** as the Value of the Global Edge Length
i. Click **Apply**



Delete duplicate nodes:

a. Meshing: **Equivalence** icon
b. Check out: **Equivalence/All/Tolerance Cube**
c. Click **Apply**



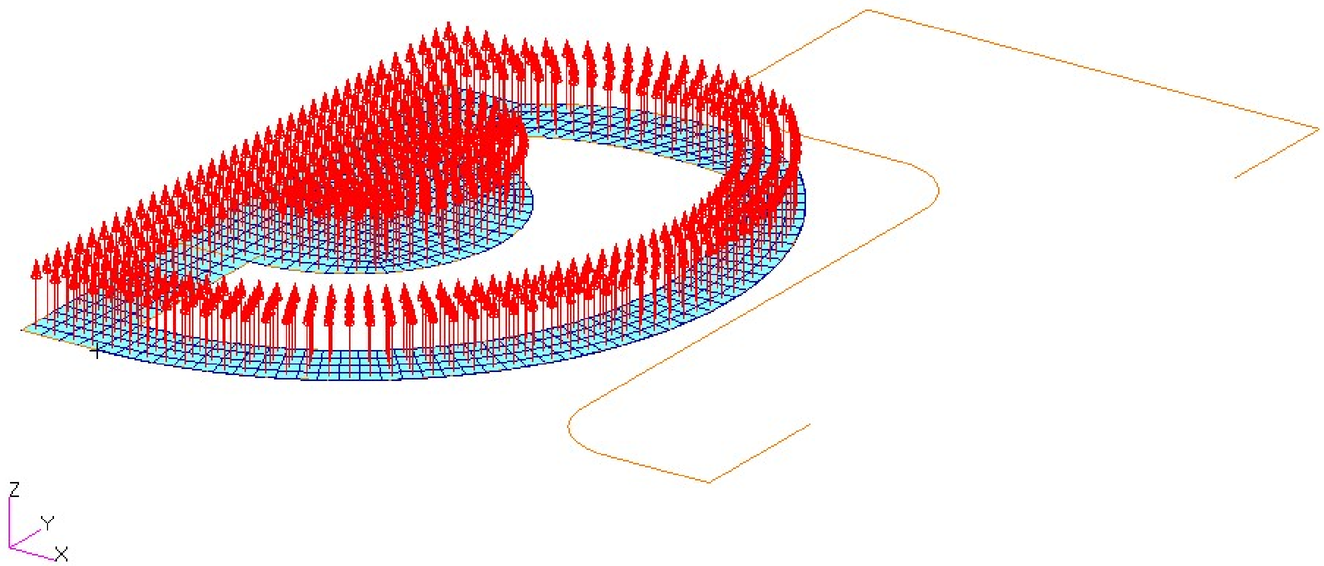


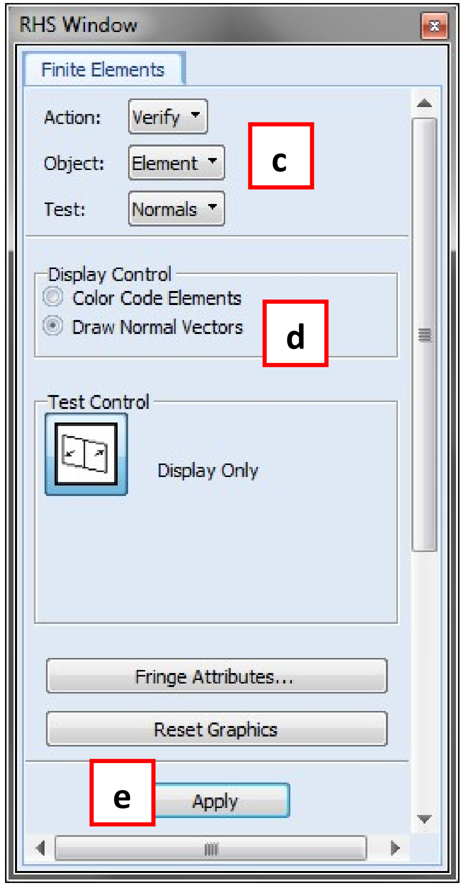
a. Click on the **Iso 3 View** icon
b. Click on the **Fit view** icon

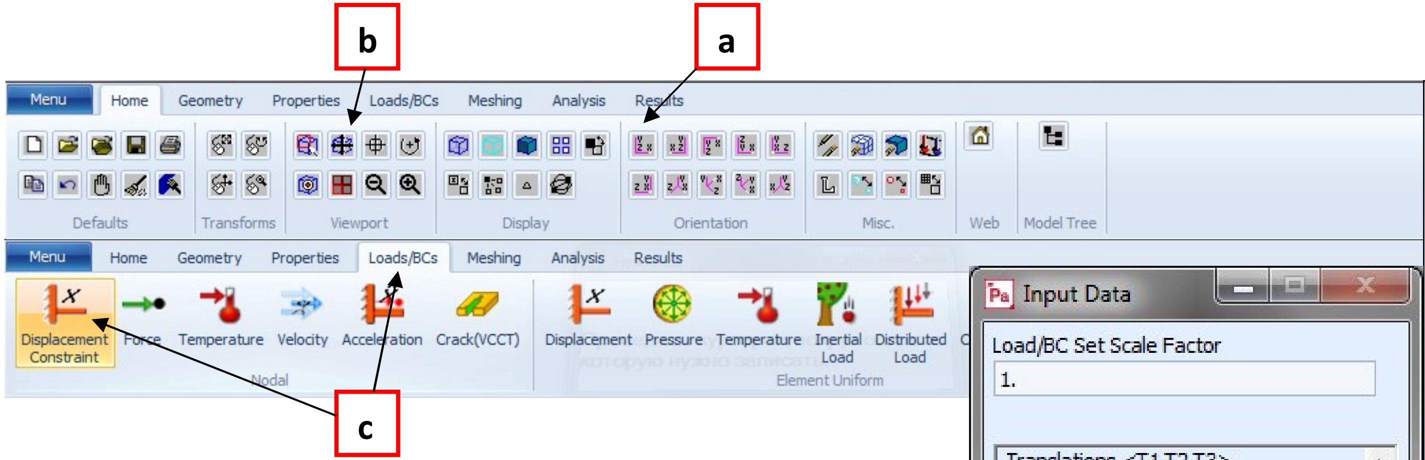
Verify element normals:

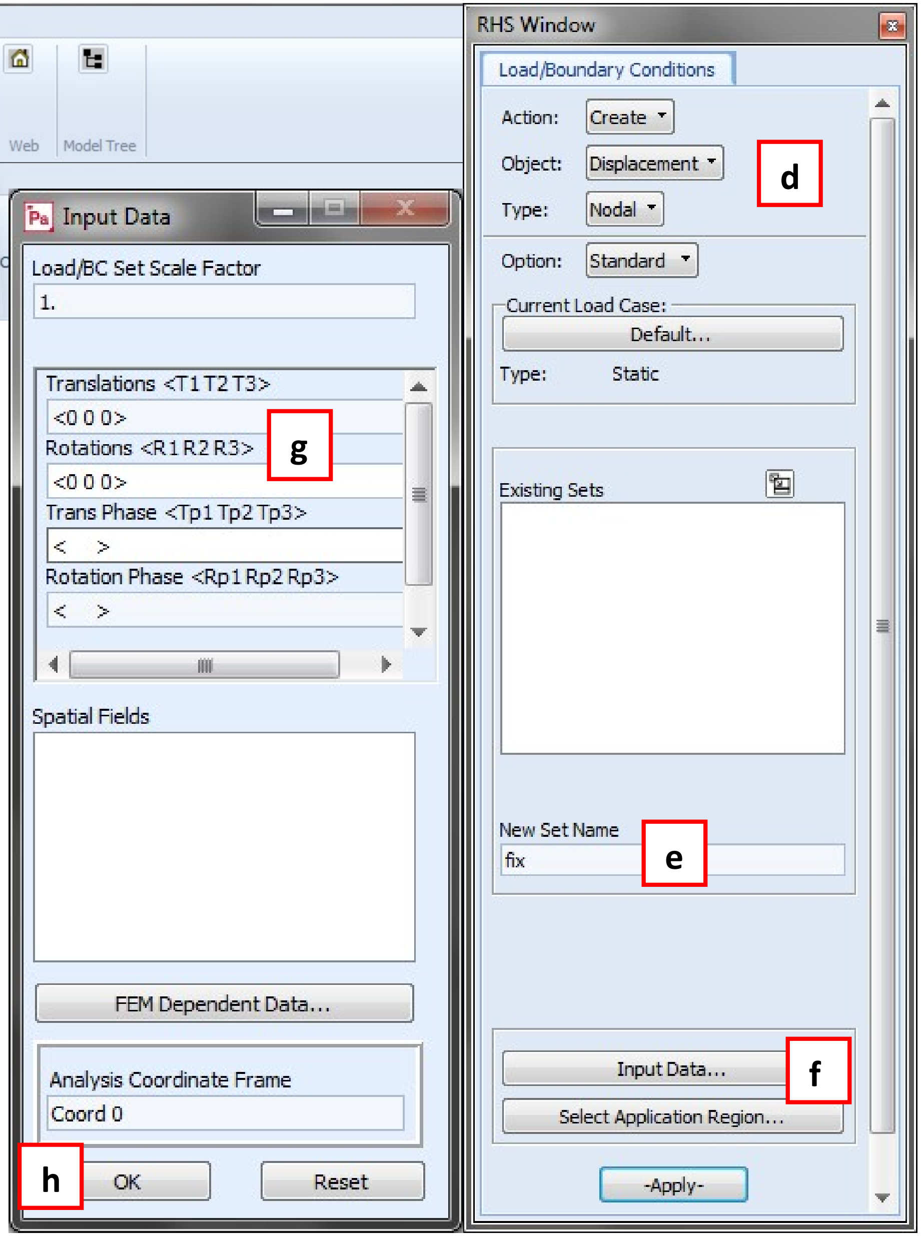
c. Elements: **Verify/Element/Normals**
d. Select **Draw Normal Vectors**
e. Click **Apply**

Remark: Make sure that all normal vectors of the elements are pointed towards the positive direction of Z-axis. If not, use **Elements: Modify/Element/Reverse** to reverse element normals.





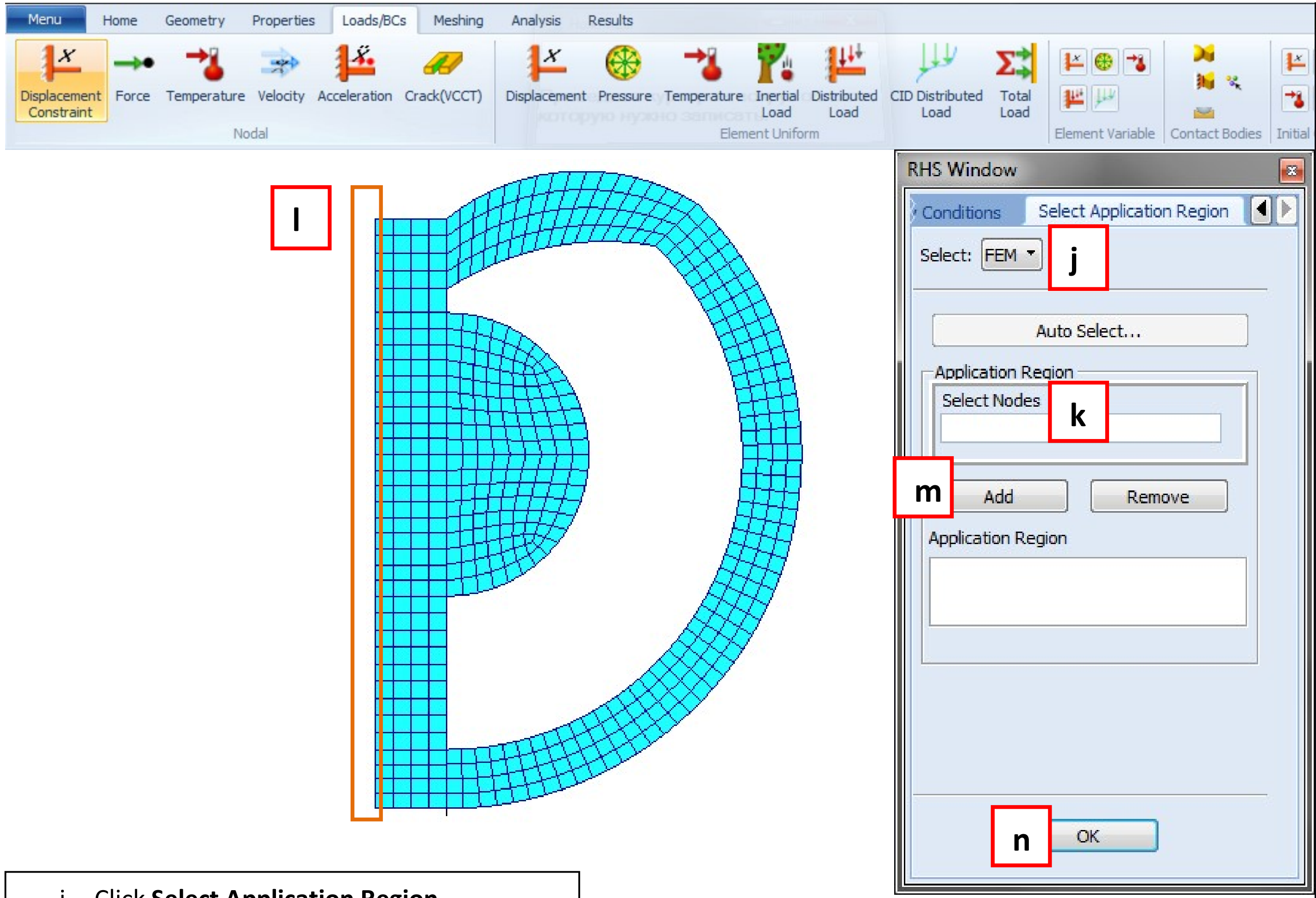


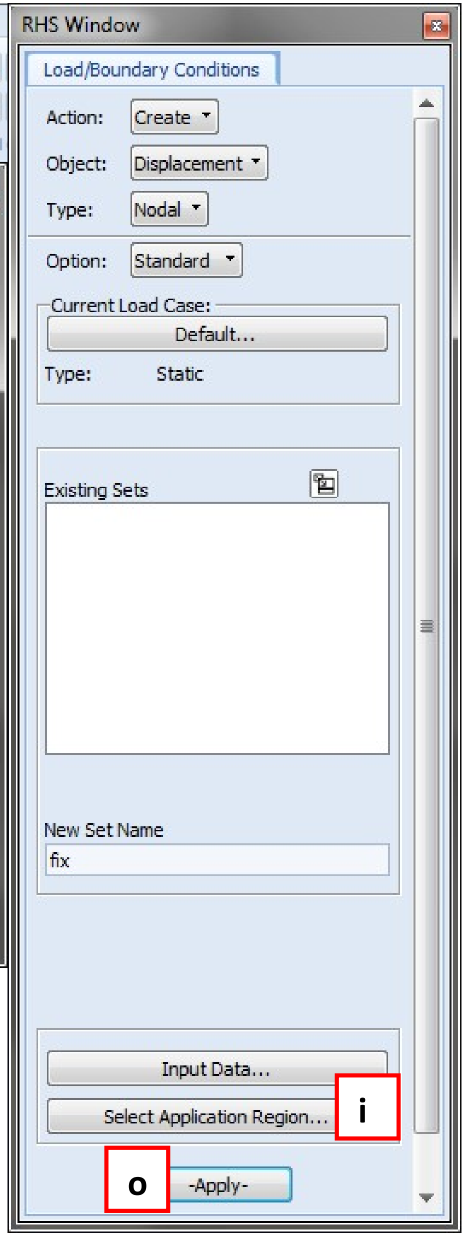


a. Click on the **Front view** icon
b. Click on the **Fit view** icon

Apply the boundary conditions:

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

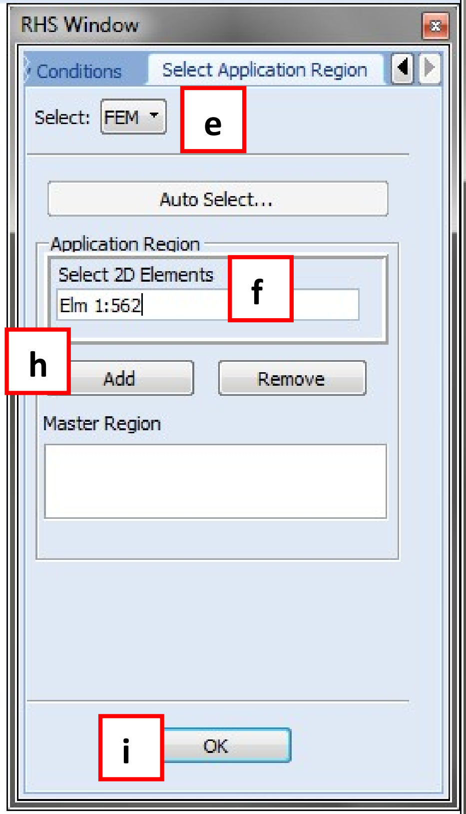


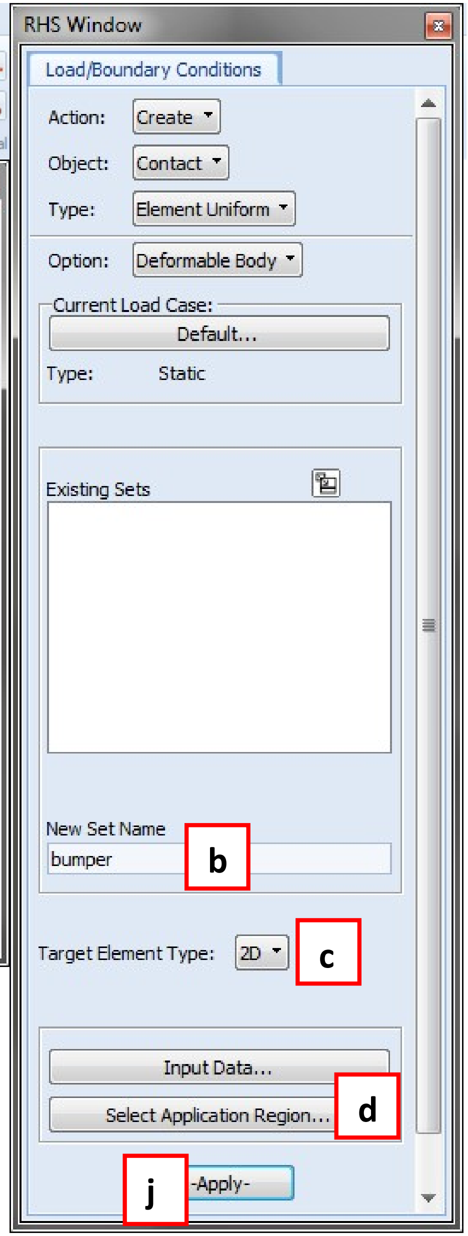


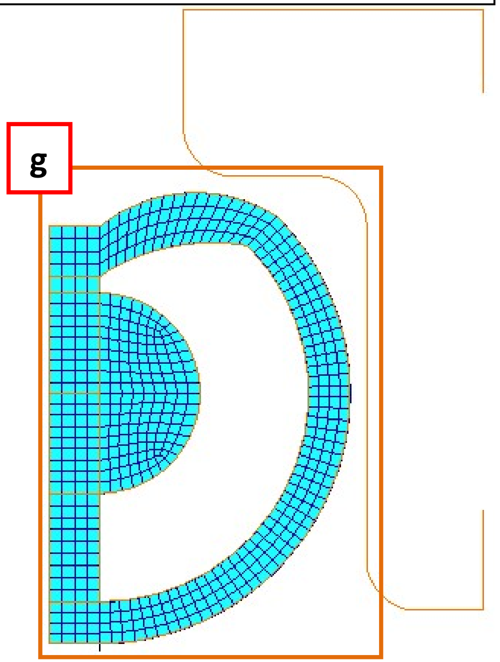
i. Click **Select Application Region…**
j. Select **FEM**
k. Click on the **Select Nodes** panel
l. Select the nodes (belonging to the left free edge of the bumper) by clicking and dragging the mouse
m. Click **Add**
n. Click **OK**
o. Click **Apply**

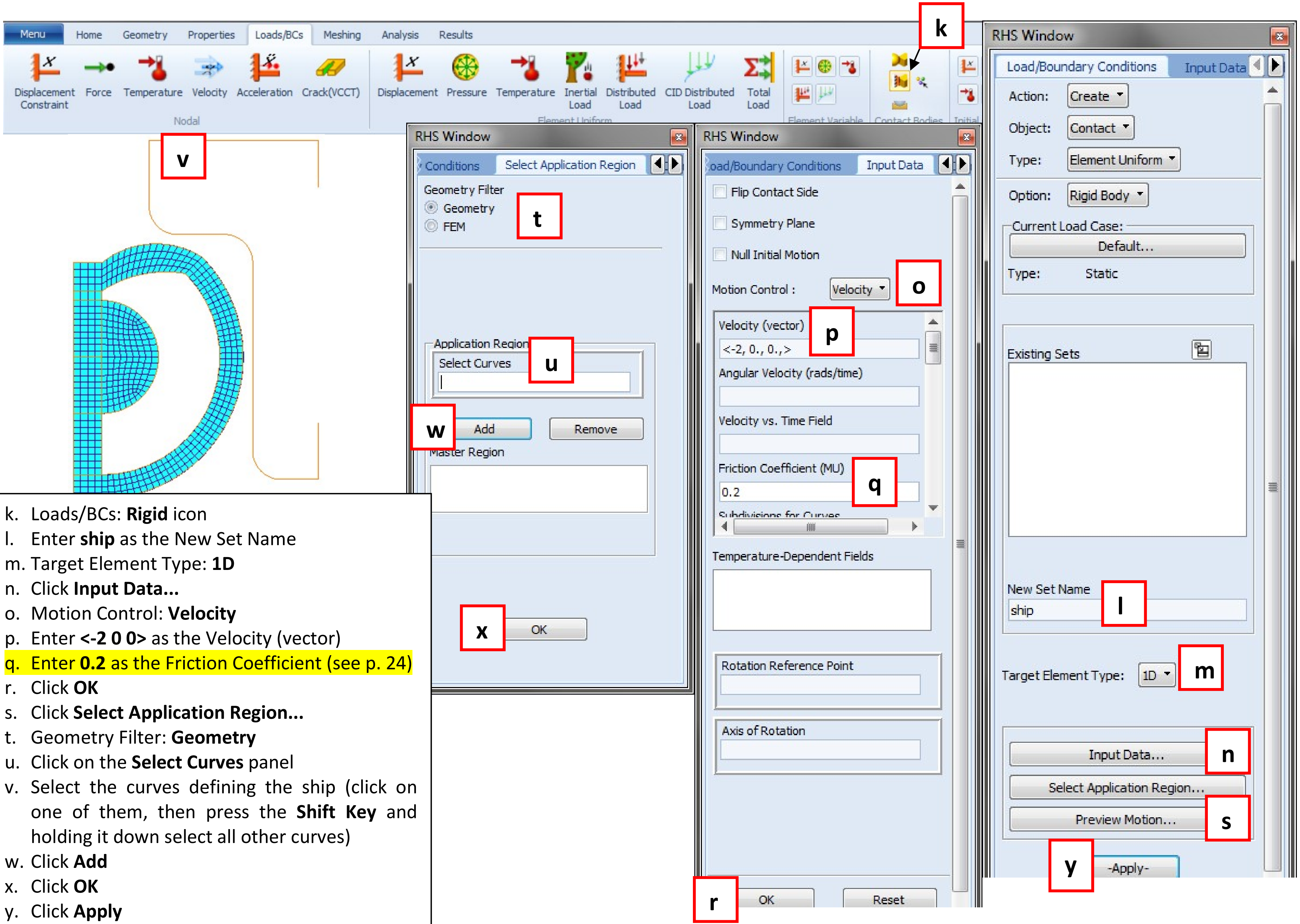
Define the contact between the rigid body (ship) and the rubber element (bumper):

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

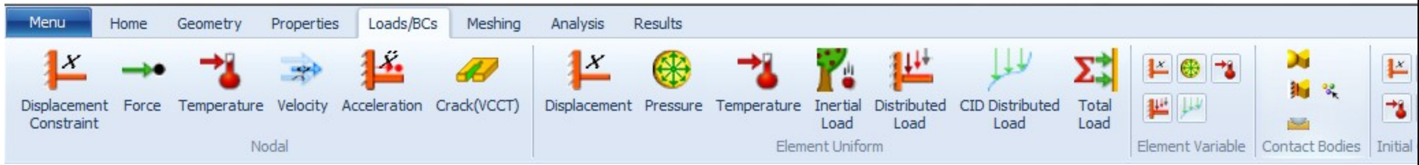








k. Loads/BCs: **Rigid** icon
l. Enter **ship** as the New Set Name
m. Target Element Type: **1D**
n. Click **Input Data...**
o. Motion Control: **Velocity**
p. Enter **<-2 0 0>** as the Velocity (vector)
q. Enter **0.2** as the Friction Coefficient (see p. 24)
r. Click **OK**
s. Click **Select Application Region...**
t. Geometry Filter: **Geometry**
u. Click on the **Select Curves** panel
v. Select the curves defining the ship (click on one of them, then press the **Shift Key** and holding it down select all other curves)
w. Click **Add**
x. Click **OK**
y. Click **Apply**

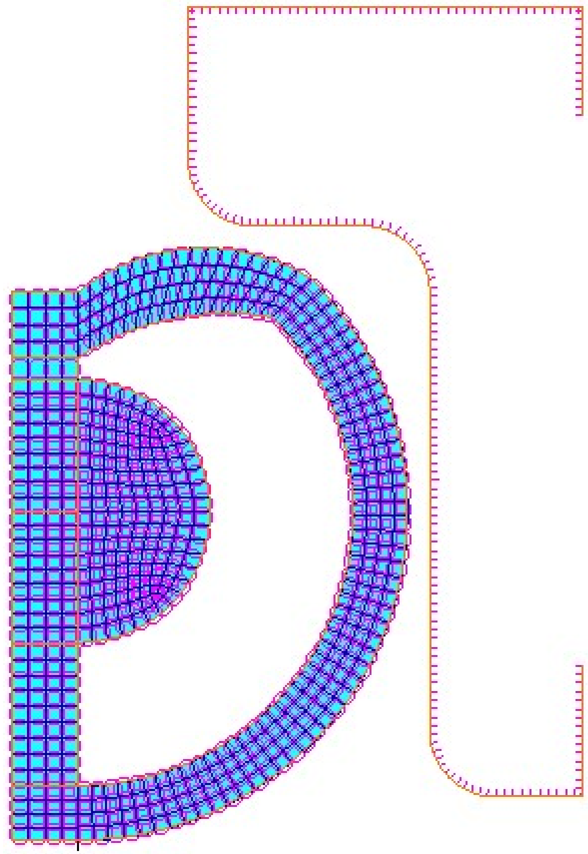


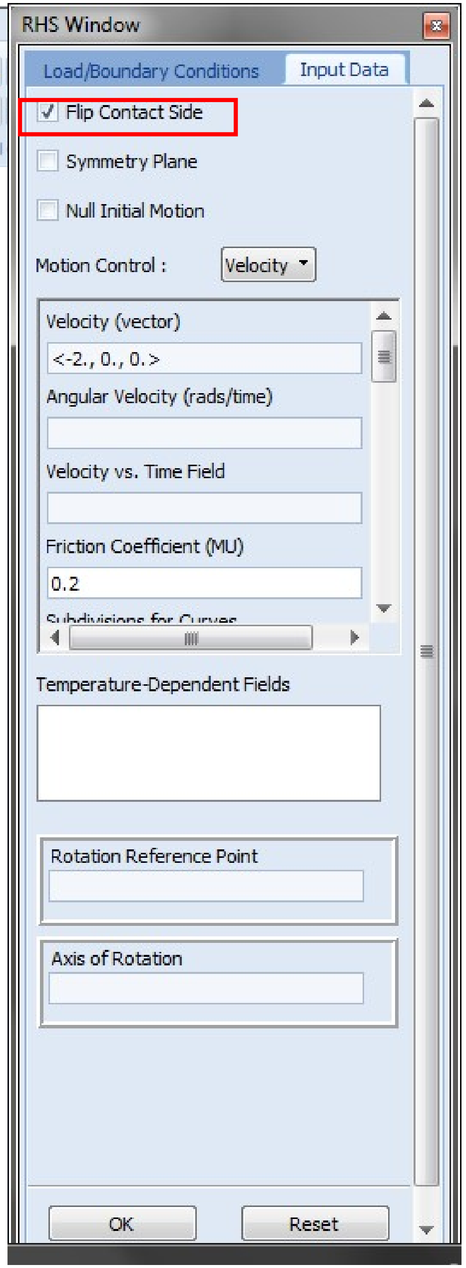
Verify your model:

a. Check if your model looks like the one shown on the right

<u>Remark</u>: If you find out that the arrows point the wrong way (i.e. pointing away from the ship) after creating the rigid contact curves, modify this LBC set (**Modify/Contact/Element Uniform**):

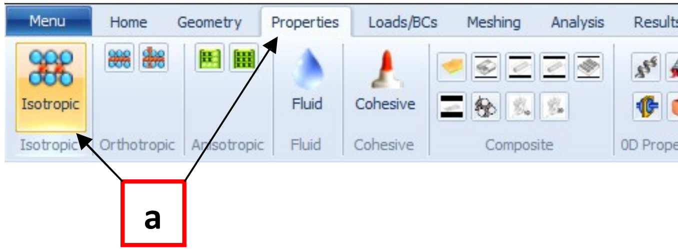
- Select **ship**
- Click **Modify Data…**
- Toggle **Flip Contact Side** to **ON**
- Accept the modifications





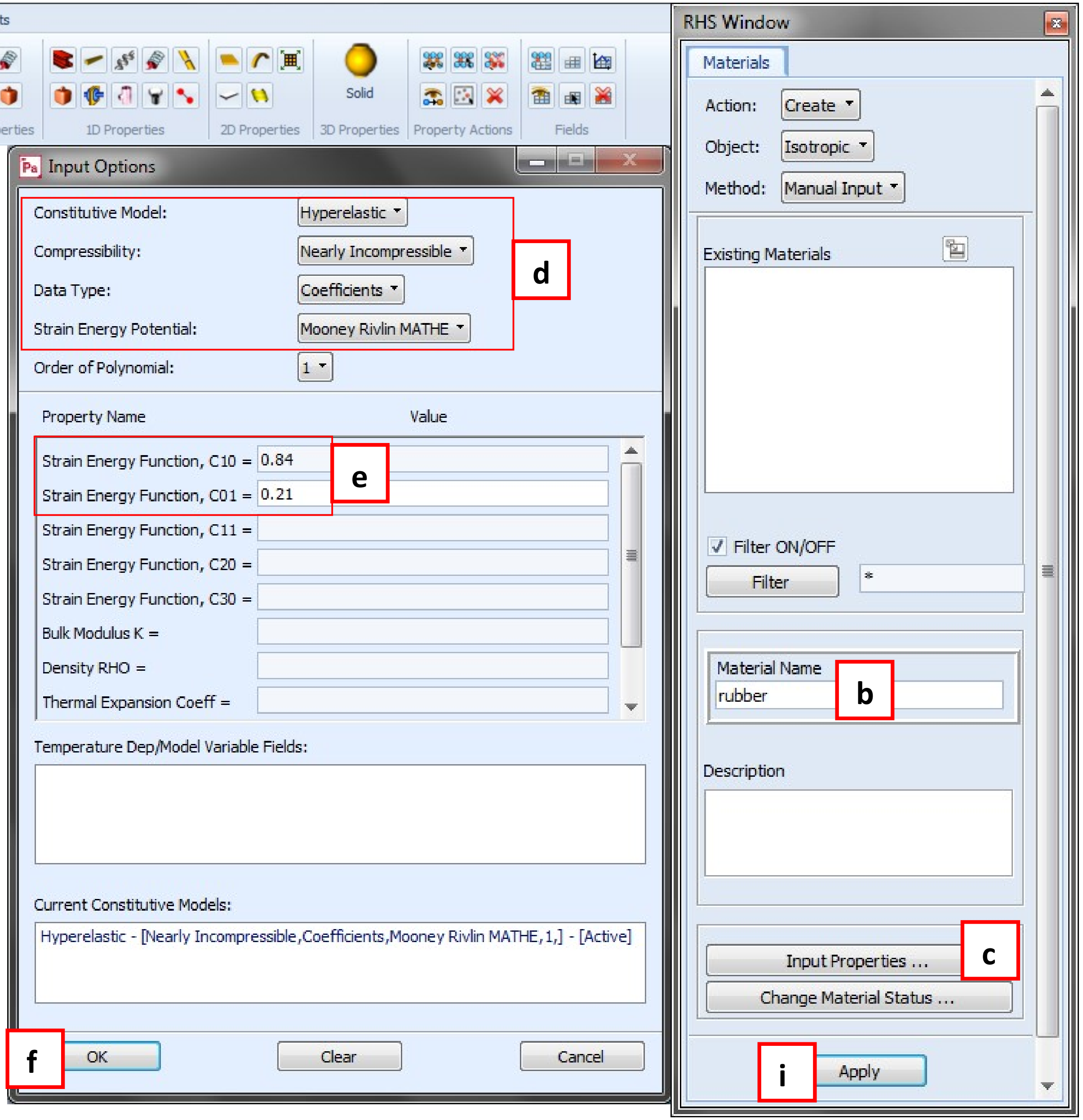
HUMAN CAPITAL
NATIONAL COHESION STRATEGY

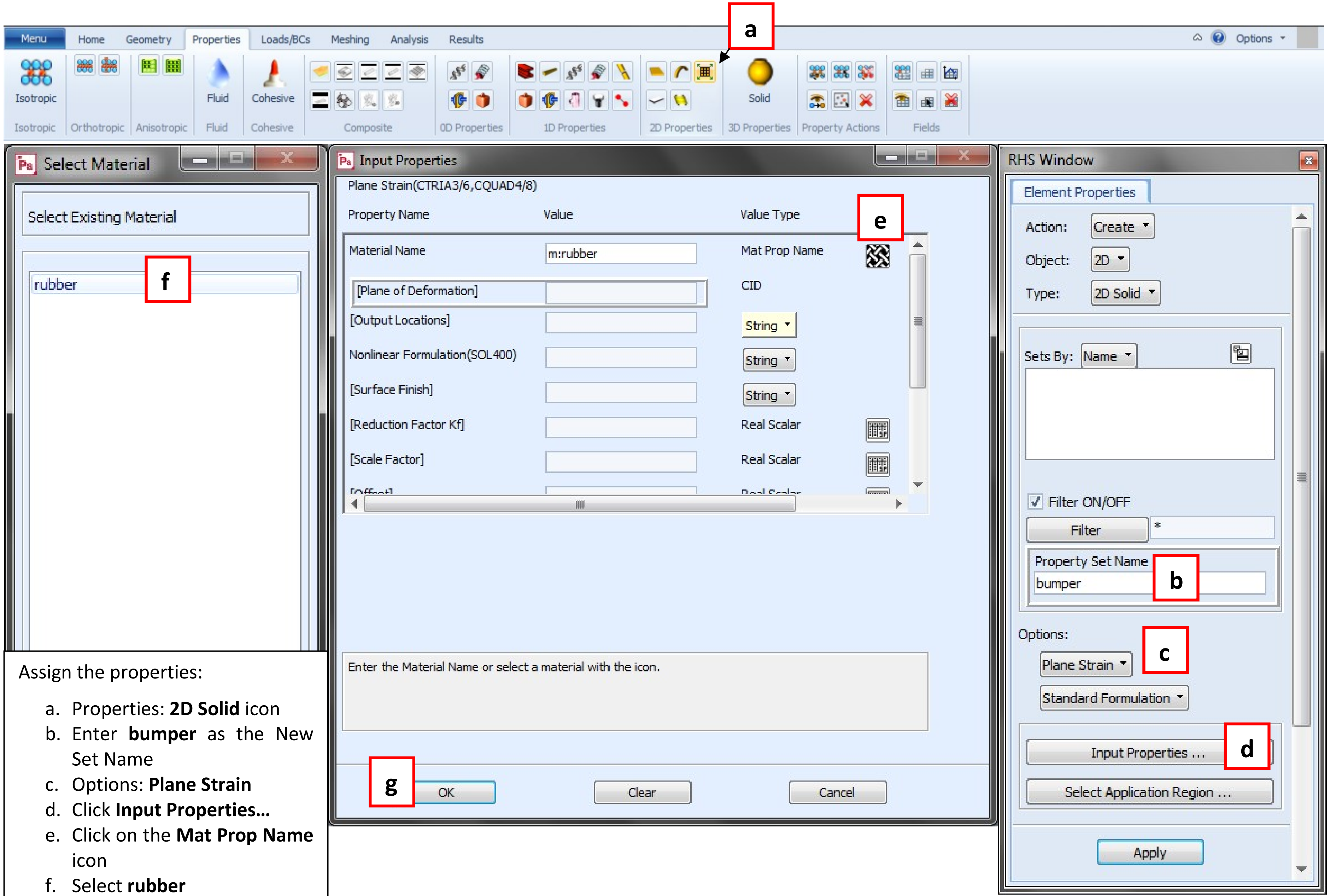
EUROPEAN UNION
EUROPEAN
SOCIAL FUND



Define a material:

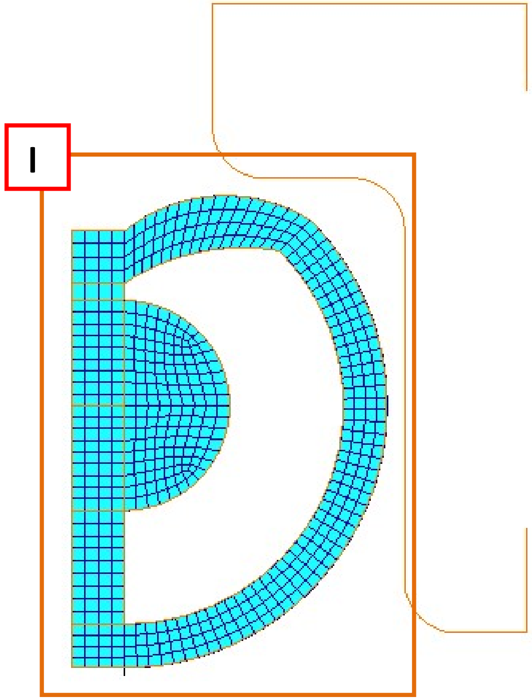
a. Click on the Properties: **Isotropic** icon
b. Enter **rubber** as the Material Name
c. Click **Input Properties…**
d. Change *Constitutive Model* to **Hyperelastic**
   Change *Data Type* to **Coefficients**
   Select **Mooney Rivlin MATHE**
e. Enter **0.84** as the C10 coefficient and **0.21** as the C01 coefficient
f. Click **OK**

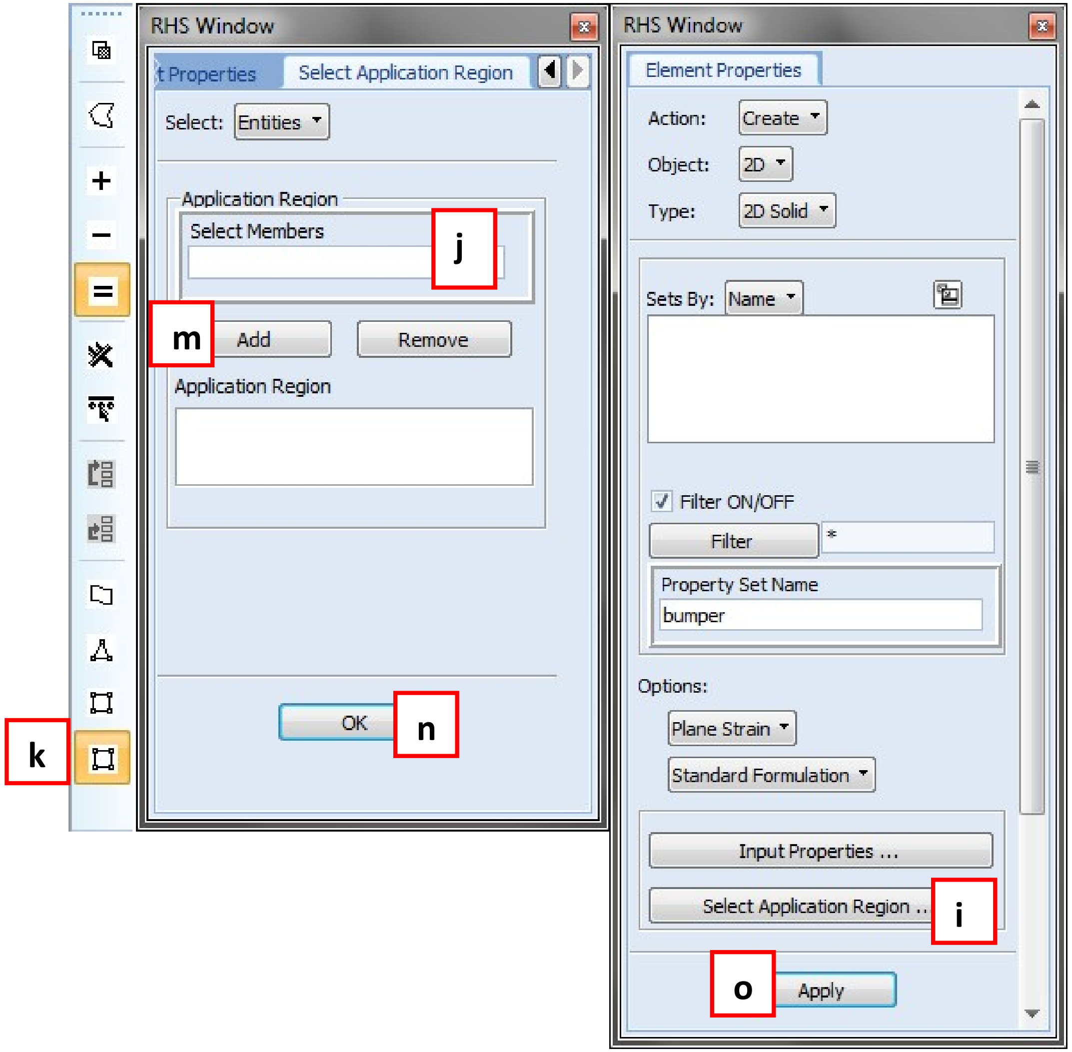




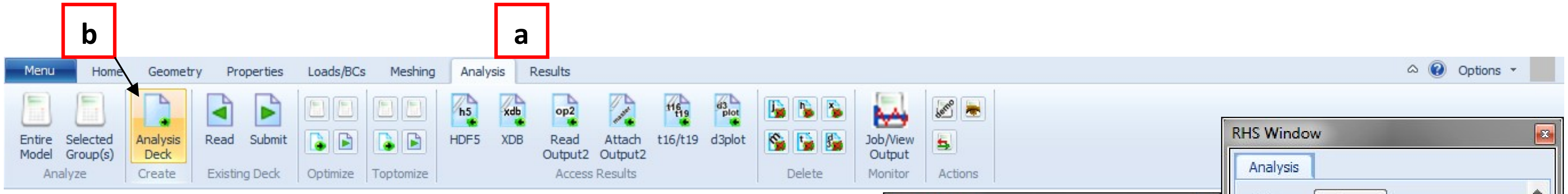
Assign the properties:

a. Properties: **2D Solid** icon
b. Enter **bumper** as the New Set Name
c. Options: **Plane Strain**
d. Click **Input Properties…**
e. Click on the **Mat Prop Name** icon
f. Select **rubber**
g. Click **OK**





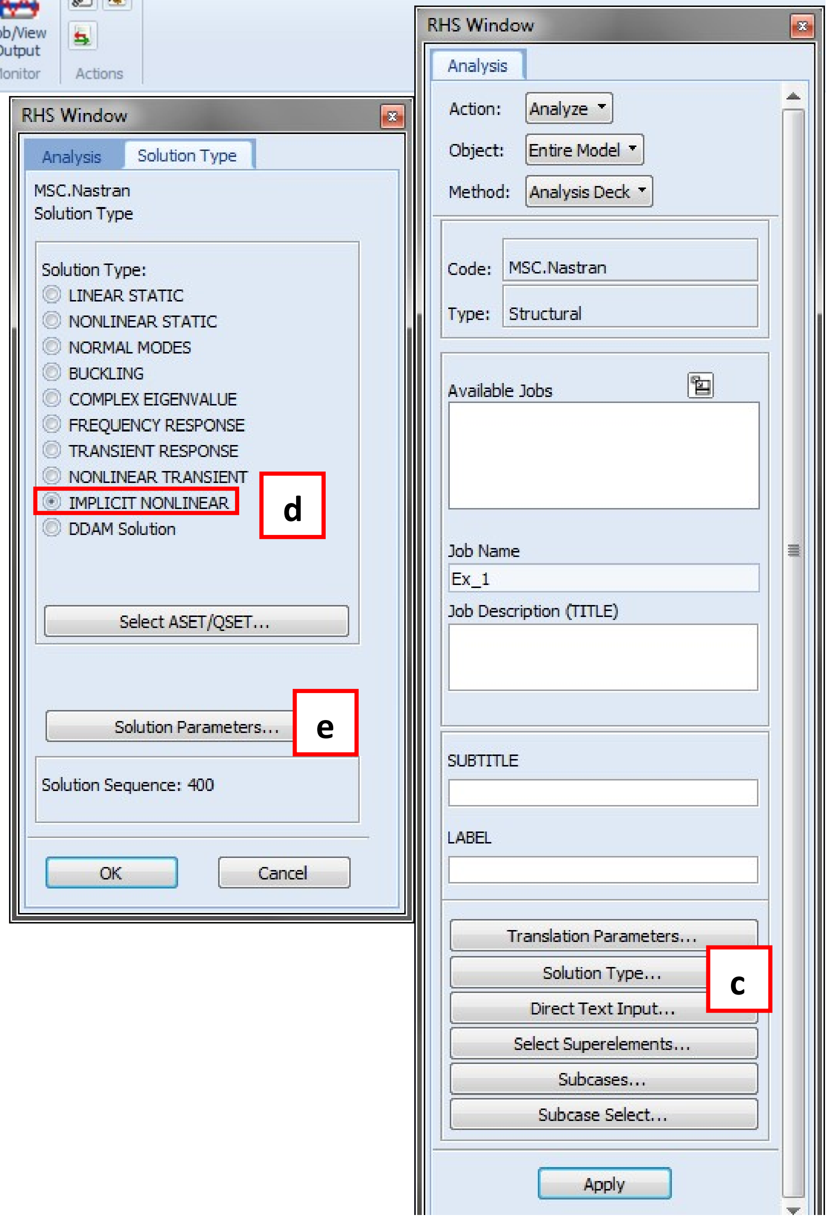
i. Click **Select Application Region…**
j. Click on the **Select Members** panel
k. Select **Shell element** icon
l. Select all shell elements by clicking and dragging the mouse
m. Click **Add**
n. Click **OK**
o. Click **Apply**

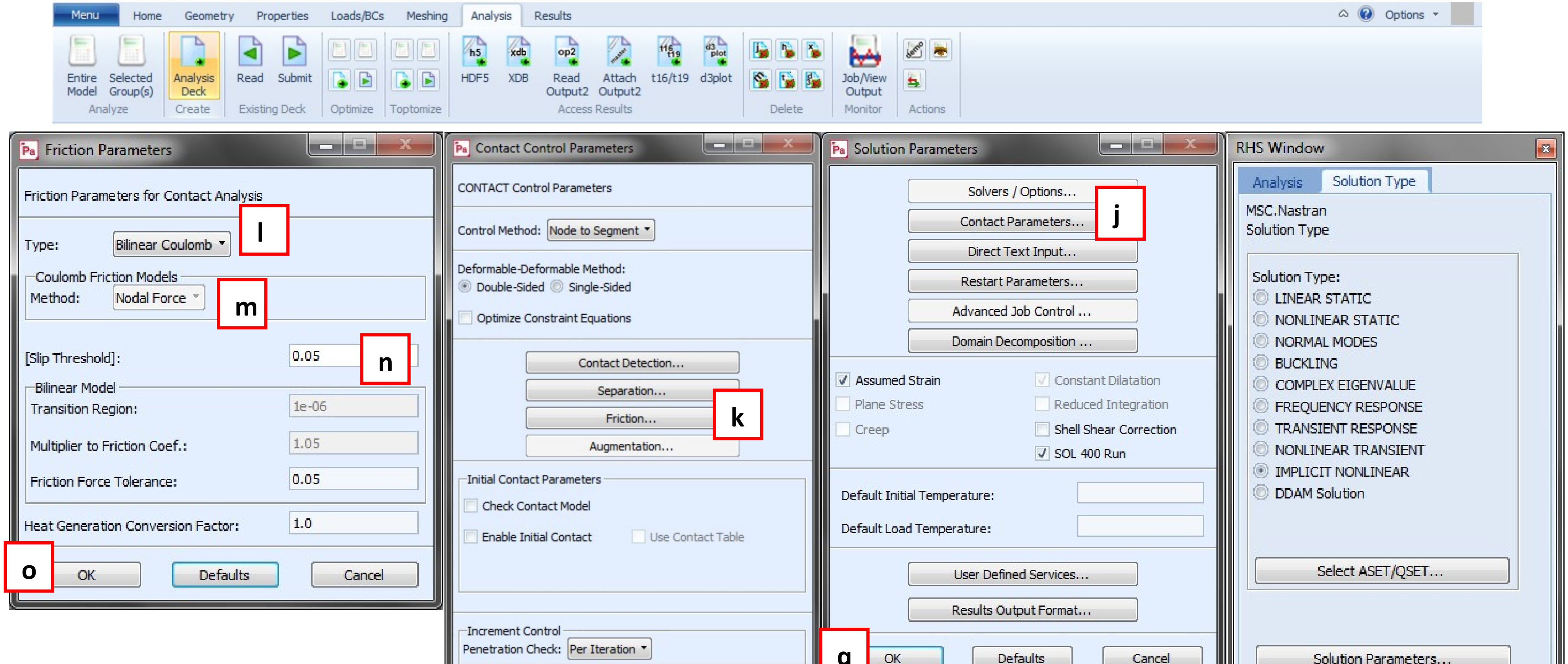


Run a nonlinear analysis:

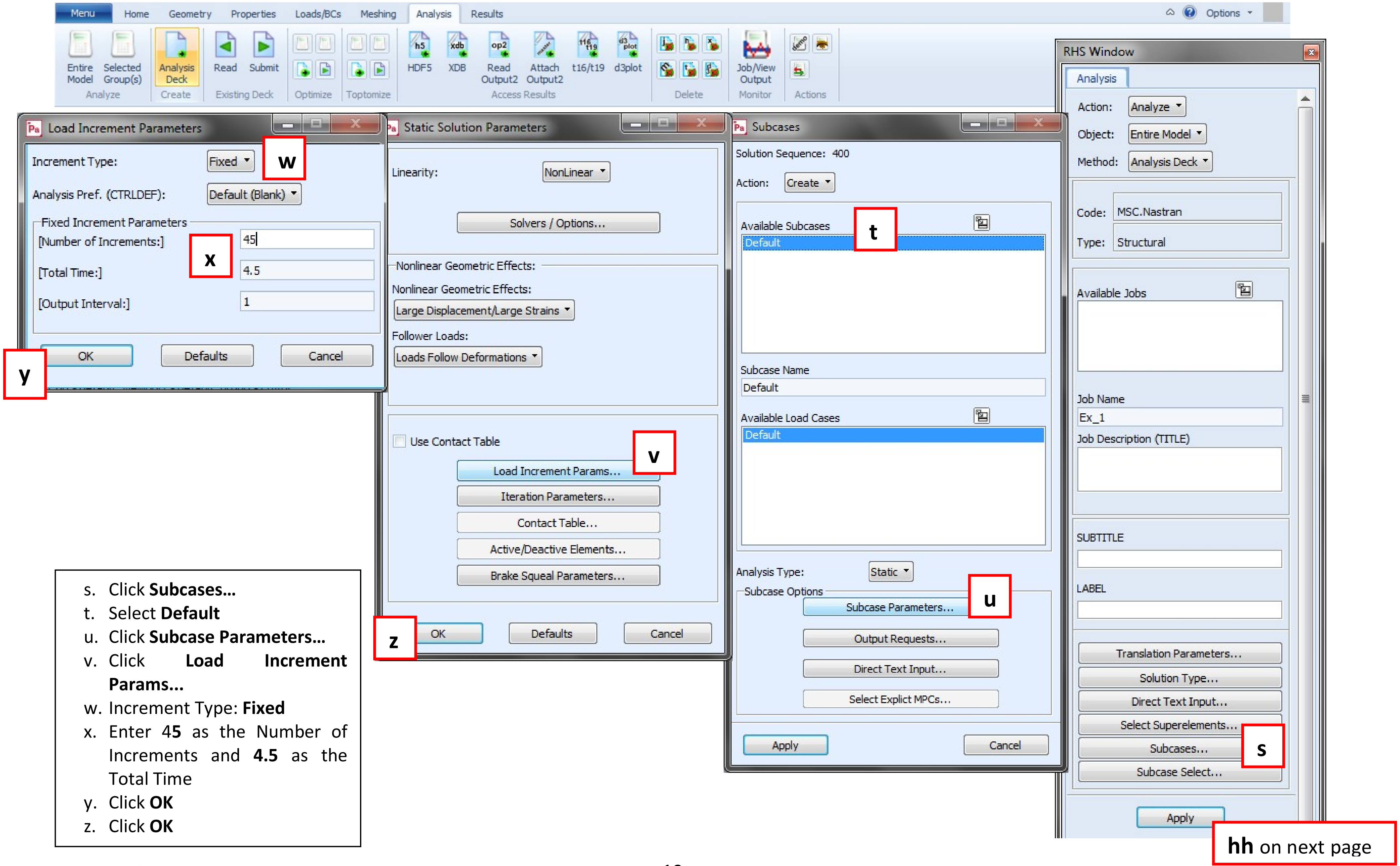
a. Click on the **Analysis** icon
b. Click on the **Analysis Deck** icon
c. Click **Solution Type…**
d. Select **IMPLICIT NONLINEAR** as the Solution Type
e. Click **Solution Parameters…**

f. – i. *Removed*

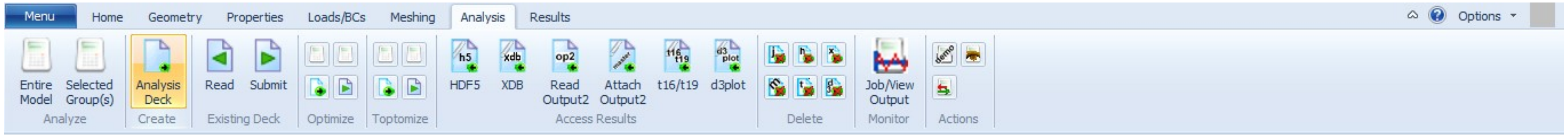




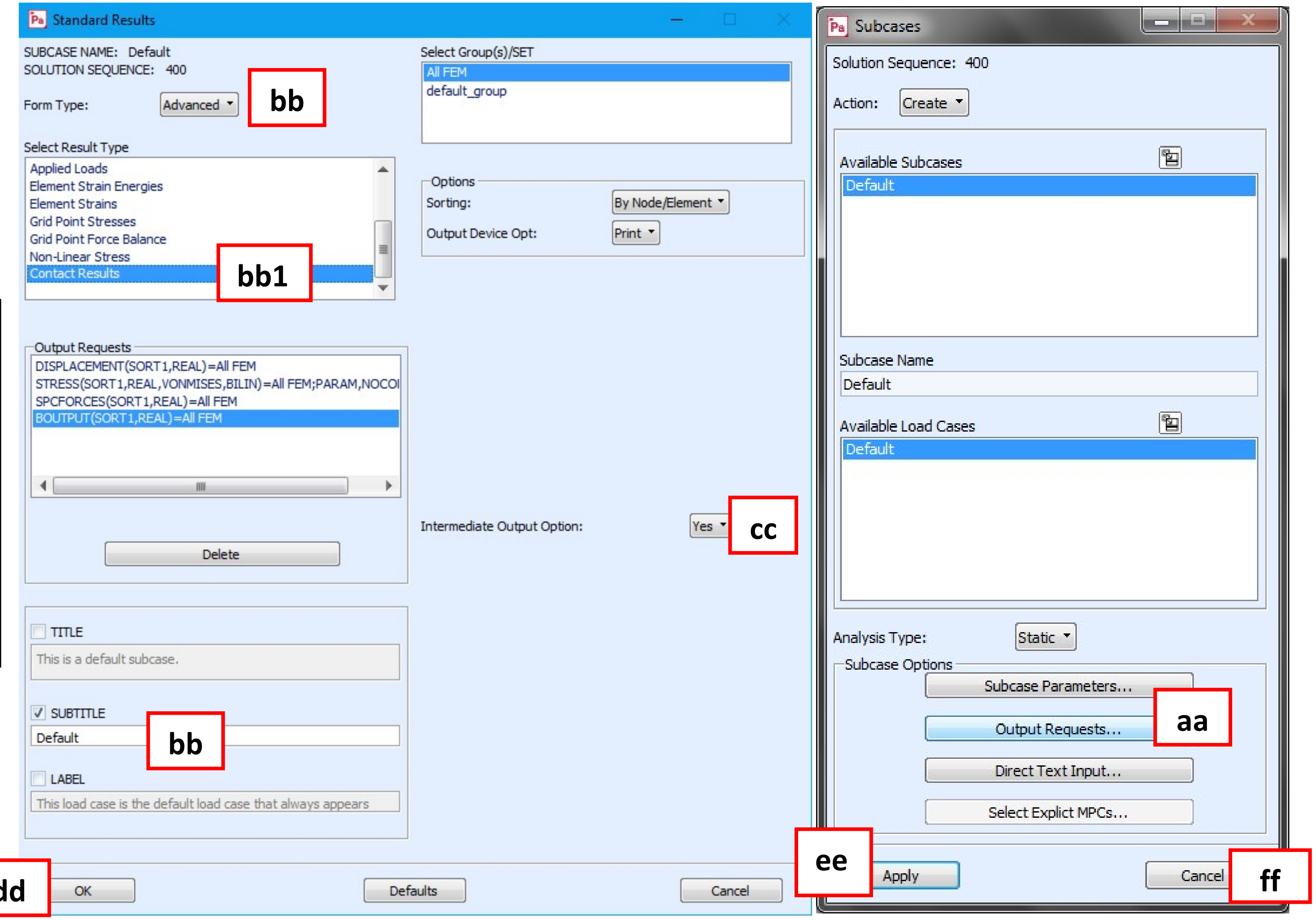
j. Click **Contact Parameters…**
k. Click **Friction…**
l. Type: **Bilinear Coulomb**
m. Method: **Nodal Force**
n. Enter **0.05** as the relative Sliding Velocity
o. Click **OK**
p. Click **OK**
q. Click **OK**
r. Click **OK**

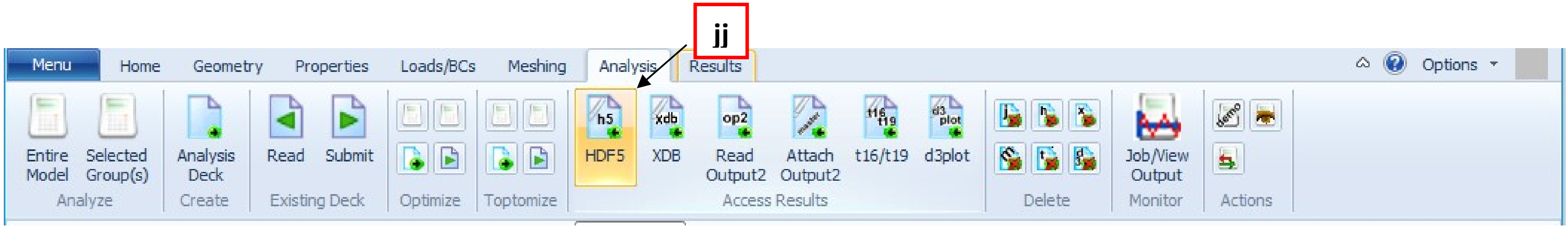


s. Click **Subcases...**
t. Select **Default**
u. Click **Subcase Parameters...**
v. Click **Load Increment Params...**
w. Increment Type: **Fixed**
x. Enter **45** as the Number of Increments and **4.5** as the Total Time
y. Click **OK**
z. Click **OK**



- aa. Click **Output Requests…**
- bb. Form Type: **Advanced**
- **bb1.** Select **"Contact Results"**
- cc. Intermediate Output Option: **Yes**
- dd. Click **OK**
- ee. Click **Apply**
- ff. Click **Cancel**
- gg. Click **Apply**
- hh. Click **Apply** (see page 19)

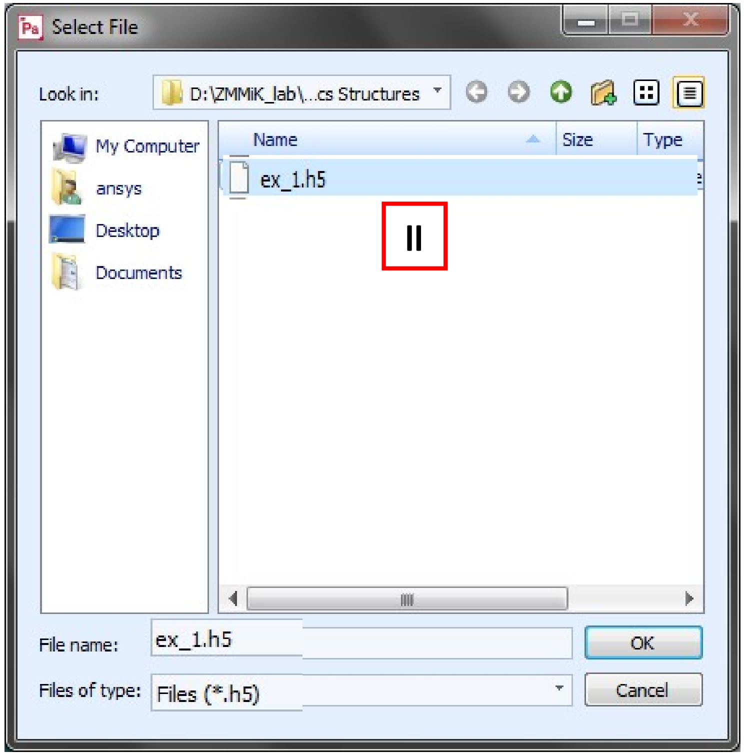


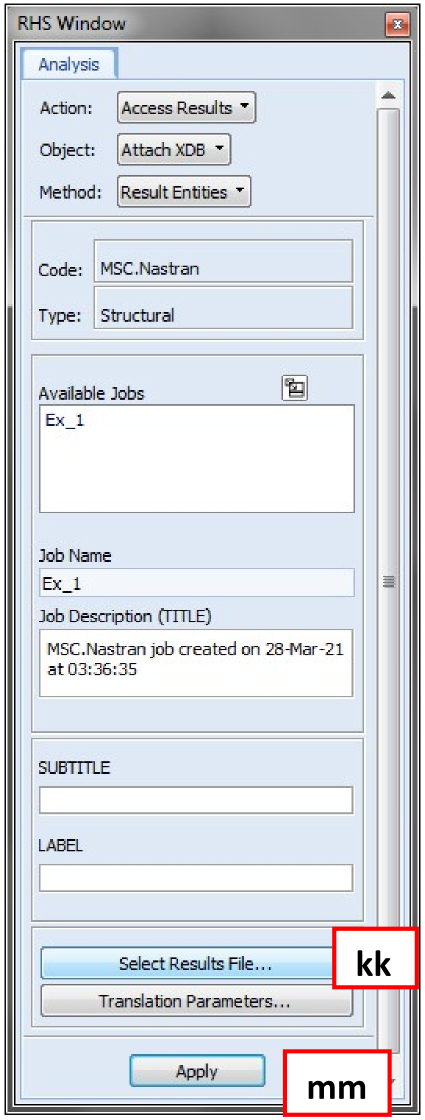


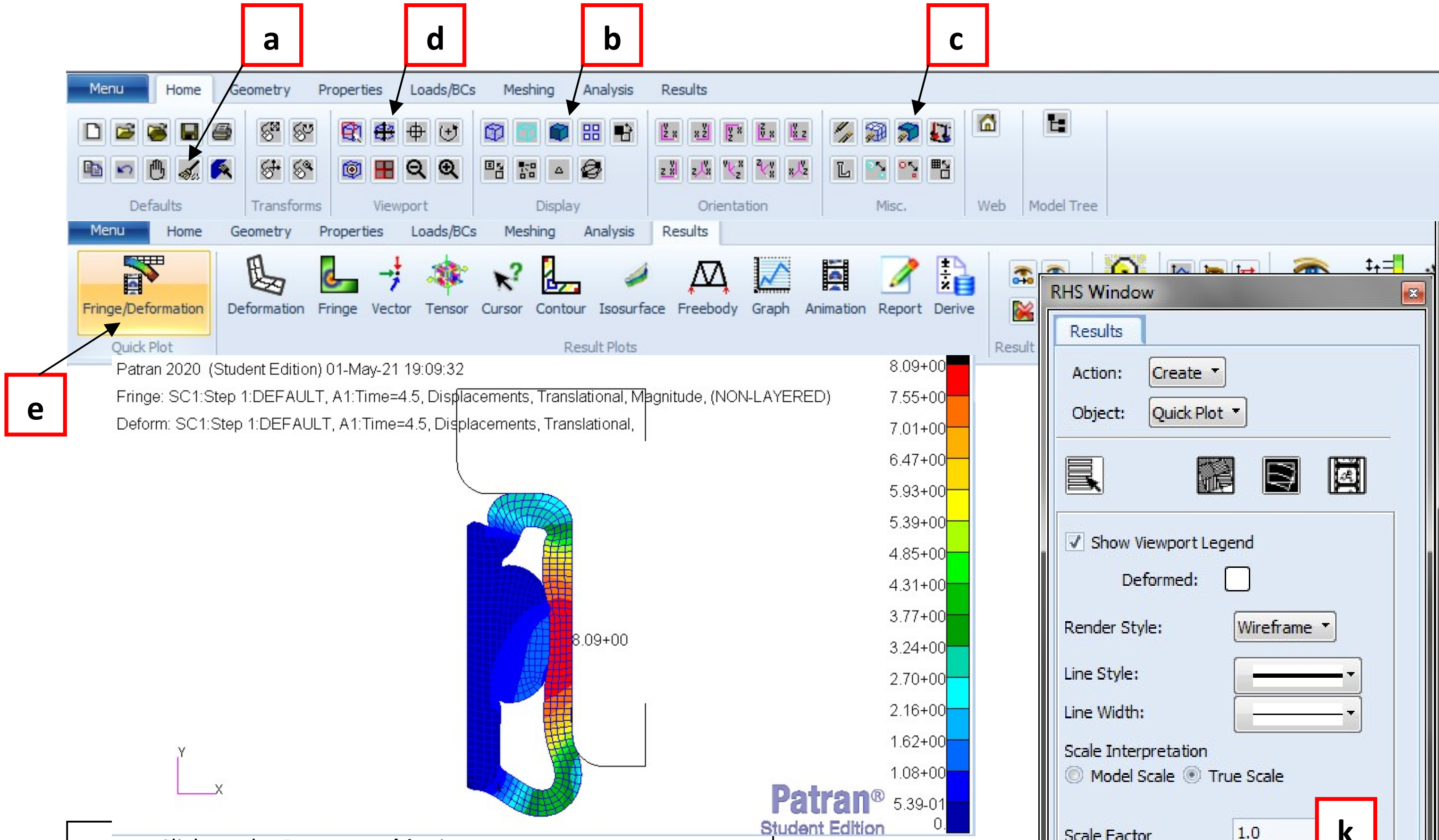
ii. Run **Nastran** analysis using **Ex_1.bdf** file from your working directory folder

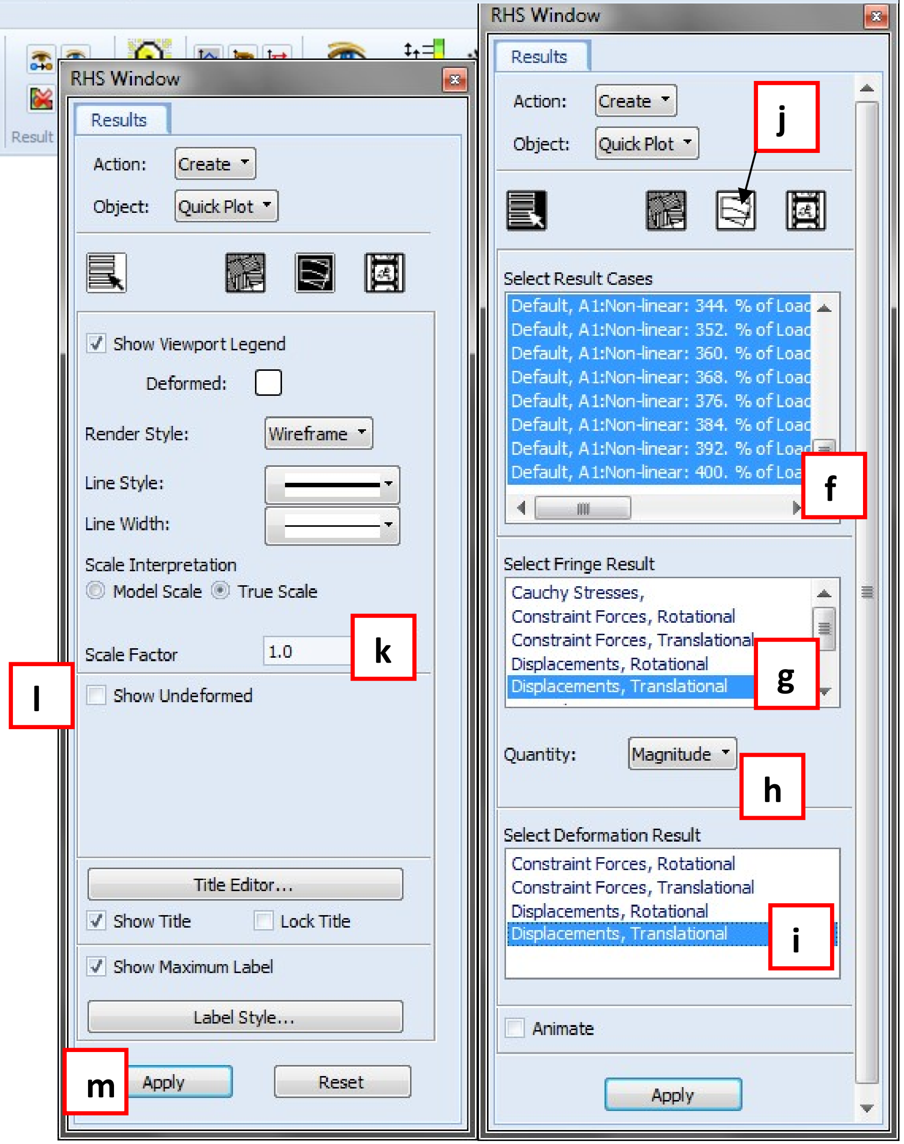
Attach the results file, when the analysis job is completed:

jj. Analysis: **HDF5** icon
kk. Click **Select Results File...**
ll. Select **ex_1.h5** file and click **OK**
mm. Click **Apply**





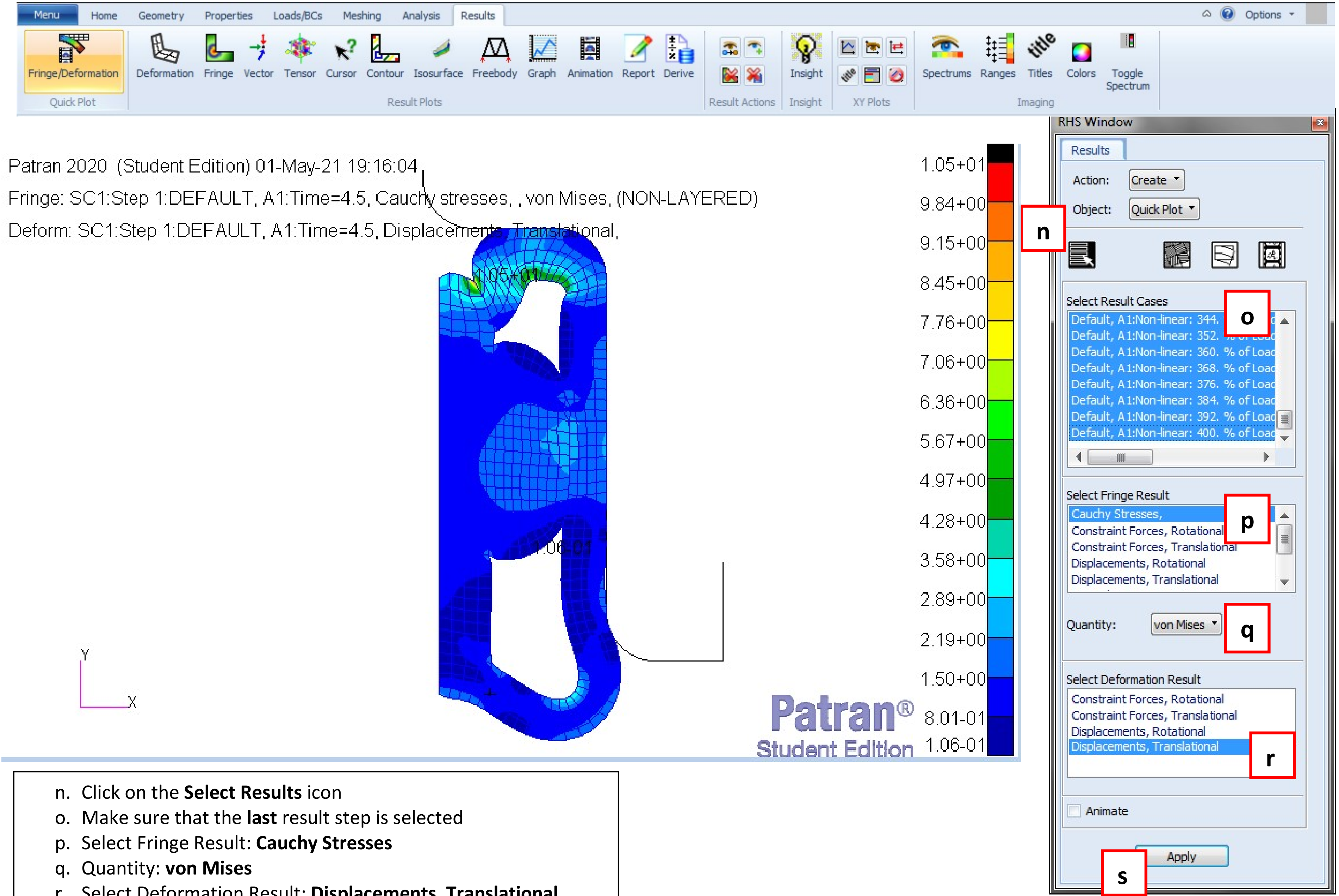




a. Click on the **Reset graphics** icon
b. Click on the **Smooth shaded** icon
c. Click on the **Plot/Erase Geometry** icon
d. Click on the **Fit view** icon

Post-process the results:

e. Click on the Results: **Fringe/Deformation** icon
f. Select **last** step
g. Select Fringe Result: **Displacements, Translational**
h. Quantity: **Magnitude**
i. Select Deformation Result: **Displacements, Translational**
j. Click on the **Deform Attributes** icon
k. Select **True Scale with the Scale Factor equals to 1.0**
l. Uncheck **Show Undeformed**
m. Click **Apply**



n. Click on the **Select Results** icon
o. Make sure that the **last** result step is selected
p. Select Fringe Result: **Cauchy Stresses**
q. Quantity: **von Mises**
r. Select Deformation Result: **Displacements, Translational**
s. Click **Apply**

And below – the same run, but WITHOUT FRICTION (slips down :-)