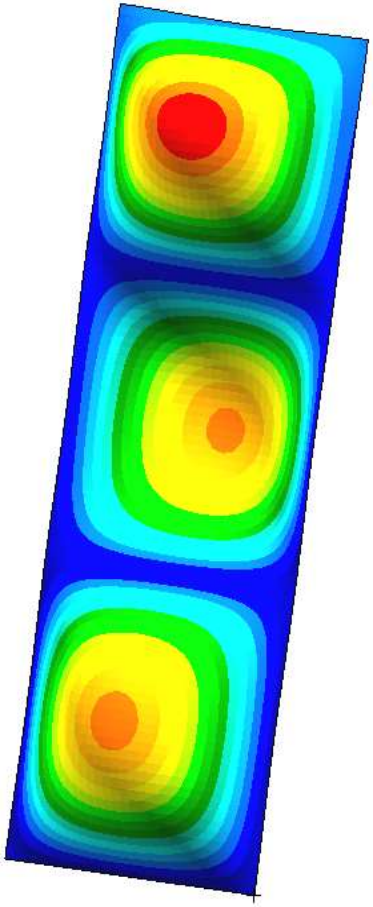
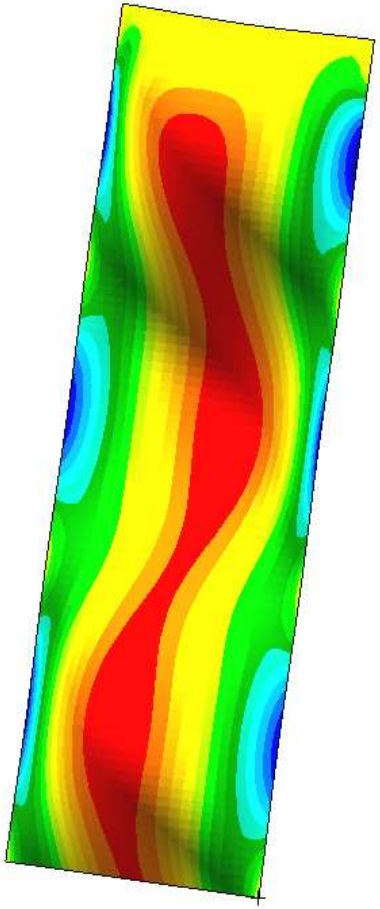
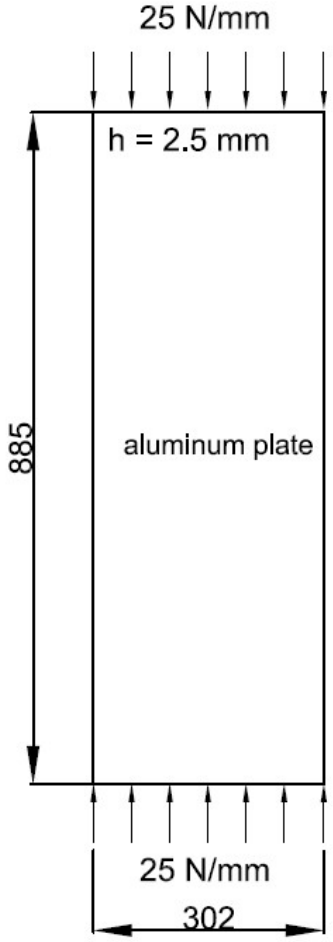
Mechanics of Thin-walled Structures



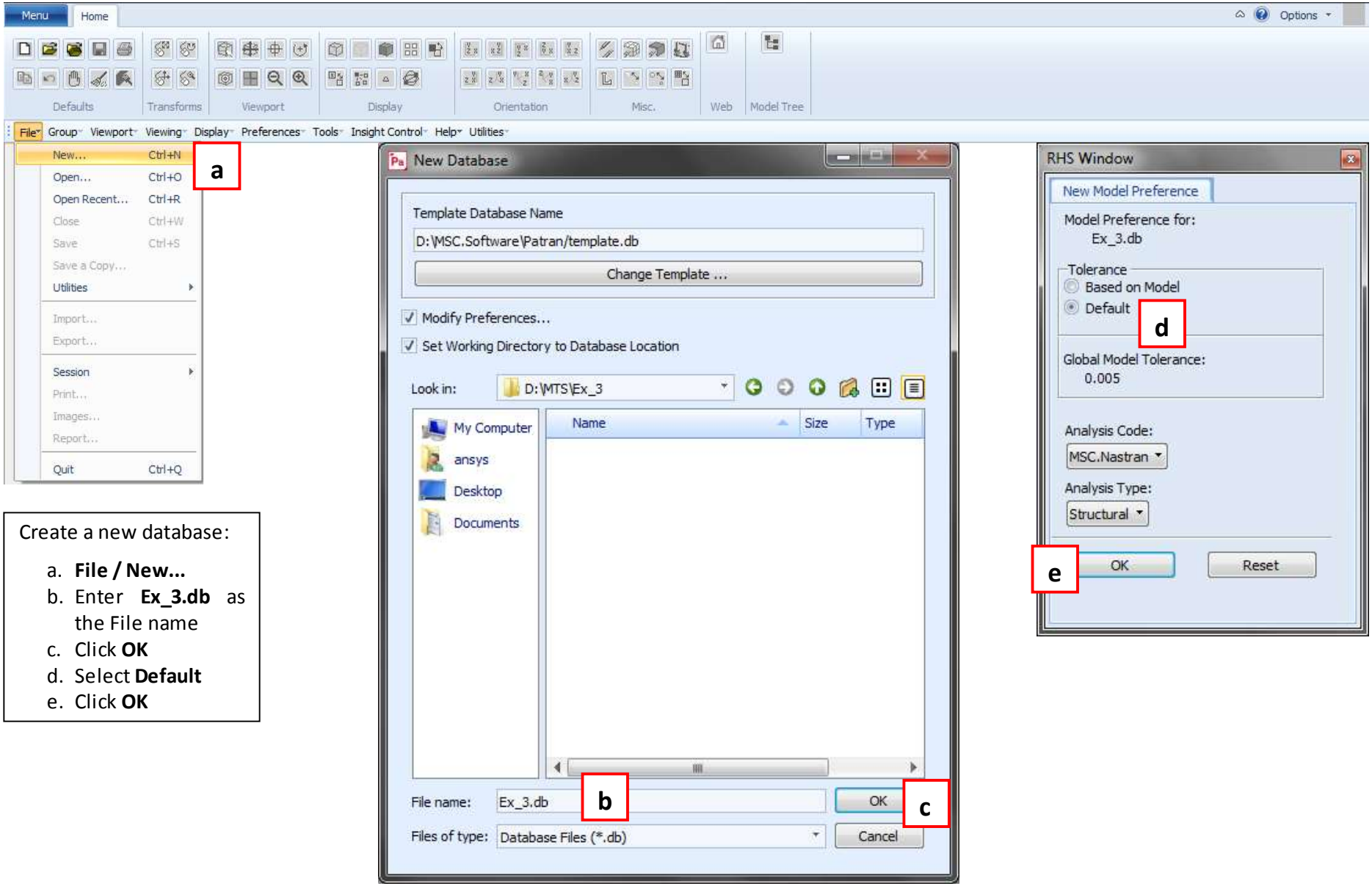


PROBLEM DESCRIPTION

The goal of this exercise is to conduct the buckling analysis of a compressed aluminum plate. The exercise is divided into two parts. In the first part the elastic buckling behavior of the plate is analyzed. In the second part – the behavior of the same plate after the loss of stability is assessed.



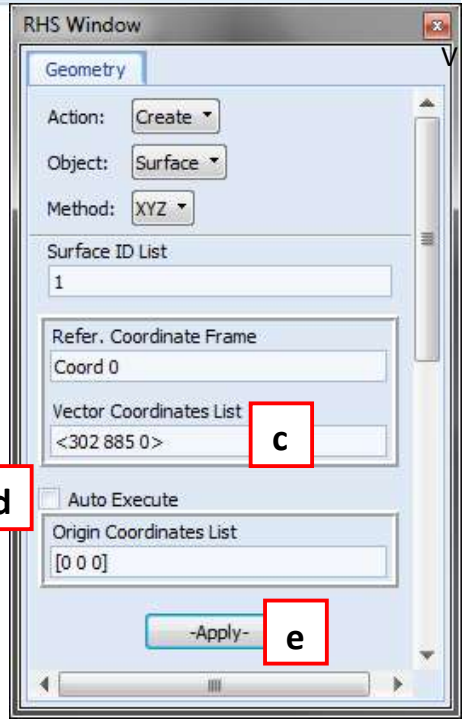
Units: mm, N, MPa



Create a new database:

a. **File / New...**
b. Enter **Ex_3.db** as the File name
c. Click **OK**
d. Select **Default**
e. Click **OK**

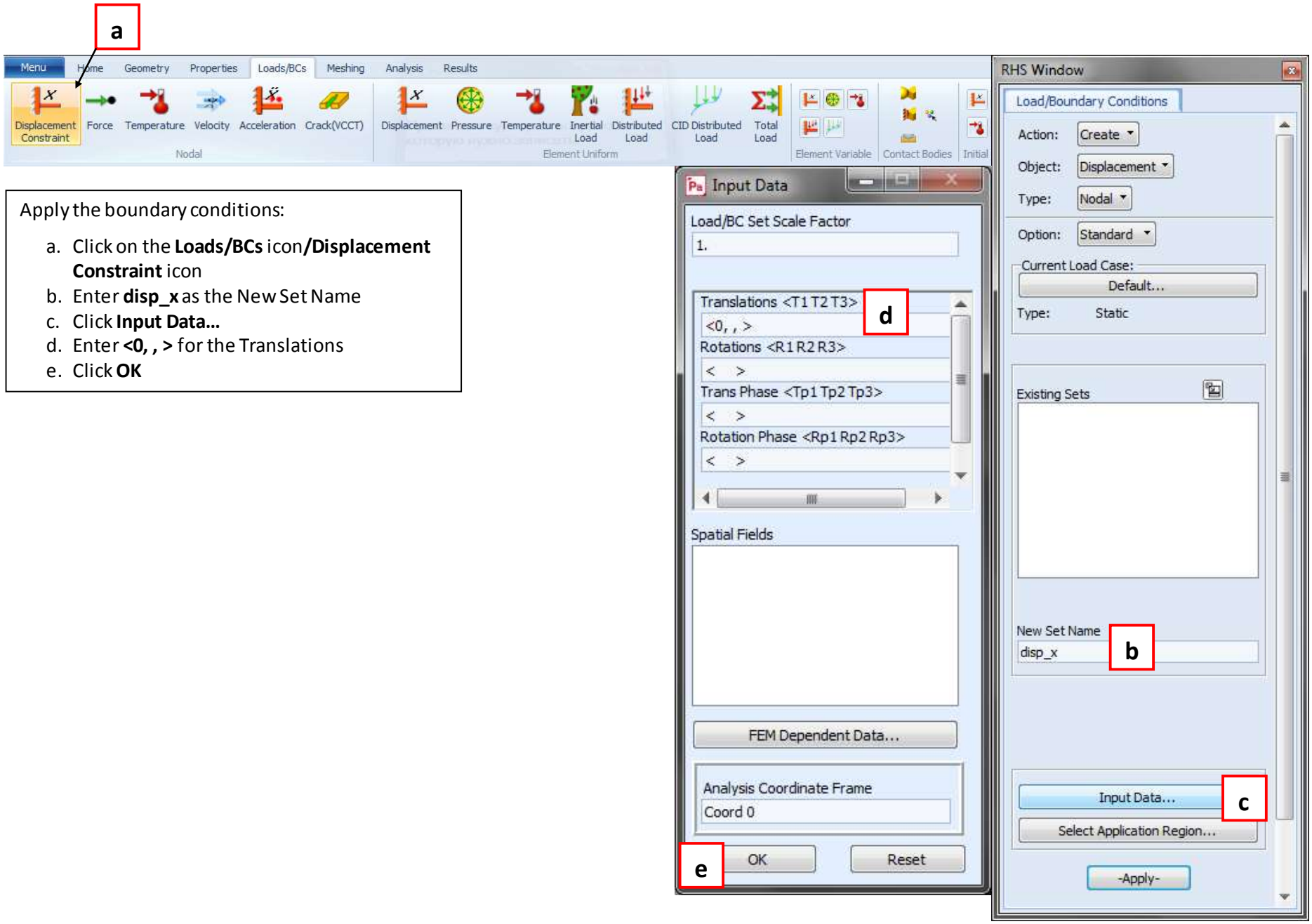
PART 1: BUCKLING ANALYSIS



a. Change *Viewport Color* to **Black** (click on the **Cycle Background** icon)

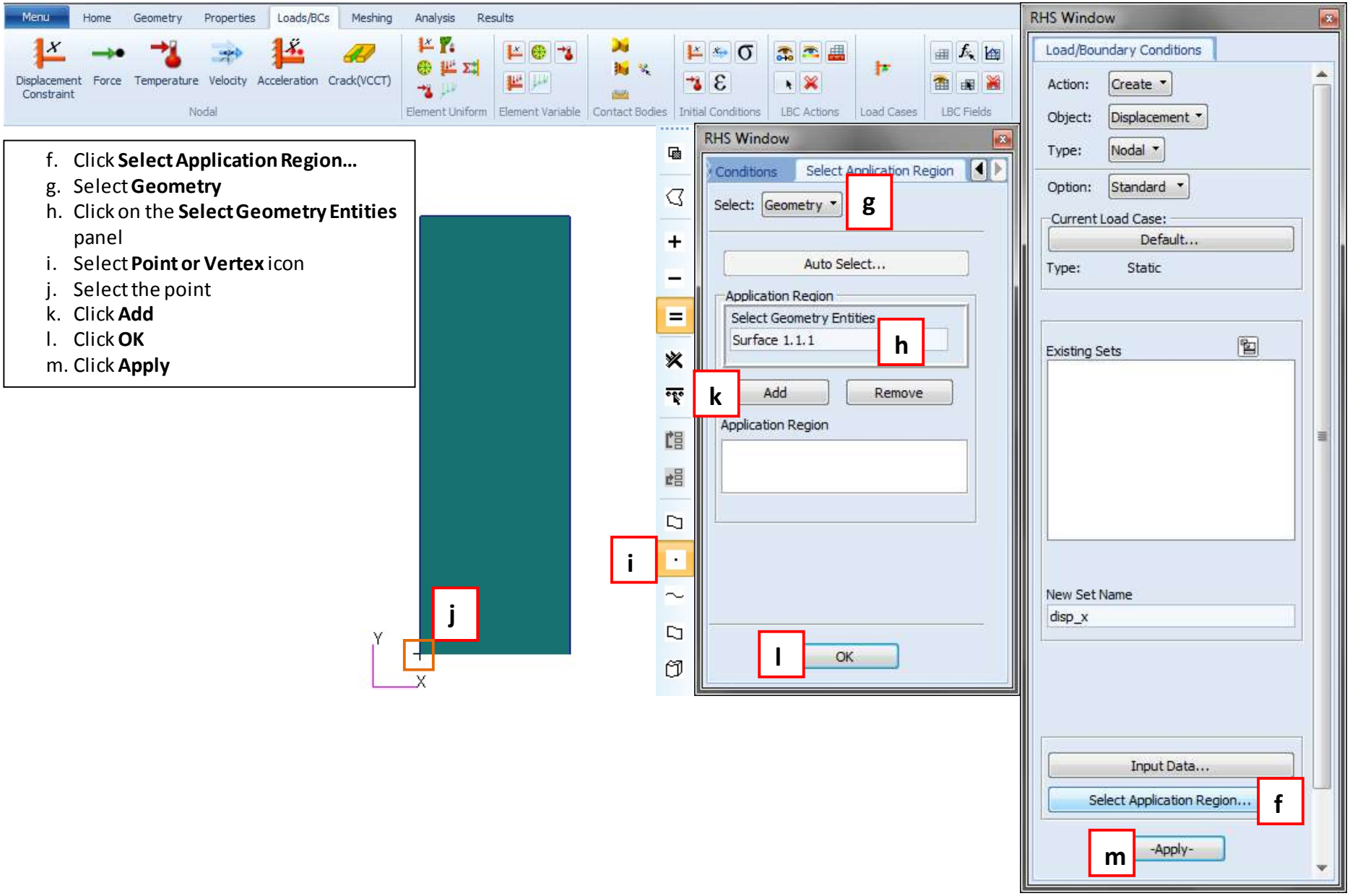
Create a surface:

b. Click on the **Geometry** icon**/Select/XYZ** *(Surfaces icon)*
c. Enter **<302 885 0>** as the Vector Coordinates List
d. Uncheck **Auto Execute**
e. Click **Apply**
f. Click on the **Smooth shaded** icon

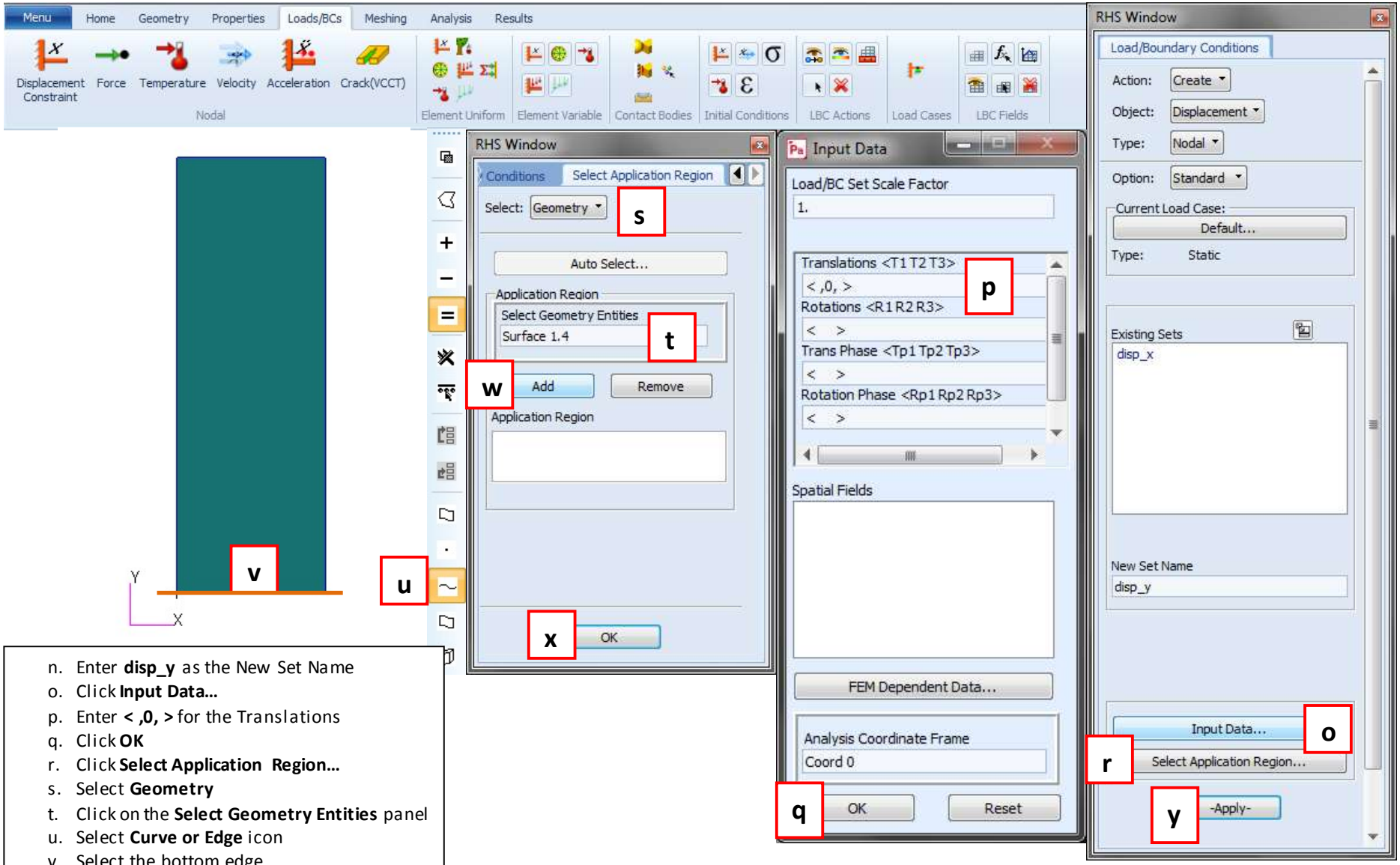


Apply the boundary conditions:

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



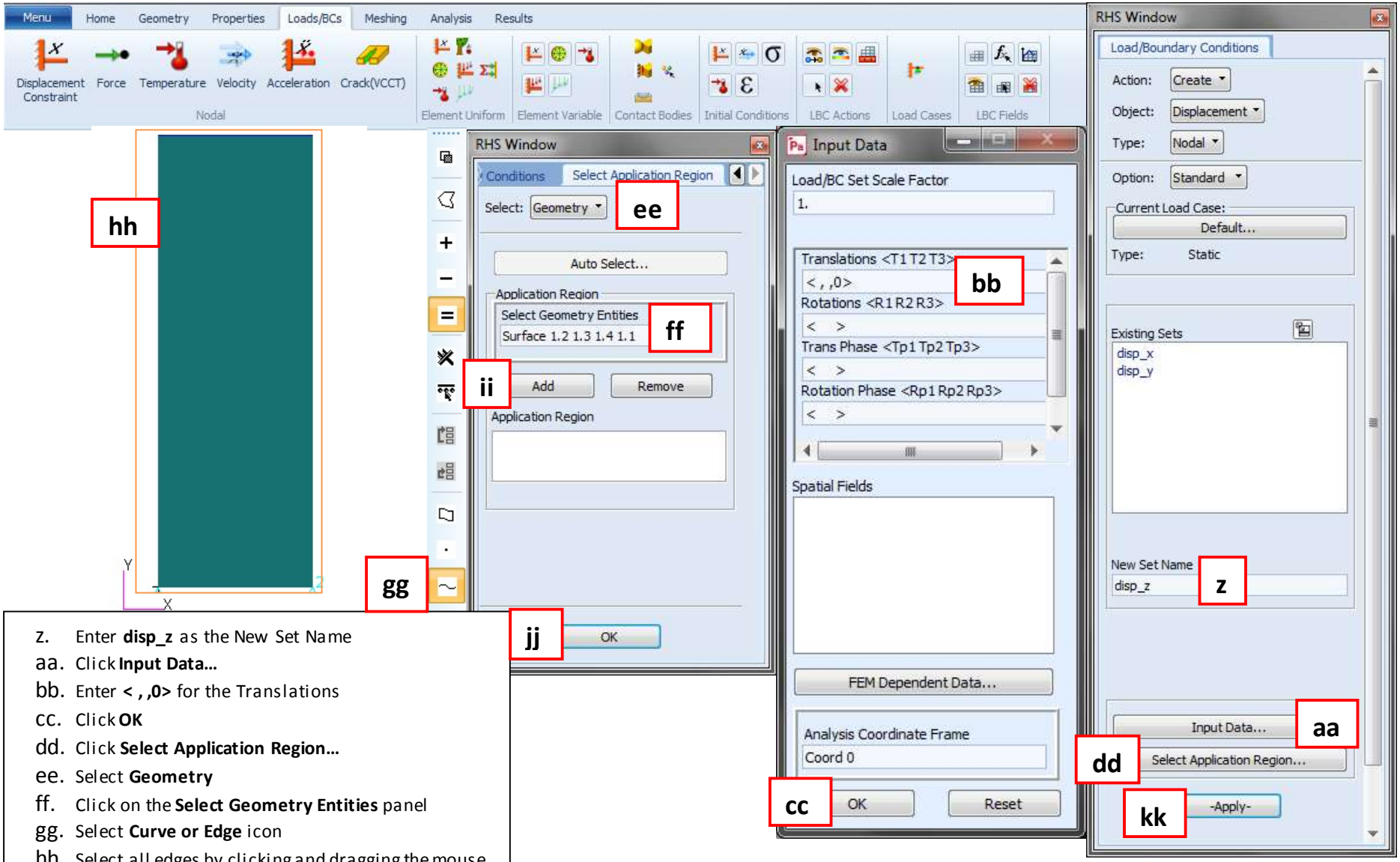
f. Click **Select Application Region…**
g. Select **Geometry**
h. Click on the **Select Geometry Entities** panel
i. Select **Point or Vertex** icon
j. Select the point
k. Click **Add**
l. Click **OK**
m. Click **Apply**



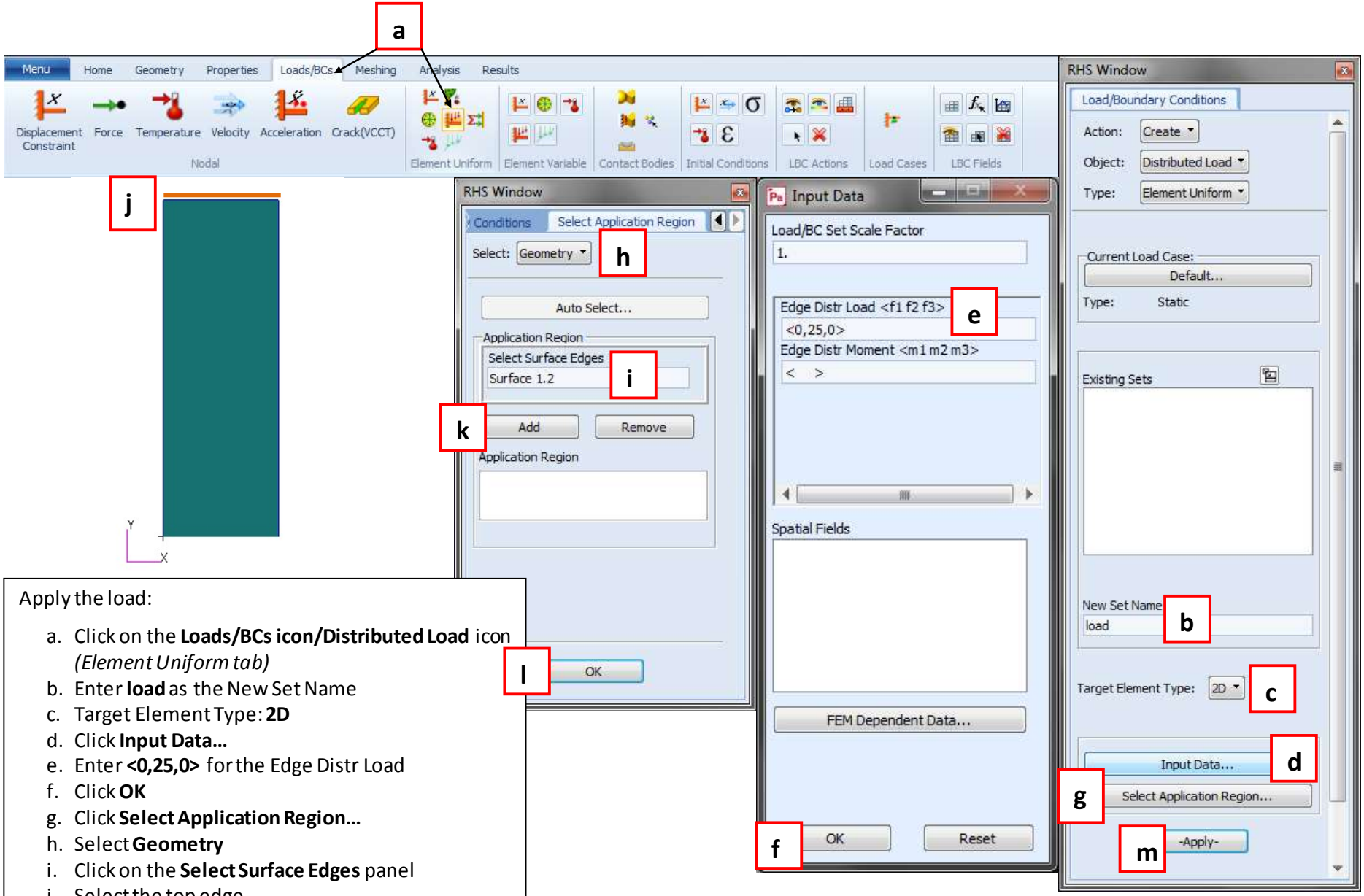
n. Enter **disp_y** as the New Set Name
o. Click **Input Data...**
p. Enter **< ,0, >** for the Translations
q. Click **OK**
r. Click **Select Application Region...**
s. Select **Geometry**
t. Click on the **Select Geometry Entities** panel
u. Select **Curve or Edge** icon
v. Select the bottom edge
w. Click **Add**
x. Click **OK**
y. Click **Apply**

HUMAN CAPITAL
NATIONAL COHESION STRATEGY

EUROPEAN UNION
EUROPEAN
SOCIAL FUND

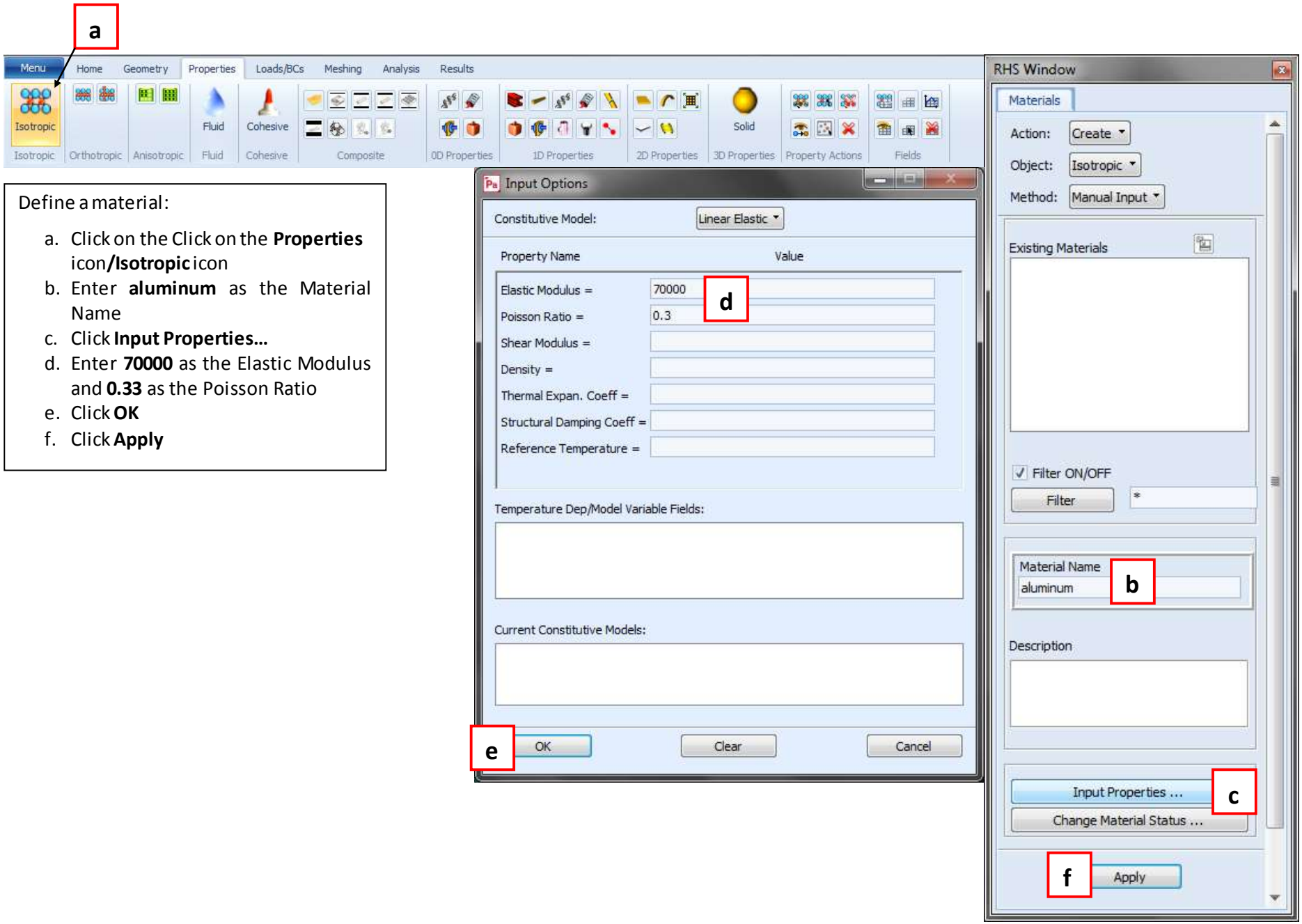


z. Enter **disp_z** as the New Set Name
aa. Click **Input Data…**
bb. Enter **< , ,0>** for the Translations
cc. Click **OK**
dd. Click **Select Application Region…**
ee. Select **Geometry**
ff. Click on the **Select Geometry Entities** panel
gg. Select **Curve or Edge** icon
hh. Select all edges by clicking and dragging the mouse
ii. Click **Add**
jj. Click **OK**
kk. Click **Apply**



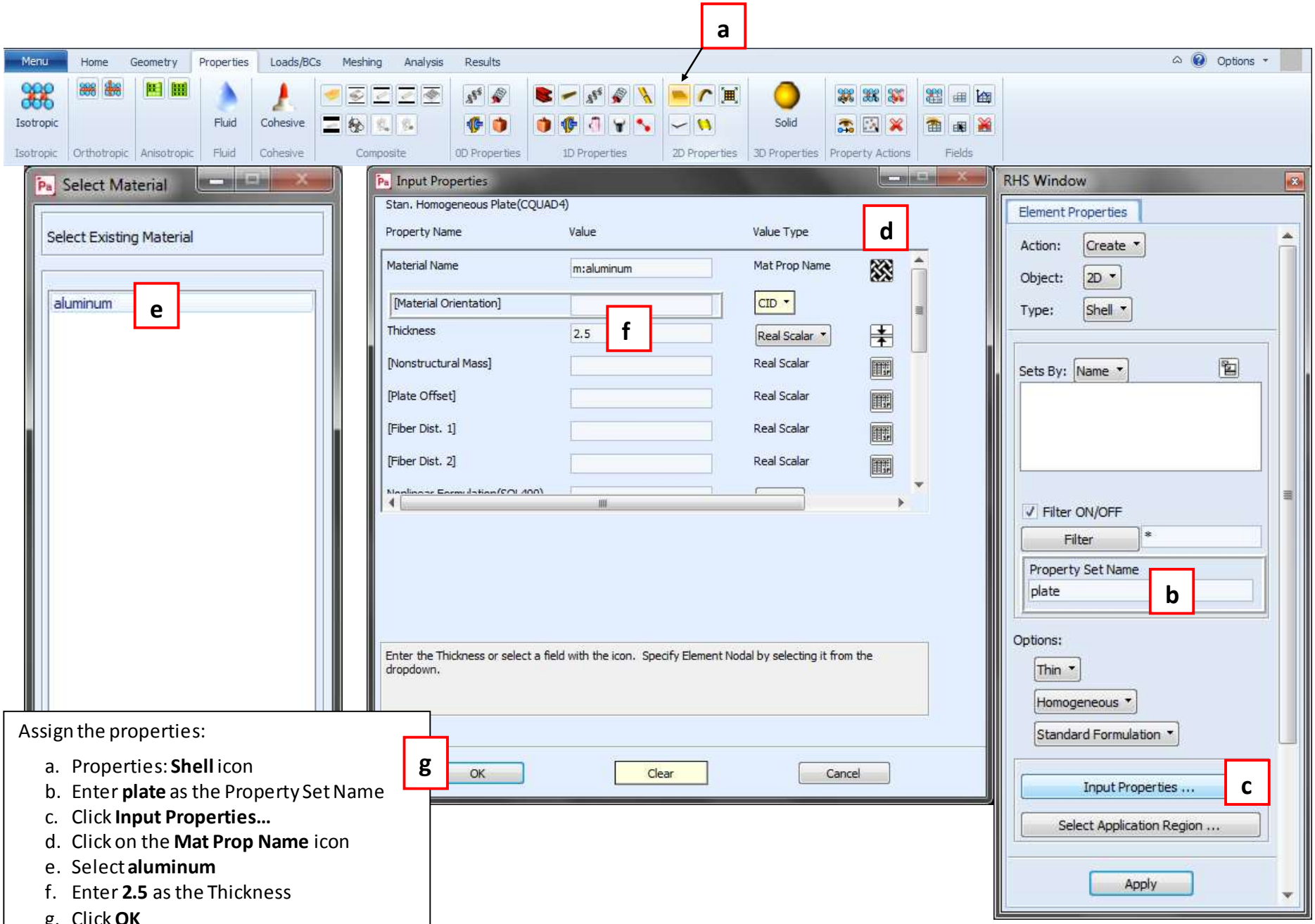
Apply the load:

a. Click on the **Loads/BCs icon/Distributed Load** icon *(Element Uniform tab)*
b. Enter **load** as the New Set Name
c. Target Element Type: **2D**
d. Click **Input Data…**
e. Enter **<0,25,0>** for the Edge Distr Load
f. Click **OK**
g. Click **Select Application Region…**
h. Select **Geometry**
i. Click on the **Select Surface Edges** panel
j. Select the top edge
k. Click **Add**
l. Click **OK**
m. Click **Apply**



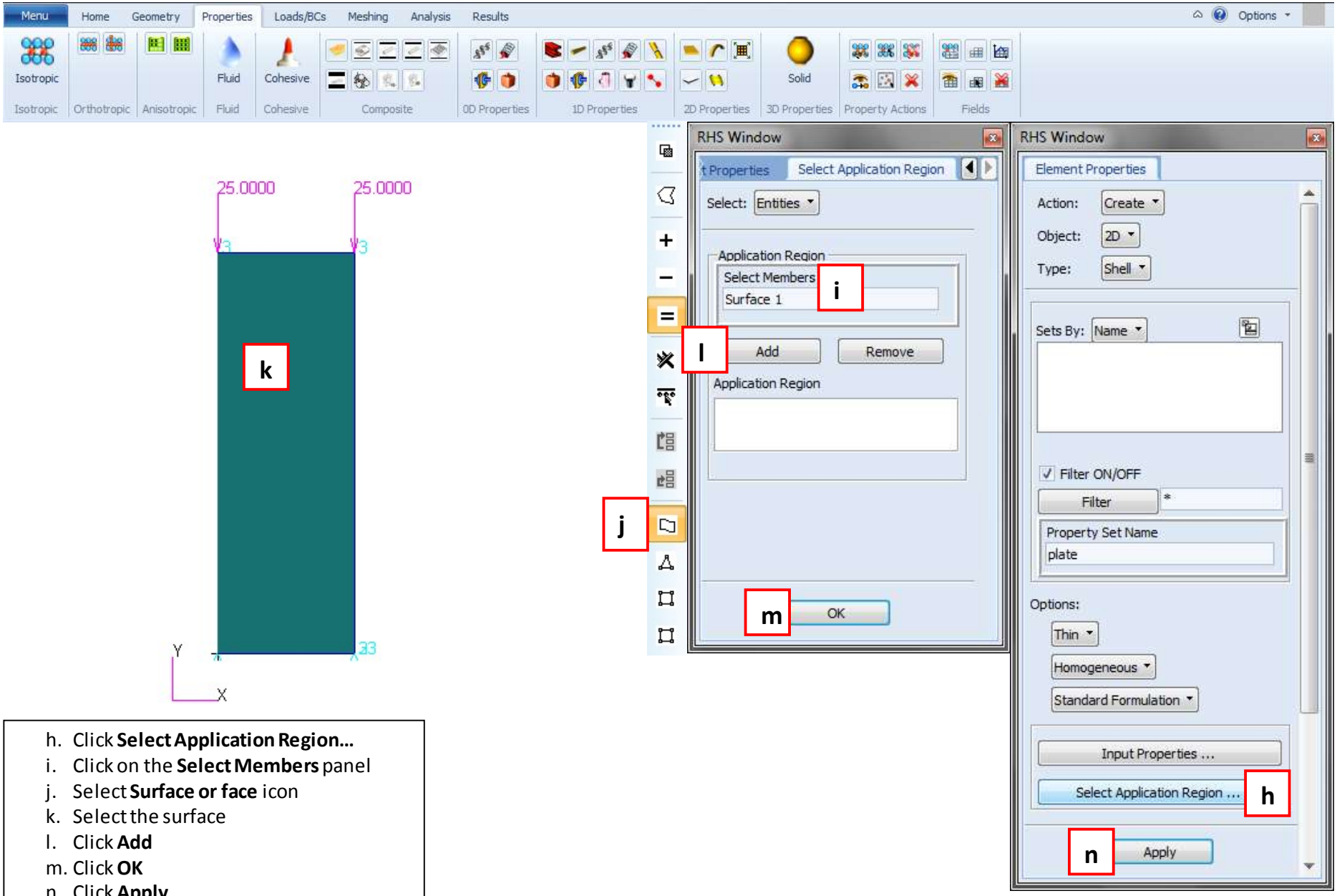
Define a material:

a. Click on the 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**



Assign the properties:

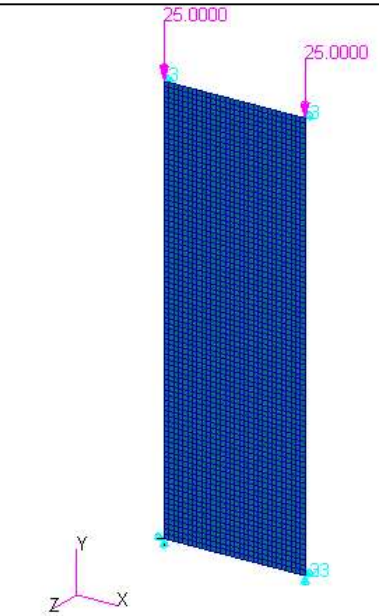
a. Properties: **Shell** icon
b. Enter **plate** as the Property Set Name
c. Click **Input Properties...**
d. Click on the **Mat Prop Name** icon
e. Select **aluminum**
f. Enter **2.5** as the Thickness
g. Click **OK**

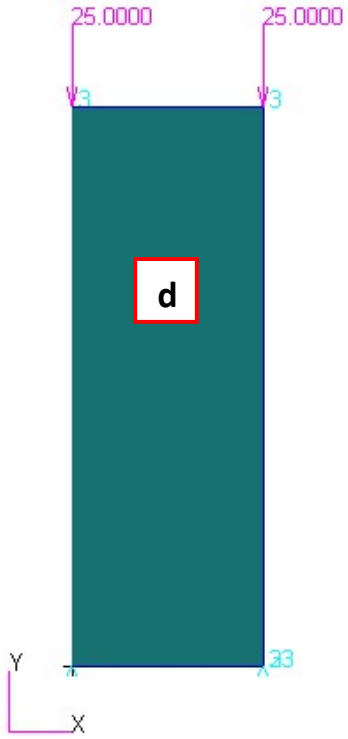


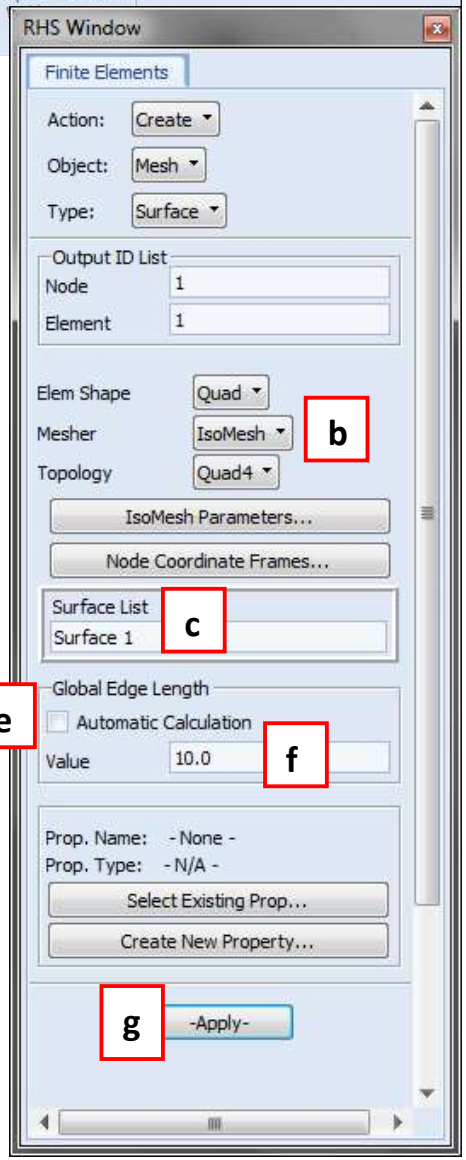
h. Click **Select Application Region…**
i. Click on the **Select Members** panel
j. Select **Surface or face** icon
k. Select the surface
l. Click **Add**
m. Click **OK**
n. Click **Apply**

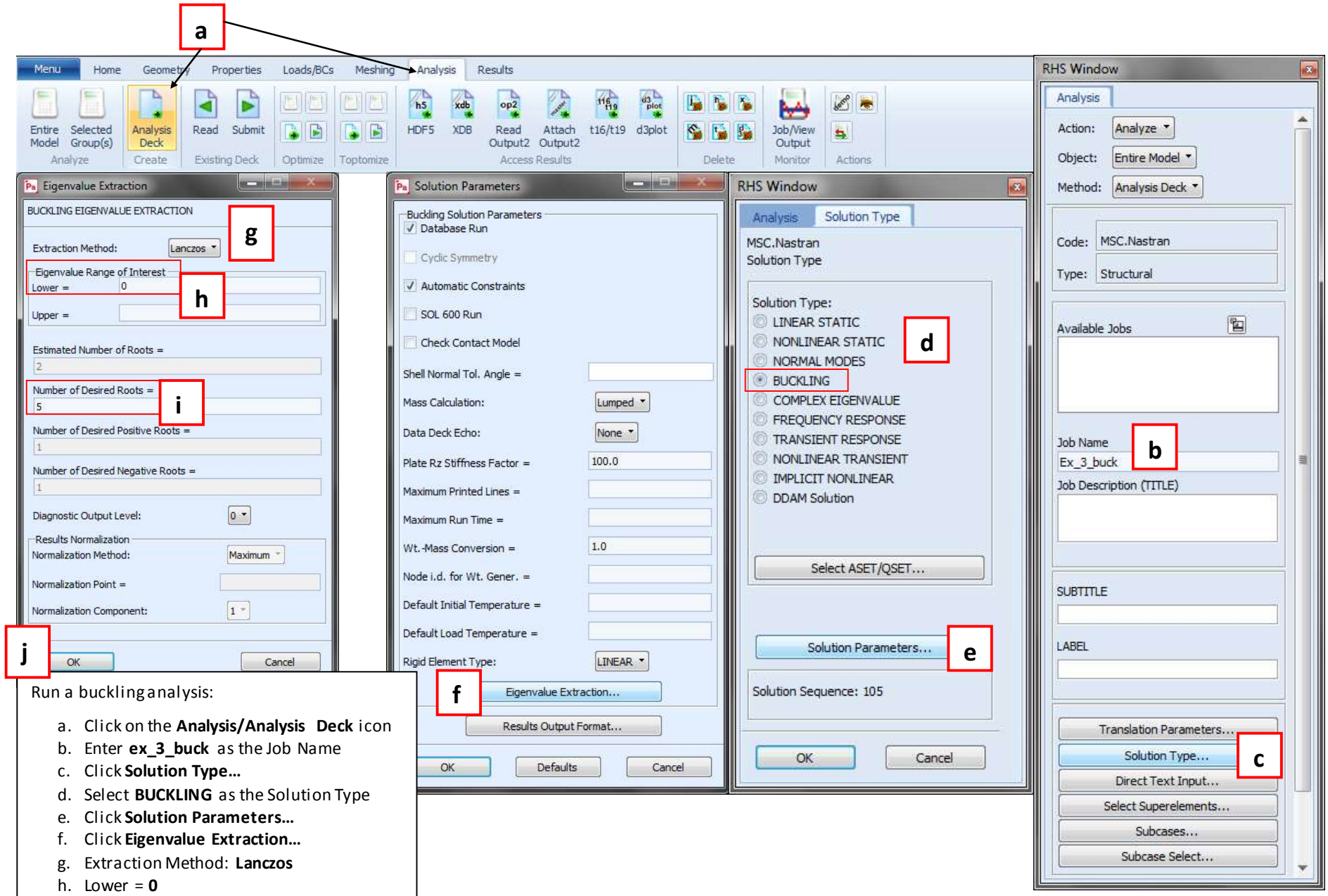
Mesh the surface:

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 surface
e. Uncheck **Automatic Calculation**
f. Enter **10** as the Value of the Global Edge Length
g. Click **Apply**
h. Click **Iso 1 View**



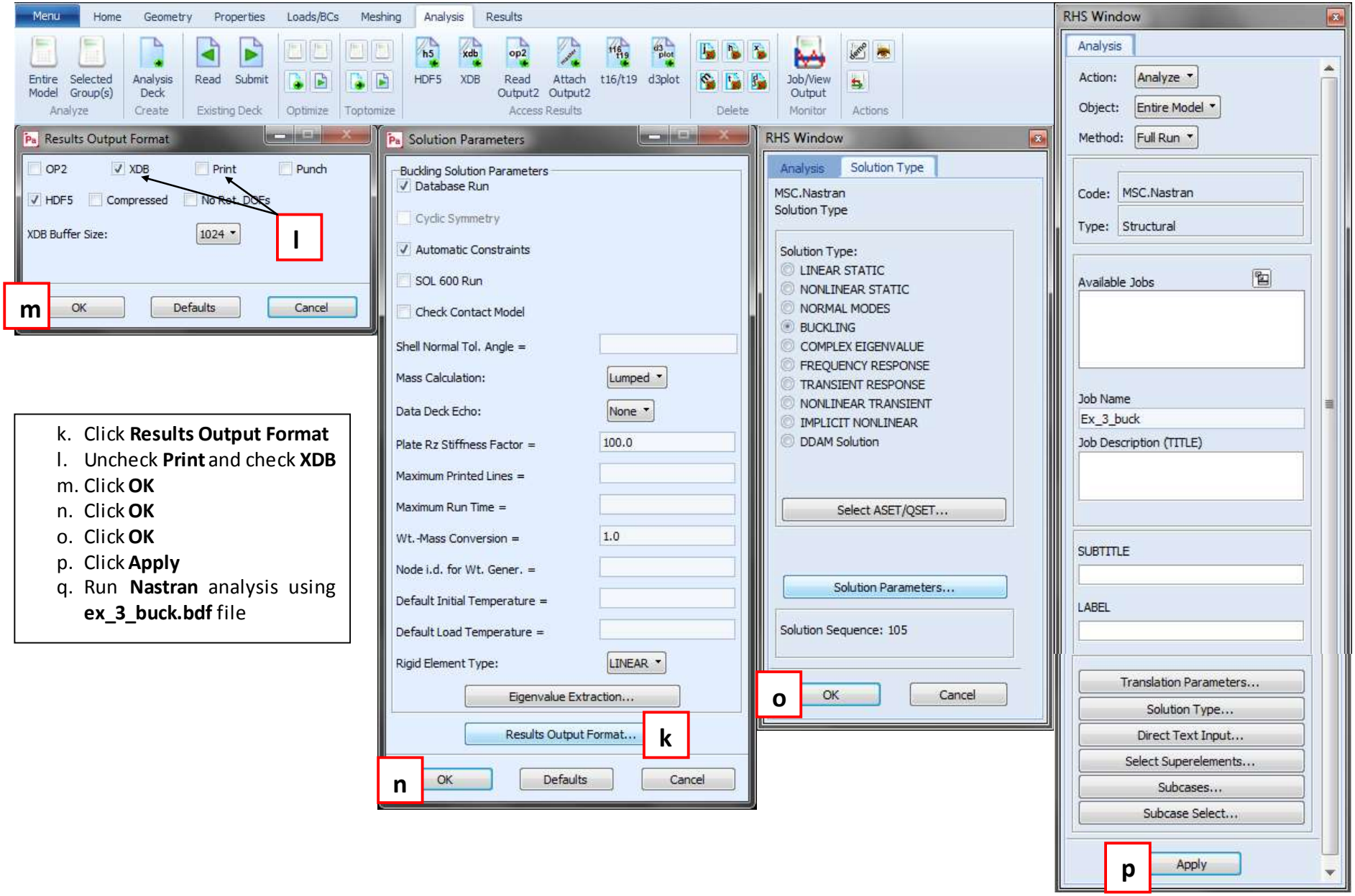




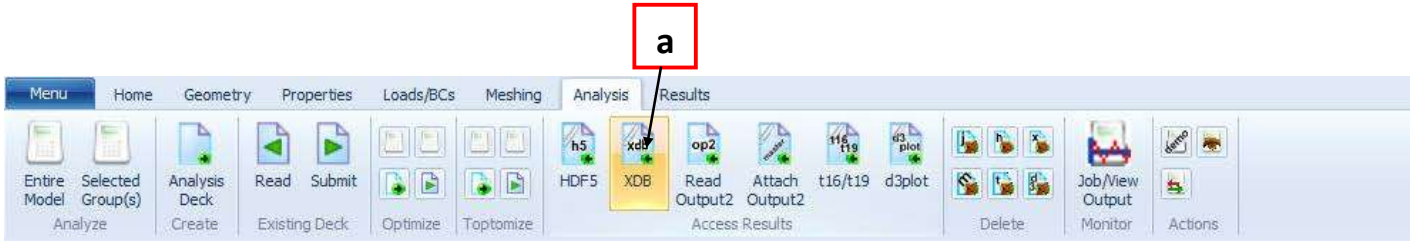


Run a buckling analysis:

a. Click on the **Analysis/Analysis Deck** icon
b. Enter **ex_3_buck** as the Job Name
c. Click **Solution Type...**
d. Select **BUCKLING** as the Solution Type
e. Click **Solution Parameters...**
f. Click **Eigenvalue Extraction...**
g. Extraction Method: **Lanczos**
h. Lower = **0**
i. Enter **5** as the Number of Desired Roots
j. Click **OK**

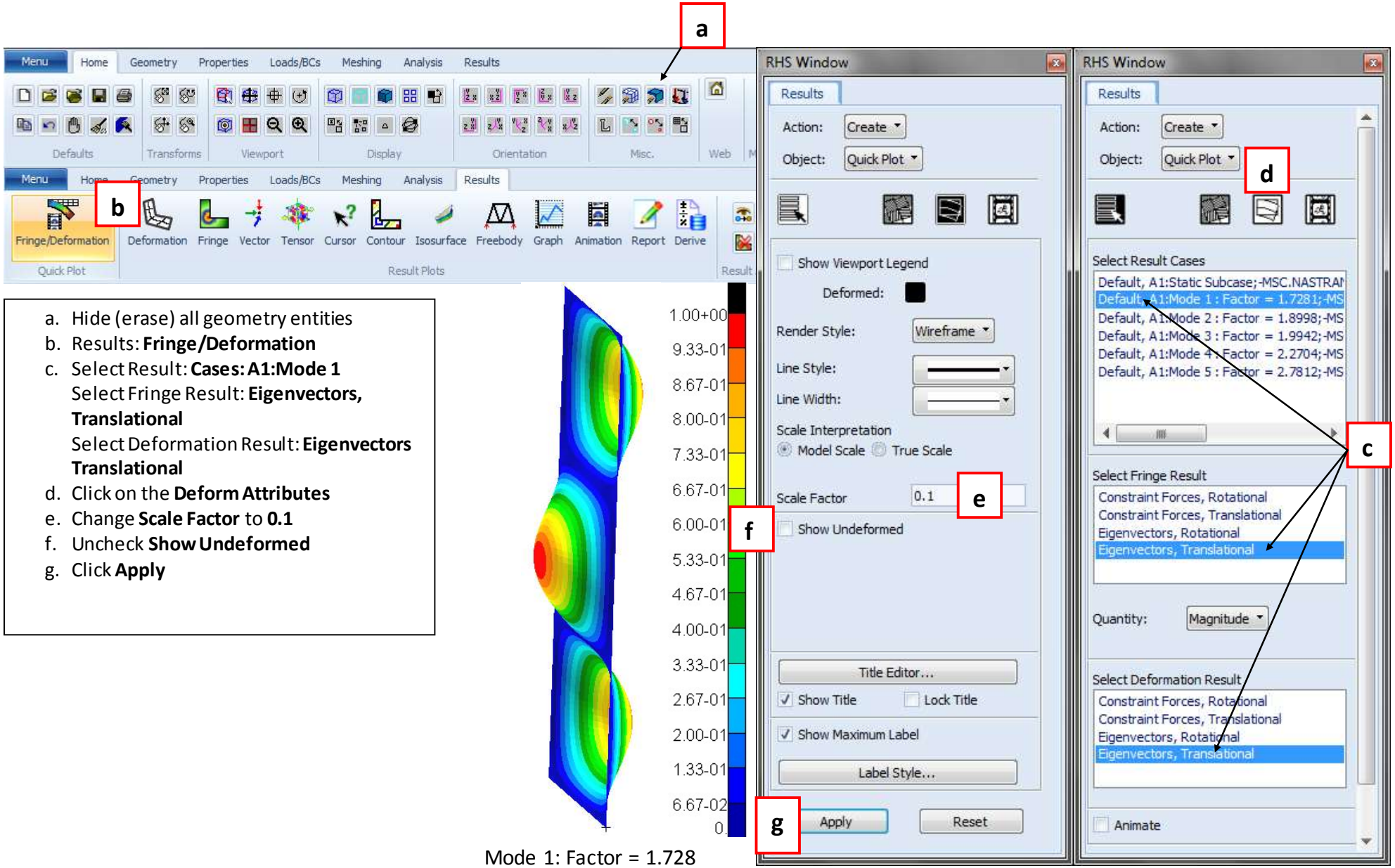


k. Click **Results Output Format**
l. Uncheck **Print** and check **XDB**
m. Click **OK**
n. Click **OK**
o. Click **OK**
p. Click **Apply**
q. Run **Nastran** analysis using **ex_3_buck.bdf** file



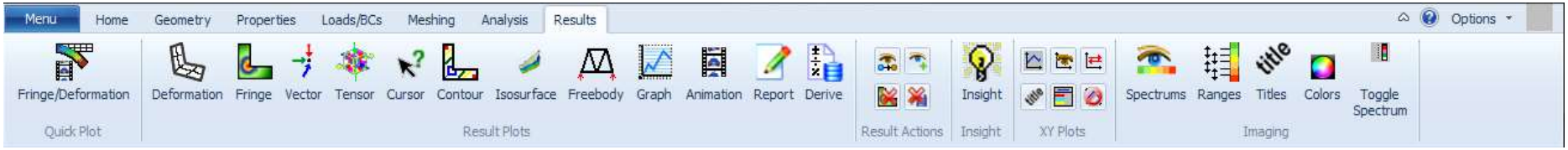
Attach the results file, when the analysis job is completed:

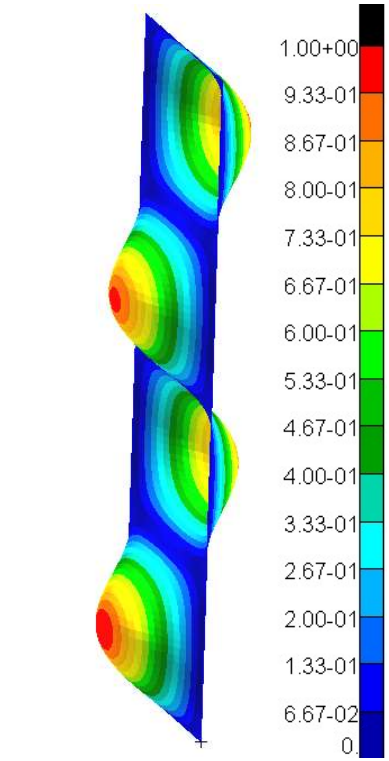
a. Analysis: **XDB** icon
b. Click **Select Results File…**
c. Select **ex_3_buck.xdb** file and click **OK**
d. Click **Apply**



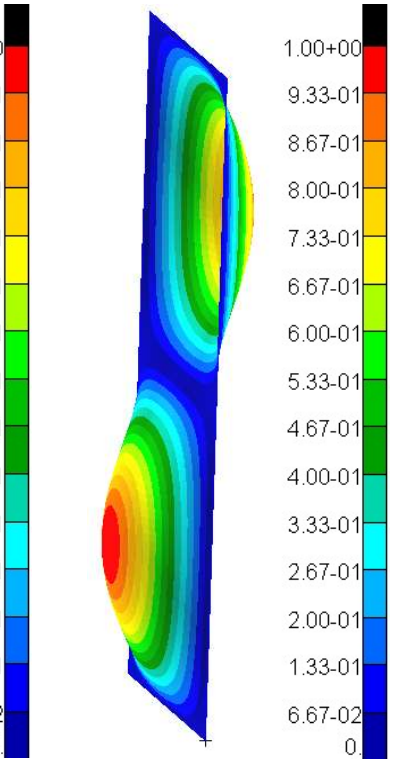
a. Hide (erase) all geometry entities
b. Results: **Fringe/Deformation**
c. Select Result: **Cases: A1:Mode 1**
   Select Fringe Result: **Eigenvectors, Translational**
   Select Deformation Result: **Eigenvectors Translational**
d. Click on the **Deform Attributes**
e. Change **Scale Factor** to **0.1**
f. Uncheck **Show Undeformed**
g. Click **Apply**

Mode 1: Factor = 1.728

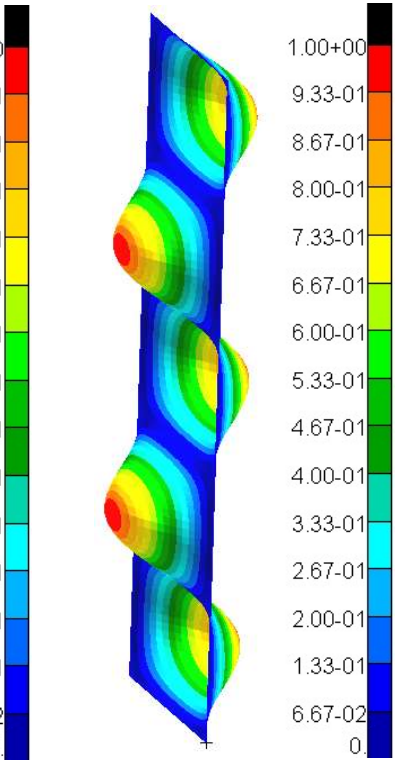




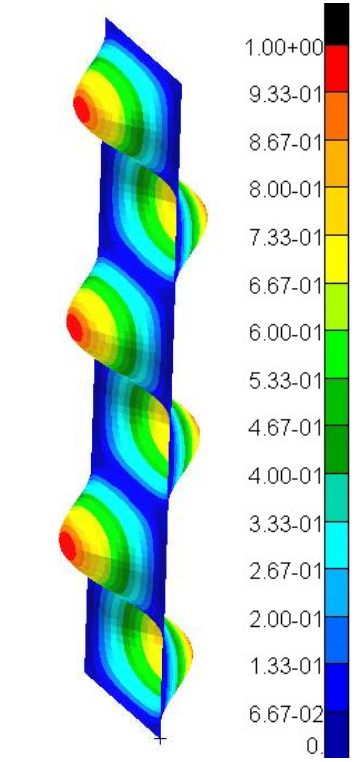
Mode 2: Factor = 1.899



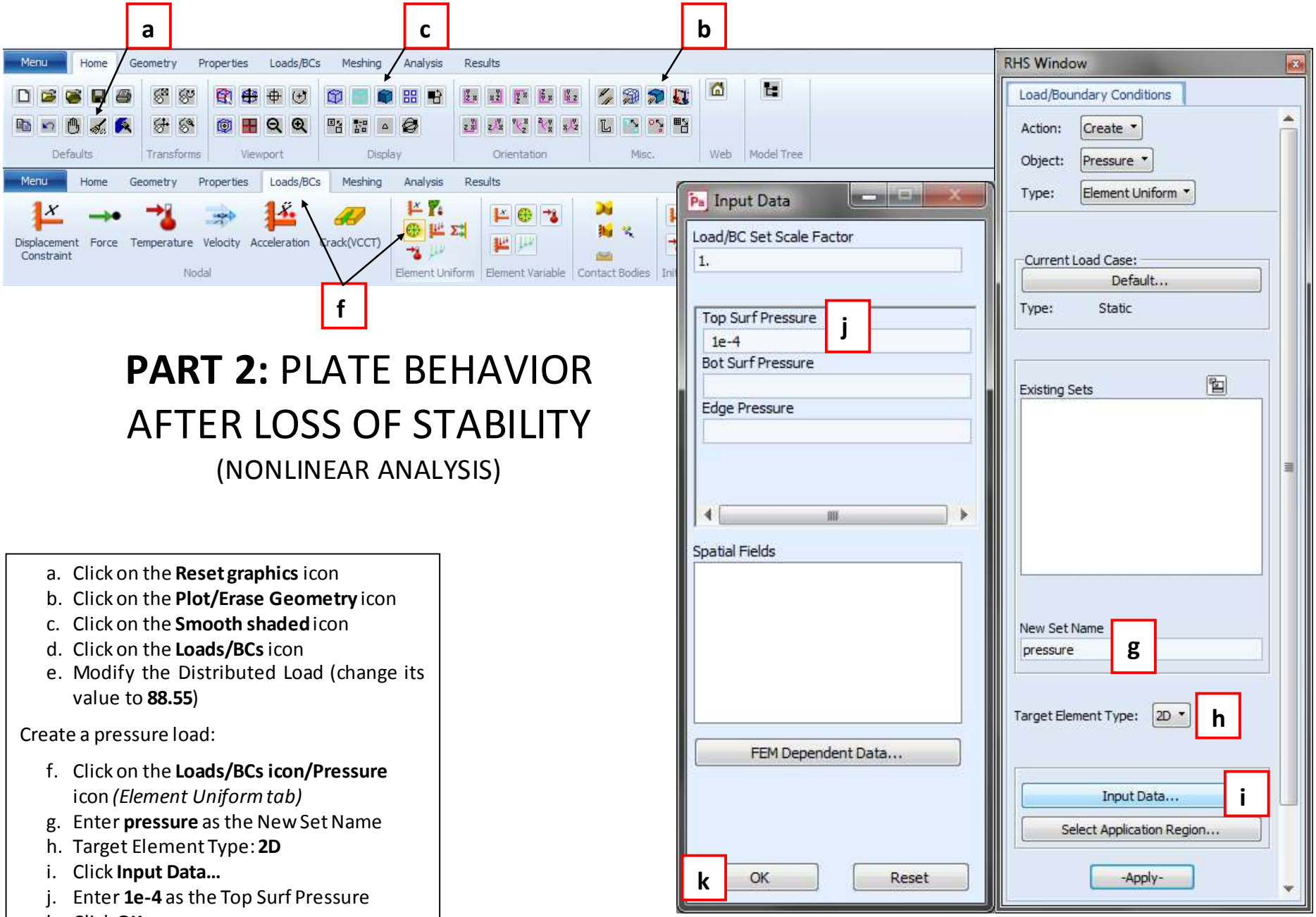
Mode 3: Factor = 1.994



Mode 4: Factor = 2.27



Mode 5: Factor = 2.78



PART 2: PLATE BEHAVIOR AFTER LOSS OF STABILITY

(NONLINEAR ANALYSIS)

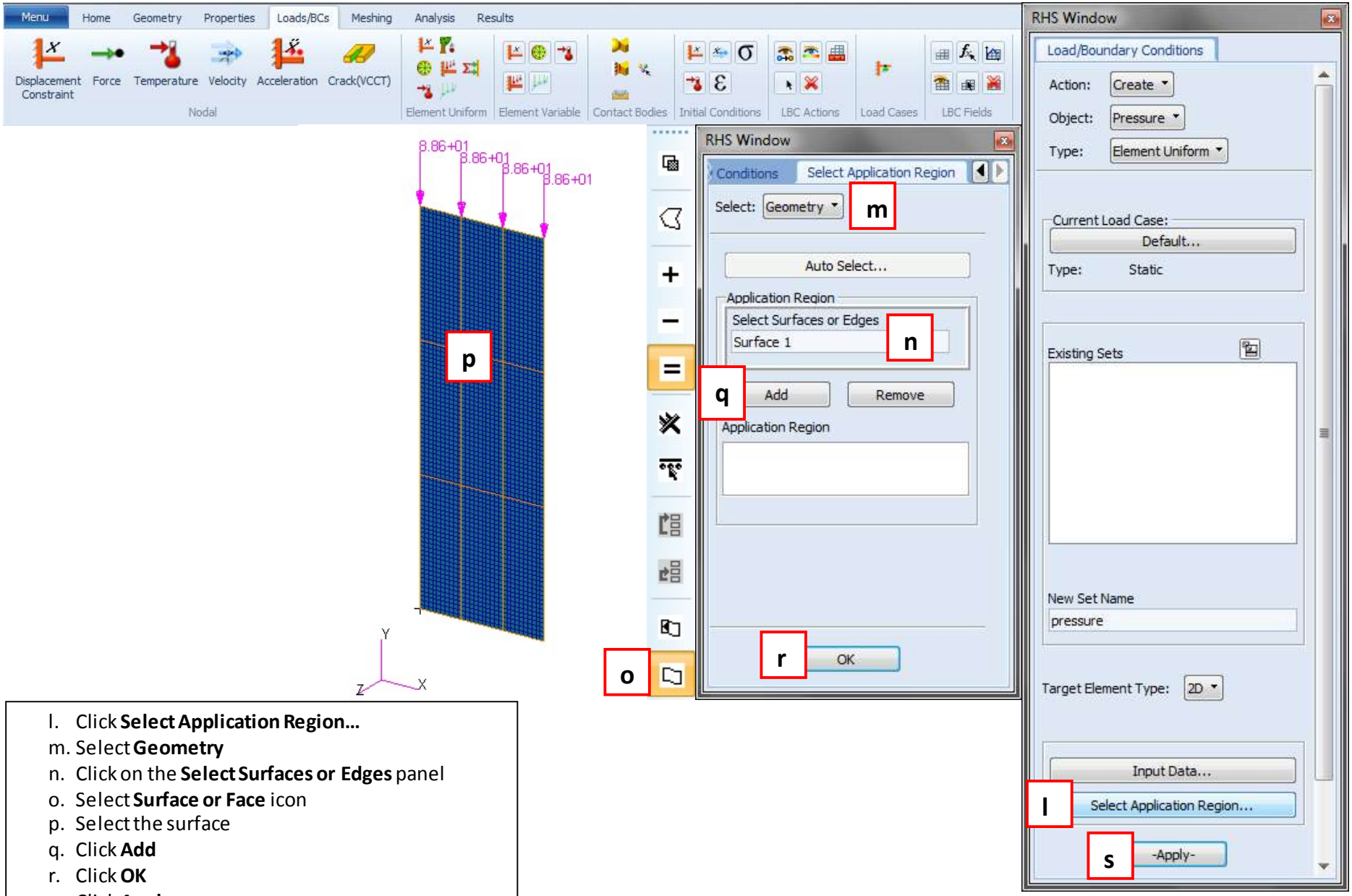
a. Click on the **Reset graphics** icon
b. Click on the **Plot/Erase Geometry** icon
c. Click on the **Smooth shaded** icon
d. Click on the **Loads/BCs** icon
e. Modify the Distributed Load (change its value to **88.55**)

Create a pressure load:

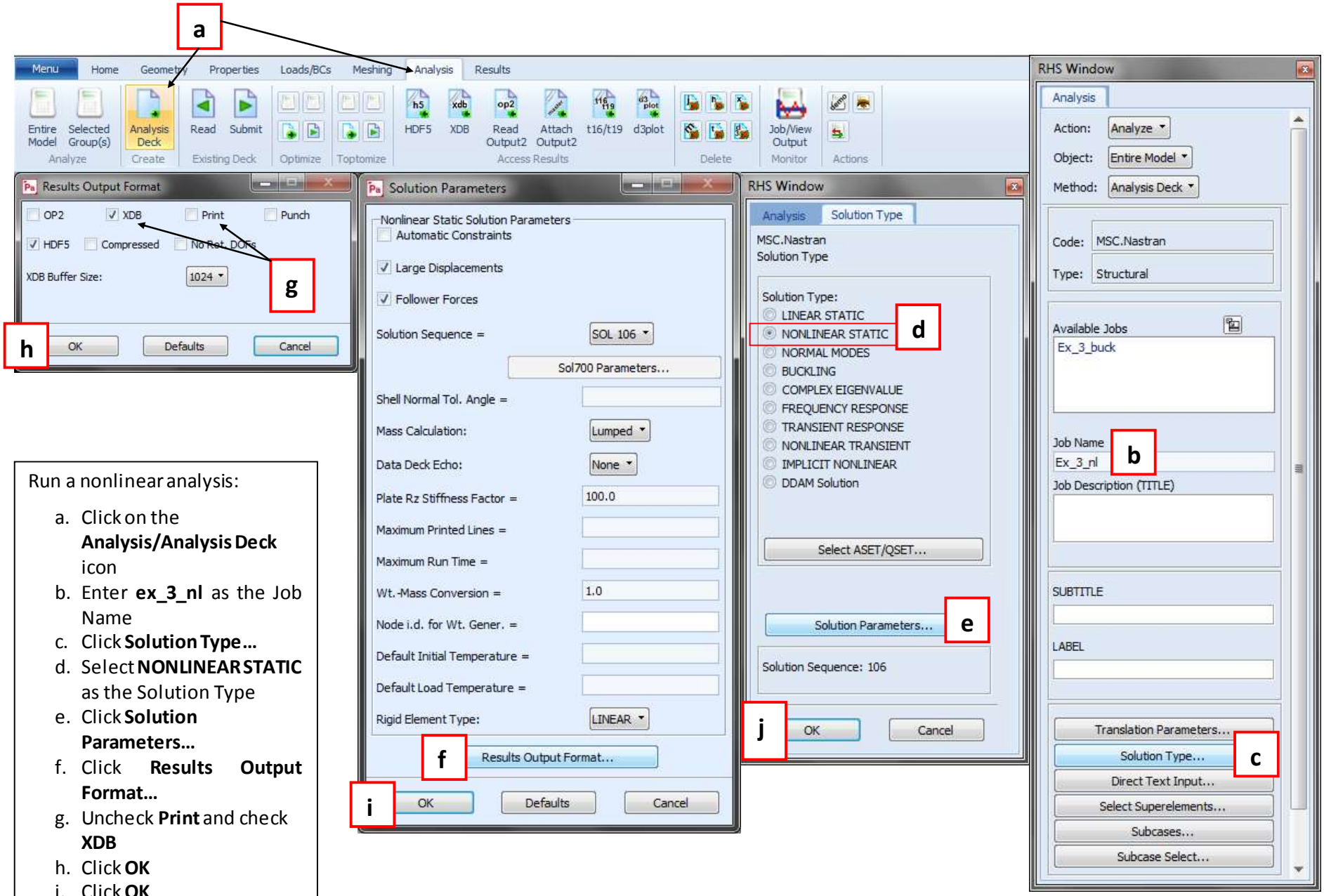
f. Click on the **Loads/BCs icon/Pressure** icon *(Element Uniform tab)*
g. Enter **pressure** as the New Set Name
h. Target Element Type: **2D**
i. Click **Input Data…**
j. Enter **1e-4** as the Top Surf Pressure
k. Click **OK**

HUMAN CAPITAL
NATIONAL COHESION STRATEGY

EUROPEAN UNION
EUROPEAN
SOCIAL FUND



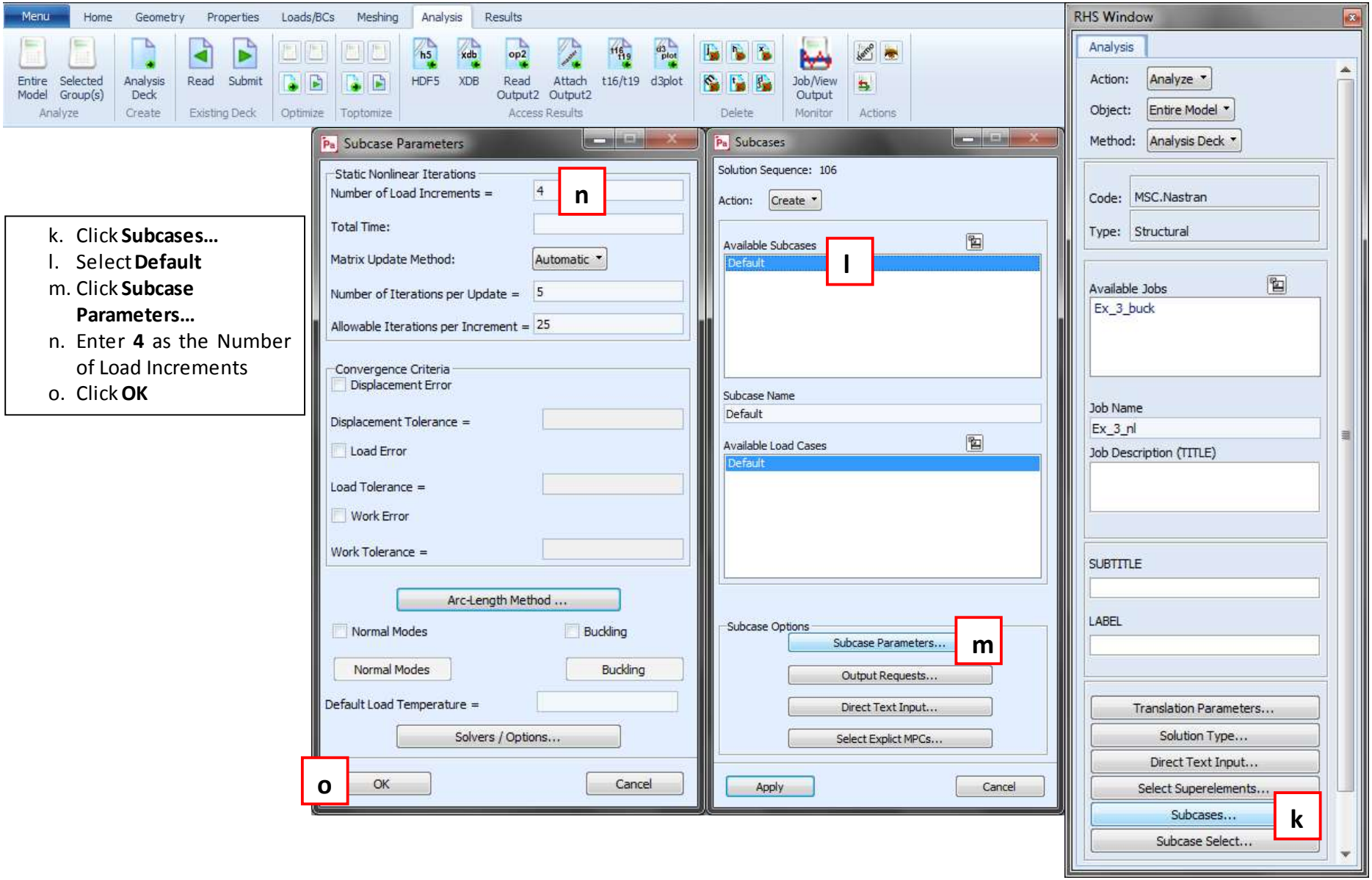
l. Click **Select Application Region...**
m. Select **Geometry**
n. Click on the **Select Surfaces or Edges** panel
o. Select **Surface or Face** icon
p. Select the surface
q. Click **Add**
r. Click **OK**
s. Click **Apply**

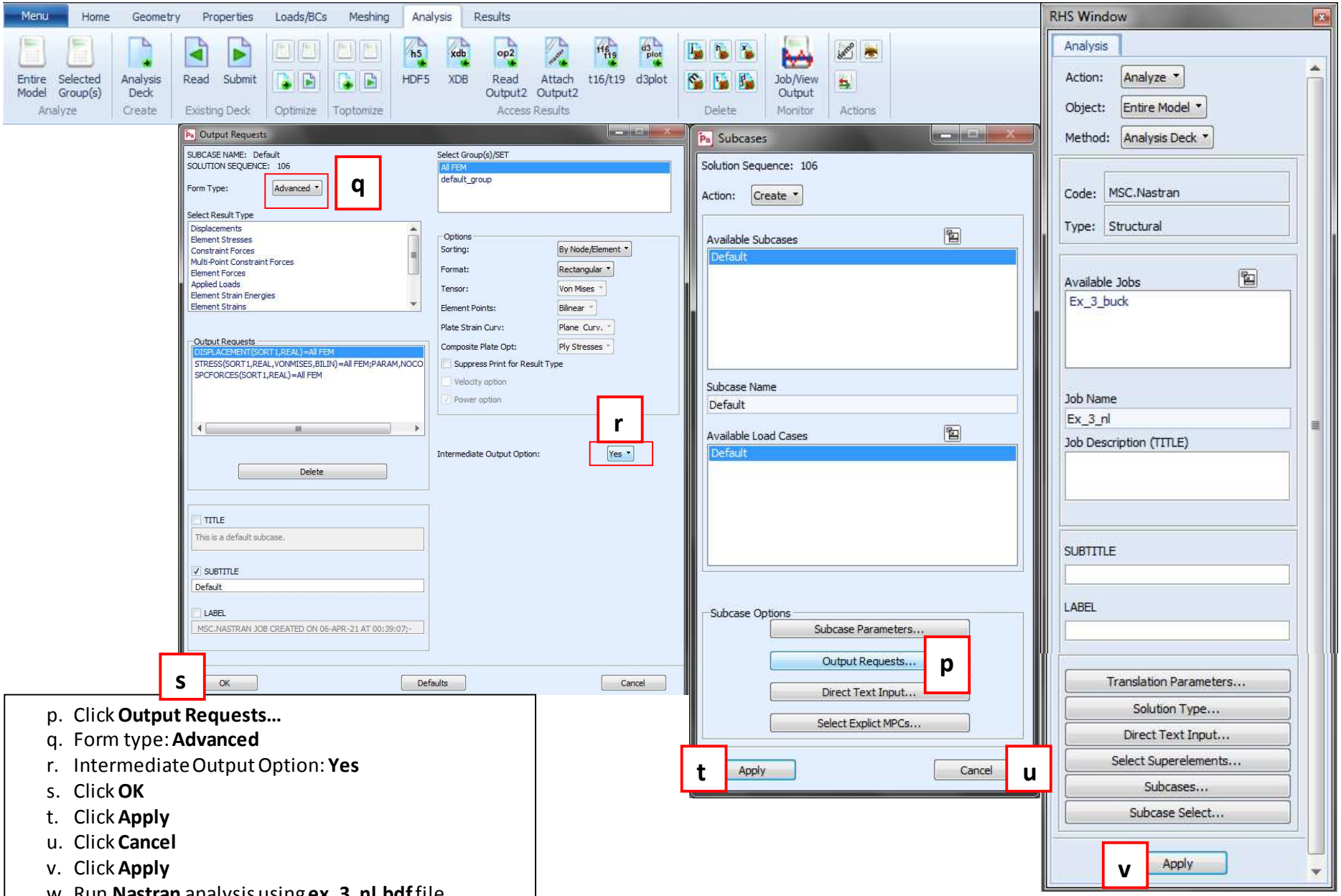


Run a nonlinear analysis:

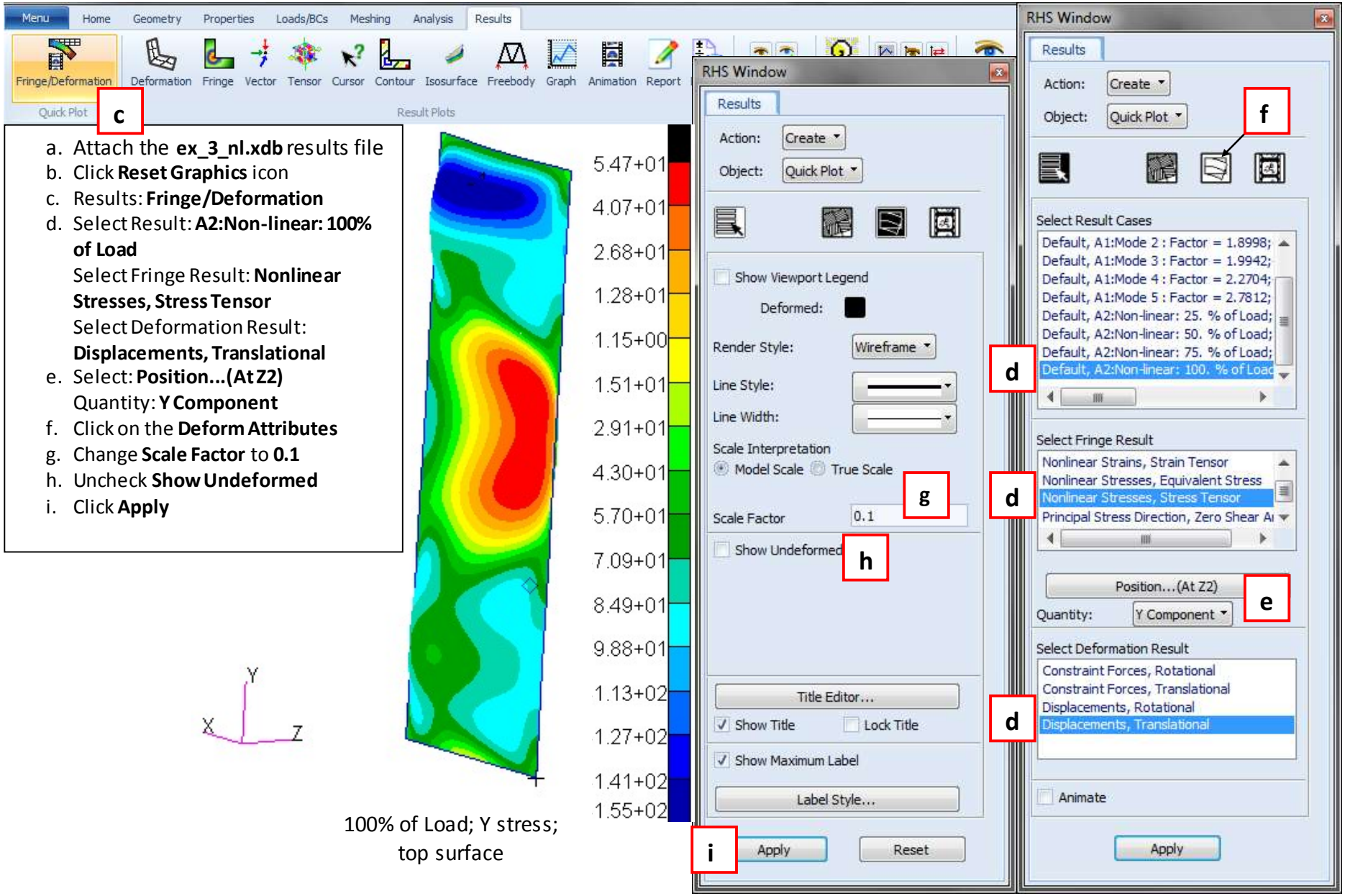
a. Click on the **Analysis/Analysis Deck** icon
b. Enter **ex_3_nl** as the Job Name
c. Click **Solution Type…**
d. Select **NONLINEAR STATIC** as the Solution Type
e. Click **Solution Parameters…**
f. Click **Results Output Format…**
g. Uncheck **Print** and check **XDB**
h. Click **OK**
i. Click **OK**
j. Click **OK**

k. Click **Subcases…**
l. Select **Default**
m. Click **Subcase Parameters…**
n. Enter **4** as the Number of Load Increments
o. Click **OK**

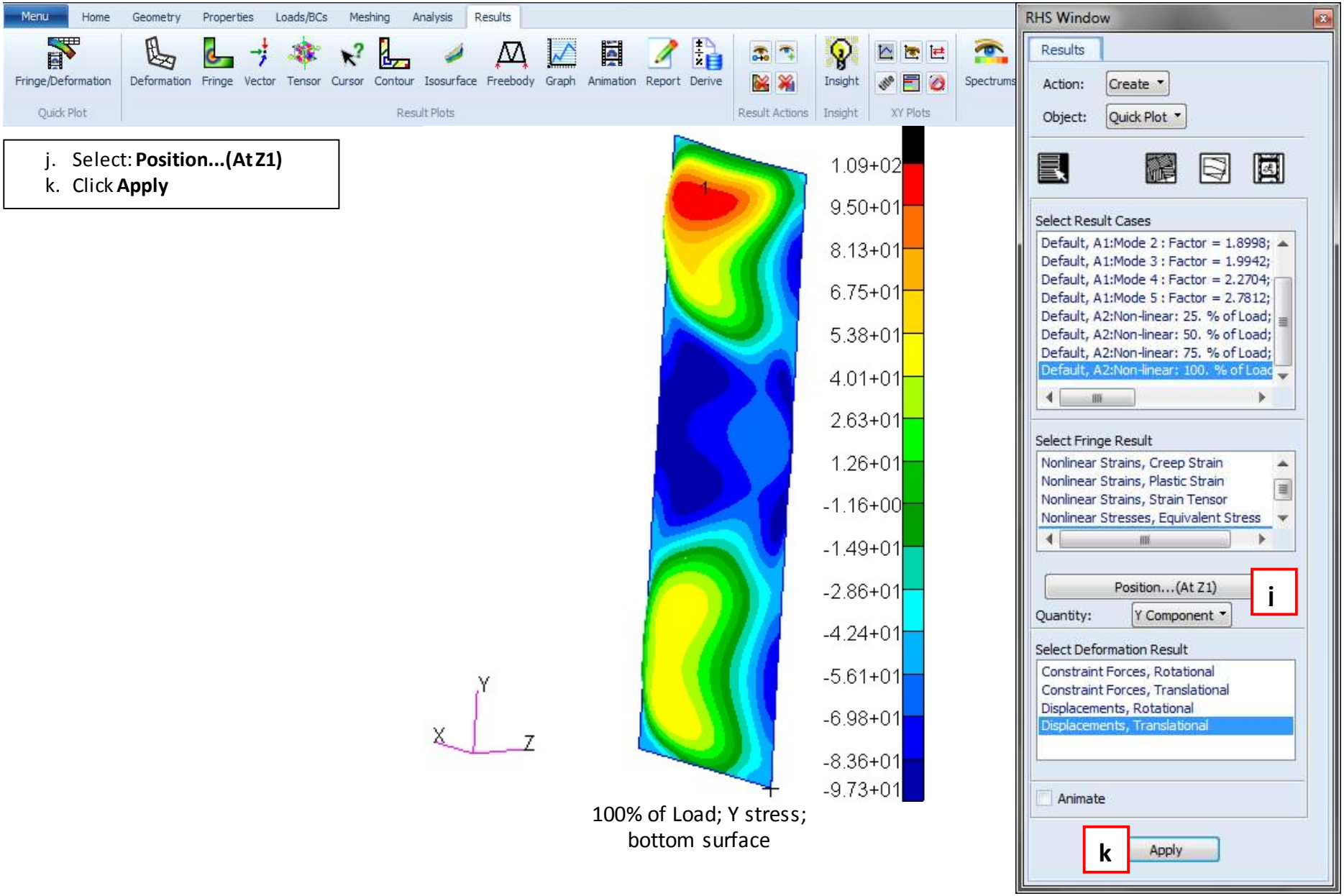




p. Click **Output Requests…**
q. Form type: **Advanced**
r. Intermediate Output Option: **Yes**
s. Click **OK**
t. Click **Apply**
u. Click **Cancel**
v. Click **Apply**
w. Run **Nastran** analysis using **ex_3_nl.bdf** file

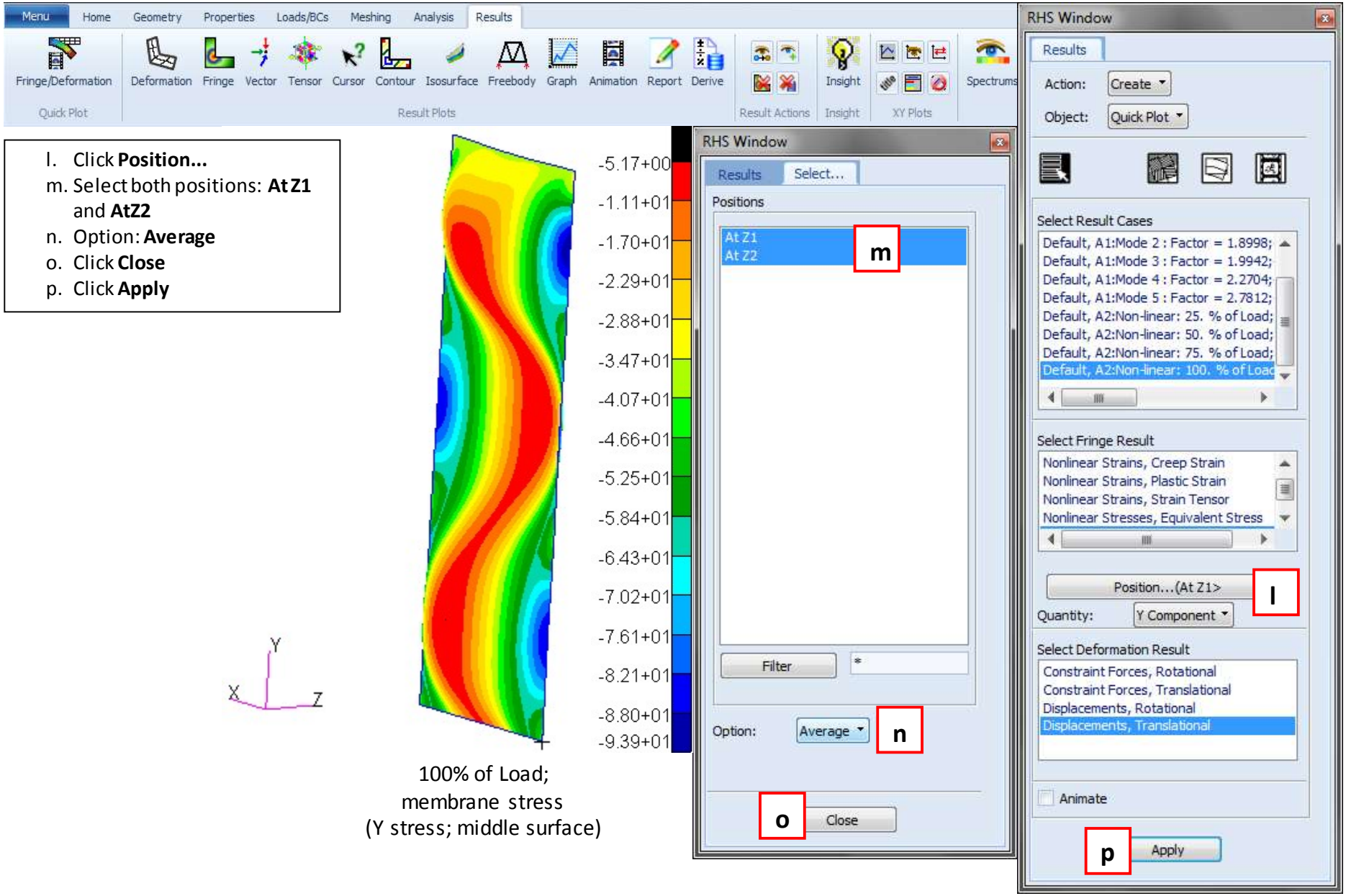


100% of Load; Y stress; top surface



j. Select: **Position...(At Z1)**
k. Click **Apply**

100% of Load; Y stress; bottom surface



l. Click **Position...**
m. Select both positions: **At Z1** and **AtZ2**
n. Option: **Average**
o. Click **Close**
p. Click **Apply**

100% of Load;
membrane stress
(Y stress; middle surface)