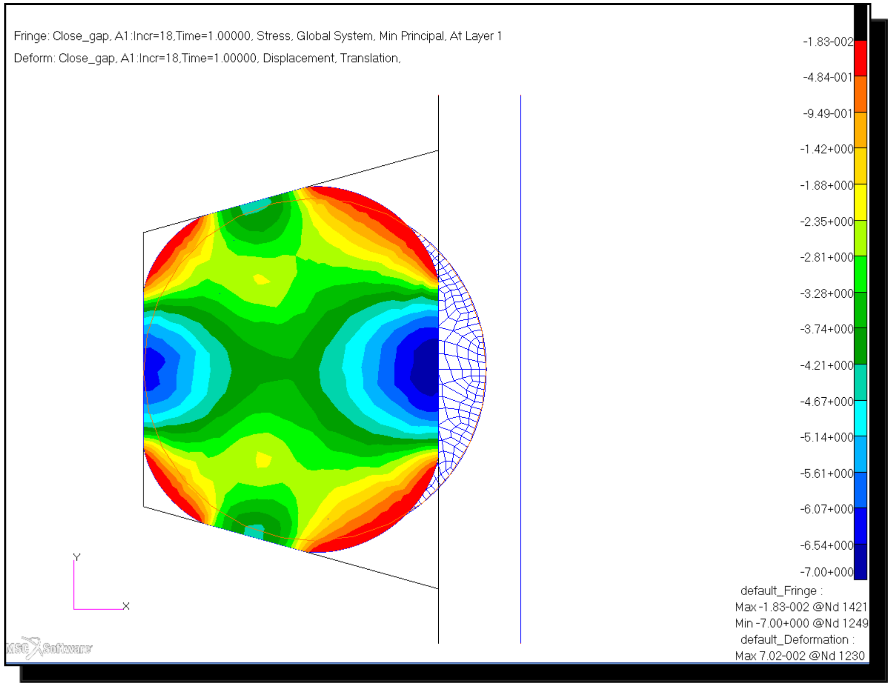
WORKSHOP
CONTACT PRESSURE BETWEEN O-RING AND HATCH

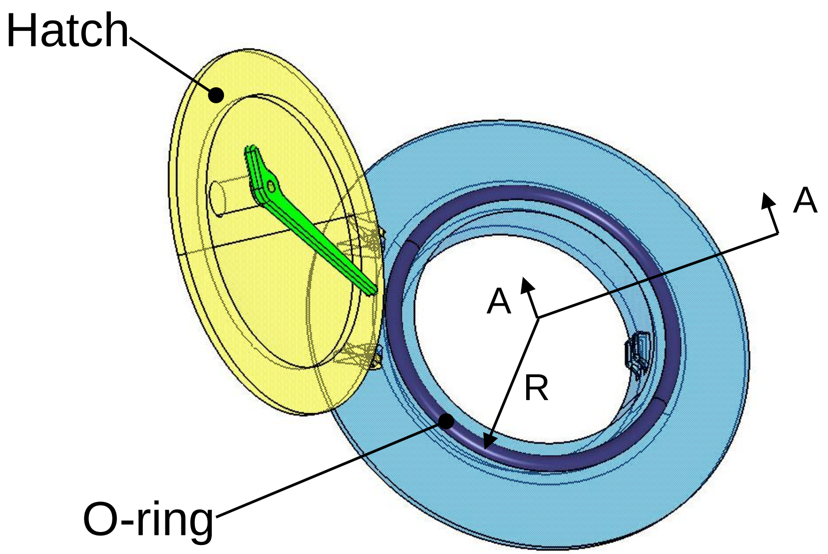


- **Objective**
  - Determine the contact pressure between the o-ring and the hatch at the closed position.
  - In order to determine the survivability of the ring, find minimum principal (maximum compressive) stresses.
  - To accomplish these objectives, create rigid and deformable bodies.

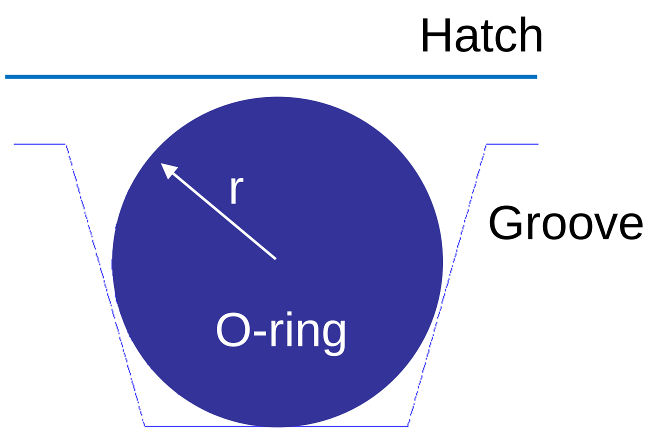
- **Software Version**
  - Patran 2012
  - Marc 2012

- **Required Files**
  - oring_model.db

- **Problem Description**
  - In this example, an o-ring with a 10 inch radius sits inside a groove on a submarine's entrance. The hatch will compress the ring sealing off the entrance. The contact pressure between the hatch and the ring must be sufficiently large enough to prevent leakage. The ring also needs to survive the maximum loading condition. The analysis of this 3D model of the O-ring is simplified to a 2D model as shown below.

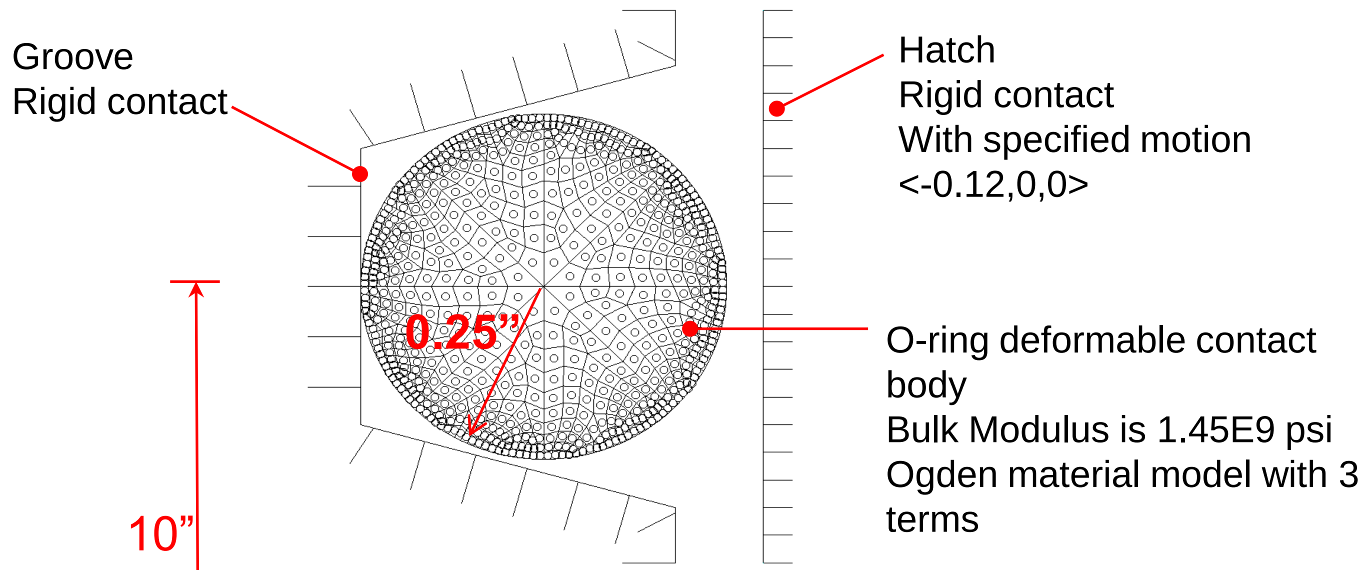


Submarine's entrance



Section A-A

- **Problem Description Continued**
  - The deformable o-ring is compressed by the hatch into the groove. An Ogden rubber model is used to model the ring. As shown below, the cross sectional radius of the ring is .25 inches. The hatch and the groove are both assumed rigid. The hatch will close with a velocity of .12 inch/sec.



Term	Modulus (psi)	Exponent
1	9.12	1.3
2	0.17	5.0
3	-0.14	-2.0

- **Suggested Exercise Steps**
  1. Create new database named *oring.db*.
  2. Import the model *oring_model.db*.
  3. Transform the model to the correct coordinates for axisymmetric analysis.
     - Transform the geometry – hatch and groove curves
     - Transform the elements – o-ring
  4. Define the o-ring as a deformable contact body.
  5. Define the hatch and groove as rigid contact bodies.
  6. Define the rubber material as isotropic, hyperelastic ogden with properties described on previous page.
  7. Define the o-ring properties as 2D solid, axisymmetric, and rubber.
  8. Create a load step named *Close_gap* and set the solution parameter *Total Time* to equal 1. 0.
  9. Request the following output results:
     - Contact Normal Stress
     - Contact Normal Force
     - Contact Status
     - Displacement

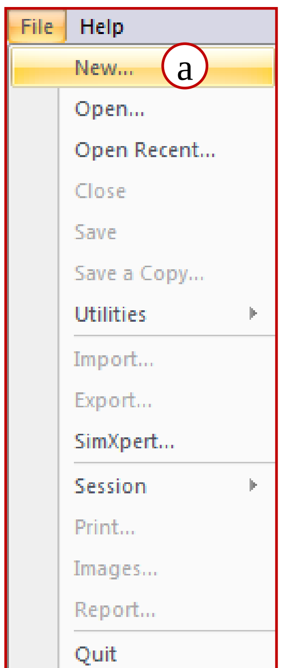
- **Suggested Exercise Steps Continued**

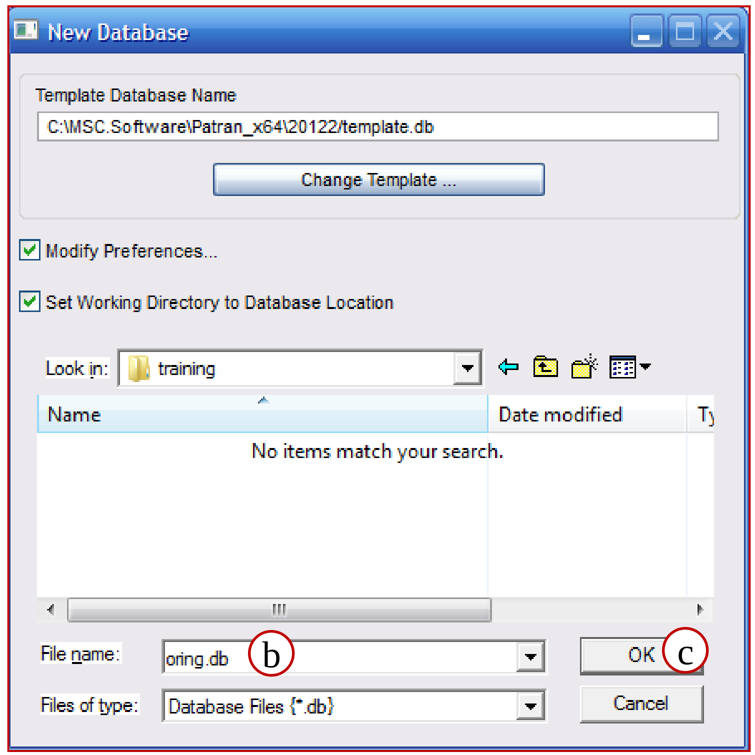
10. Select the load step *Close_gap* and run the analysis using Marc.
11. Use the job monitoring form to monitor the job as it is running.
12. Attach the Marc results file *oring.t16* to Patran.
13. Plot normal contact stresses.
14. Graph contact stresses along the surface of the ring.
    - Create a curve to represent the surface of the ring.
    - Associate nodes to the curve.
    - Create graph using the previously created curve and associated nodes.
15. Plot minimum principal stresses.

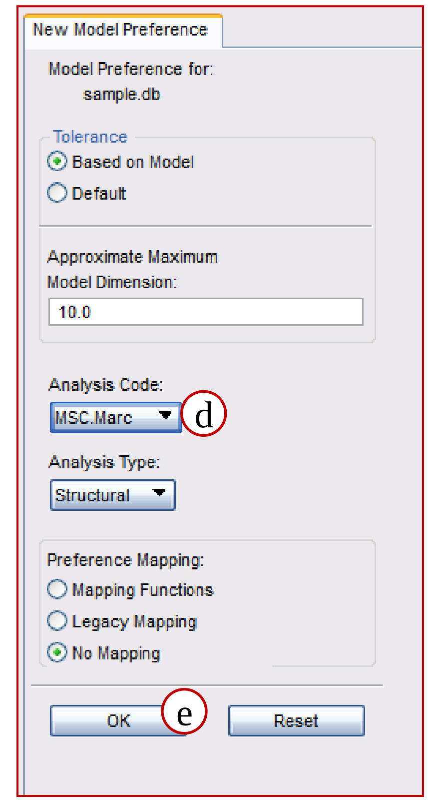
Step 1. Create a New Database

Create a new database named *oring.db*.

a. Click *File* >**New**.
b. In *File Name* text box enter **oring.db**.
c. Click **OK**.
d. Select the *Analysis Code* to be **MSC. Marc**.
e. Click **OK.**



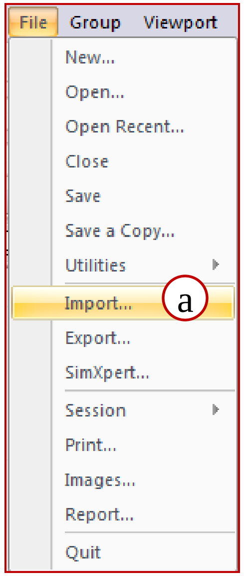


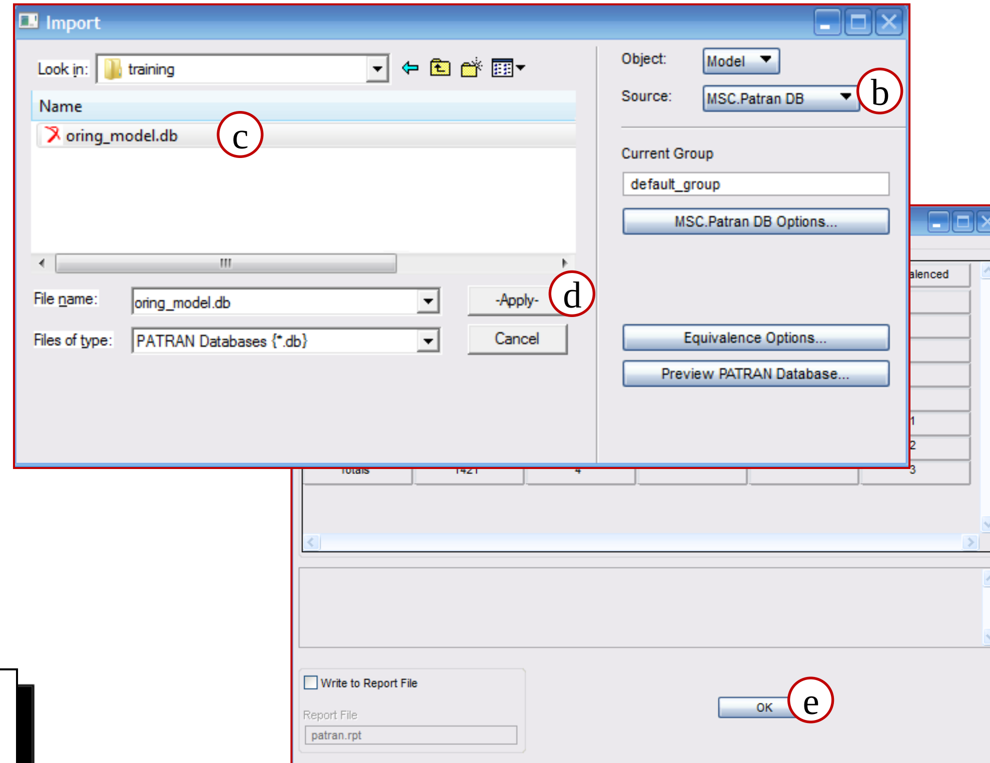


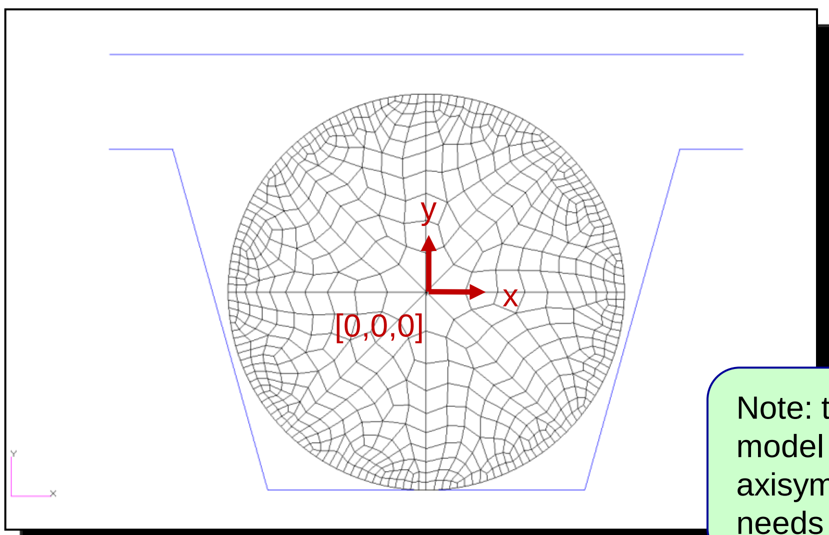
Step 2. Import Model

Import the model *oring_model.db*.

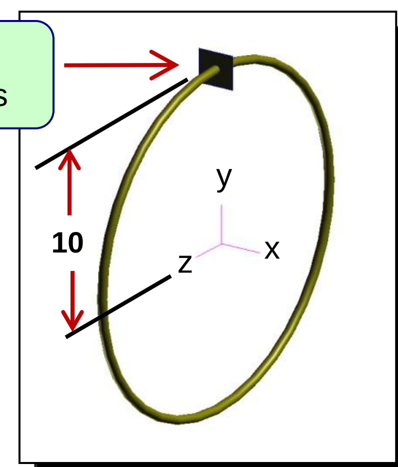
a. Click *File* > **Import**
b. Select the *Source* to be **MSC. Patran DB**.
c. Select the file **oring_model.db**
d. Click **Apply**.
e. Click **OK** on the *Database Import Summary*.





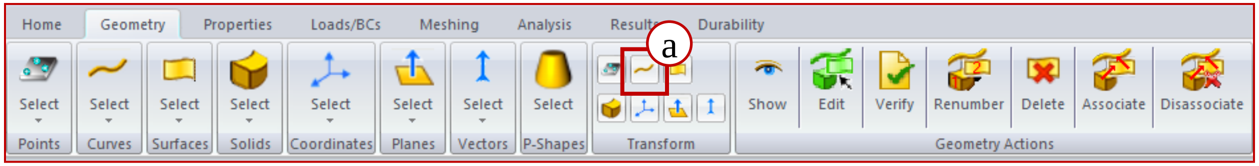


Working plane to be used with axisymmetric analysis



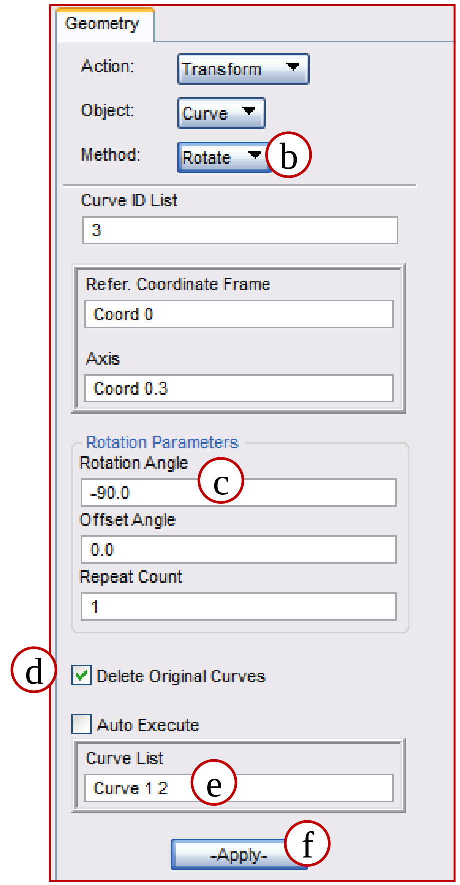
Note: the coordinates of the given model is not correct for axisymmetric analysis. The model needs to be transformed.

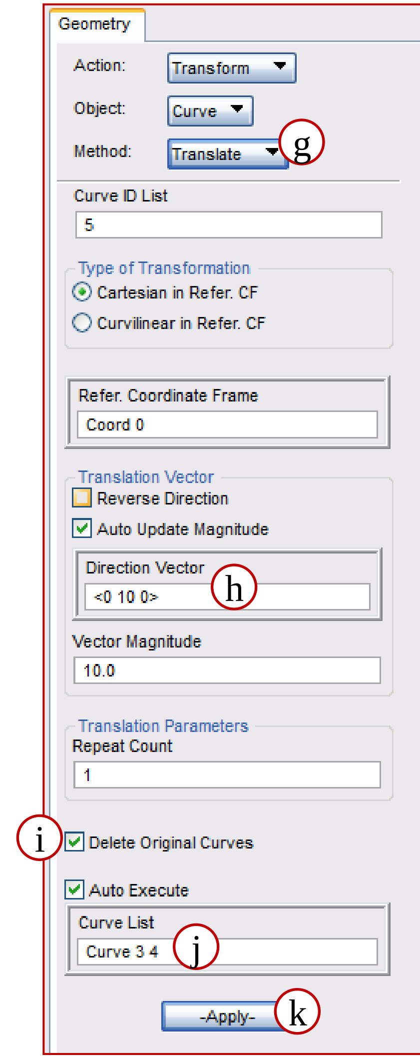
Step 3. Transform Model



Transform geometries to the correct coordinates for axisymmetric analysis.

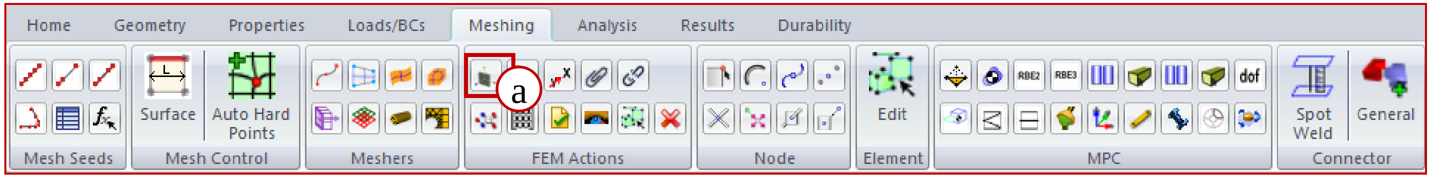
a. Under the *Geometry* tab, click **Curve** in the *Transform* group.
b. Set *Method* to be **Rotate**.
c. For the *Rotation Angle*, enter ***-90***
d. Check the *Delete Original Curves* check box.
e. In *Curve List* text box enter **Curve 1 2** (or select the **Groove** and **Hatch** curves)
f. Click **Apply** (Click **Yes** when asked to delete the original curves)
g. Change the *Method* to **Translate**.
h. For the *Direction Vector*, enter **<0 10 0>**.
i. Check the *Delete Original Curves* check box.
j. In *Curve List* text box, enter **Curve 3 4** (or select the **Groove** and **Hatch** curves)
k. Click **Apply (**Click **Yes** when asked to delete the original curves)





Note: When transforming geometries, the check box for Delete Entities should be checked. Otherwise, the entities will be duplicated. The numbering of the new entities will be different from the original ones.

Step 3. Transform Model (Cont.)

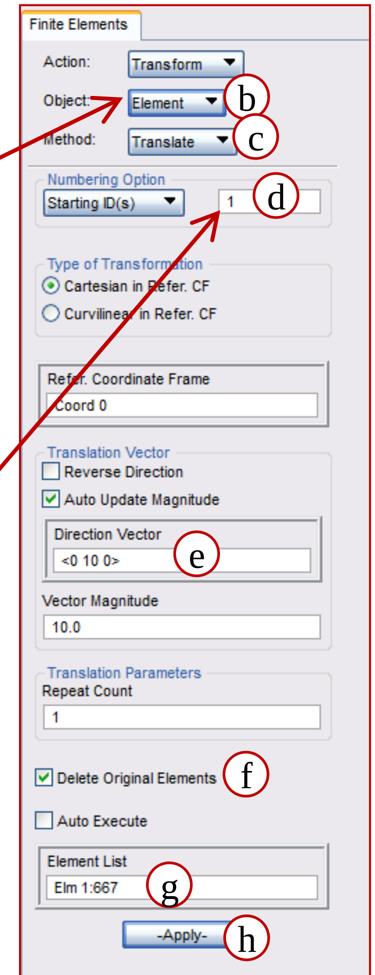


Transform elements to the correct coordinates for axisymmetric analysis

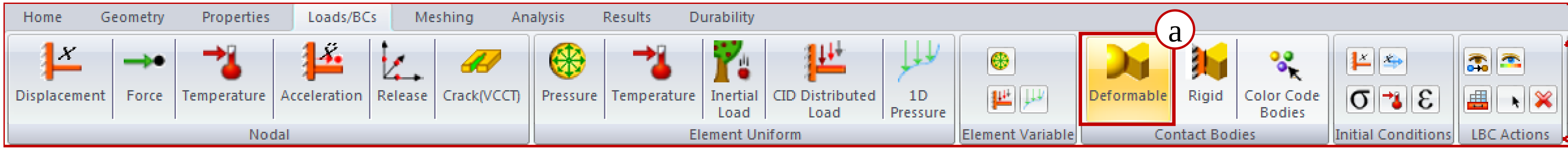
a. Under the *Meshing* tab, click **transform** in the *FEM Actions* group.
b. Set *Object* to **Element**.
c. Set *Method* to **Translate**.
d. For *Numbering Option, Starting ID(s)*, enter **1**.
e. For the *Direction Vector*, enter **<0 10 0>**.
f. Check the *Delete Original Elements* check box.
g. Click in the *Element List* text box and enter **Elm 1:667** (or select all elements from the screen).
h. Click **Apply**

Transforming elements automatically transforms nodes attached to the elements being transformed.

This will set the ELEMENT numbers of the transformed elements to start at 1. The NODE numbers of the transformed nodes, however, start after the last node number of the original node set.

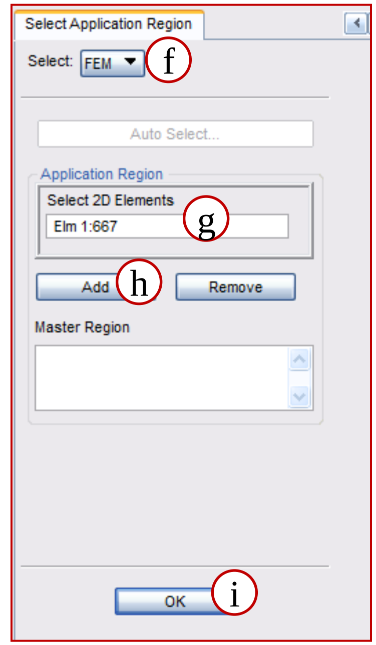


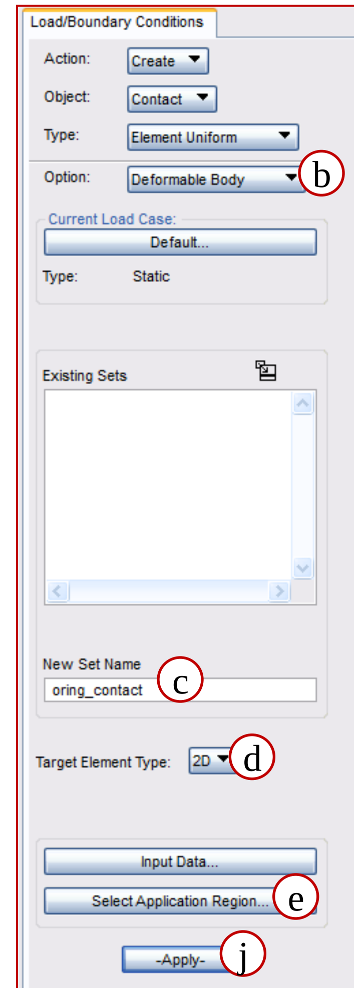
Step 4. Define the Deformable Contact Body

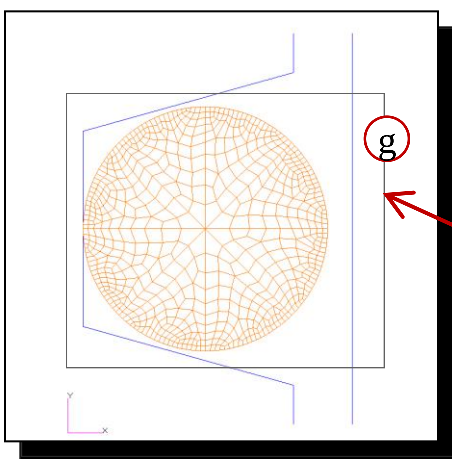


Define the ring as a deformable contact body.

a. Under the *Loads/BCs* tab, click **Deformable** in the *Contact Bodies* group.
b. Make sure the **Option** is set to **Deformable Body**.
c. For the *New Set Name*, enter **oring_contact**.
d. Set *Target Element Type* to be **2D**.
e. Click **Select Application Region**.
f. Set *Select* to be **FEM**.
g. Click in the *Select 2D Elements* text box and select all elements on screen or enter **Elm 1:667**.
h. Click **Add**.
i. Click **OK**.
j. Click **Apply**.

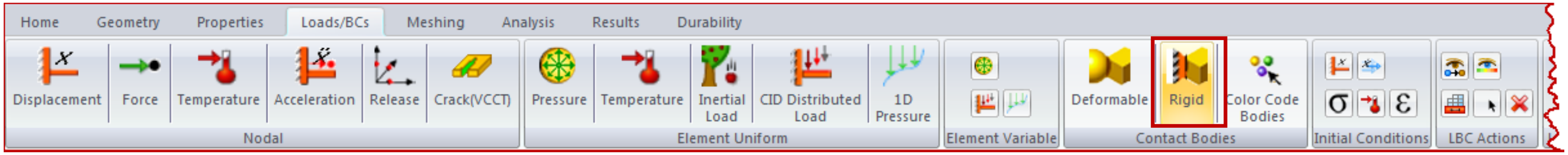






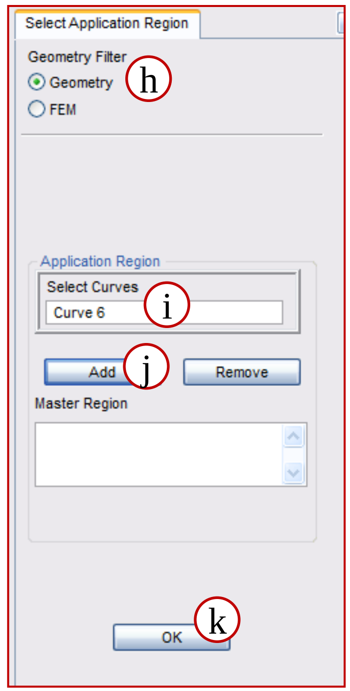
Select all elements on screen by drawing a rectangle to include all elements of the ring.

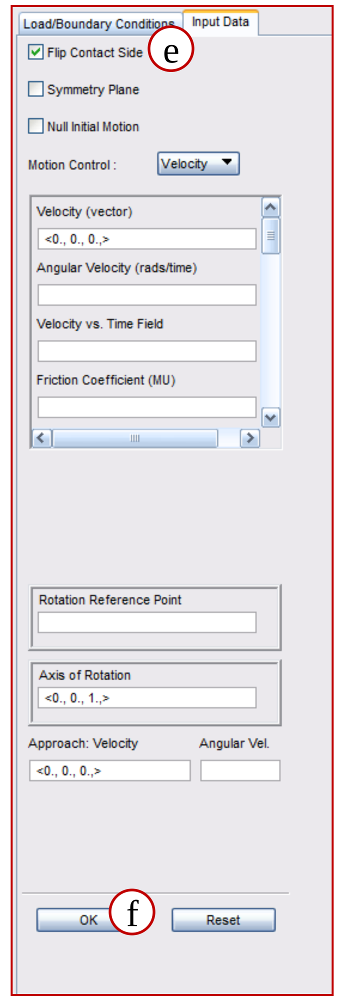
Step 5. Define the Rigid Contact bodies

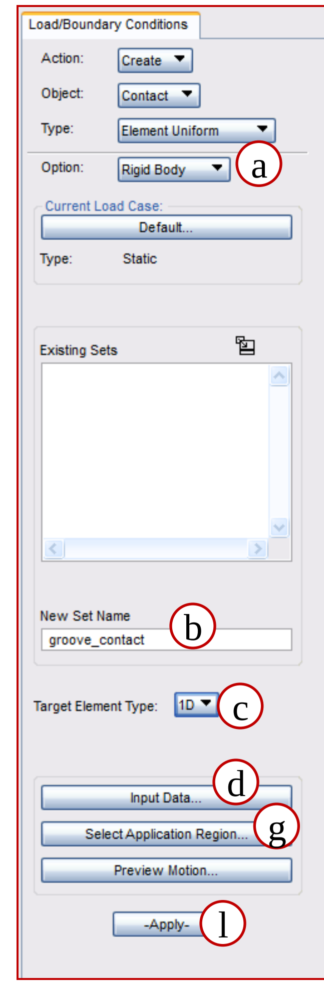


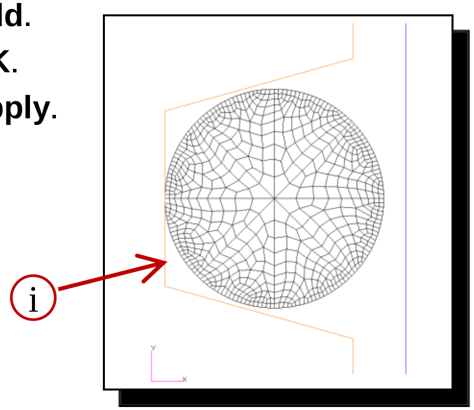
Define the grove as a rigid body.

a. Change the *Option* to be **Rigid Body**.
b. For the *New Set Name*, enter **groove_contact**.
c. Set the *Target Element Type* to be **1D**.
d. Click **Input Data**.
e. Check the *Flip Contact Side* check box.
f. Click **OK**.
g. Click **Application Region**.
h. Select **Geometry** for the *Geometry Filter*.
i. Click in the *Select Curves* text box and select the **Groove** curve on the screen or enter **Curve 6**.
j. Click **Add**.
k. Click **OK**.
l. Click **Apply**.

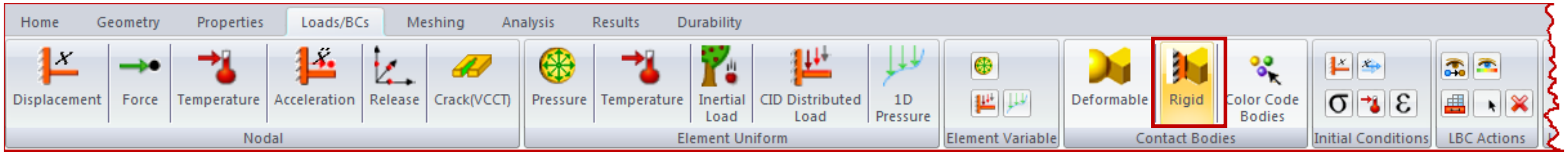






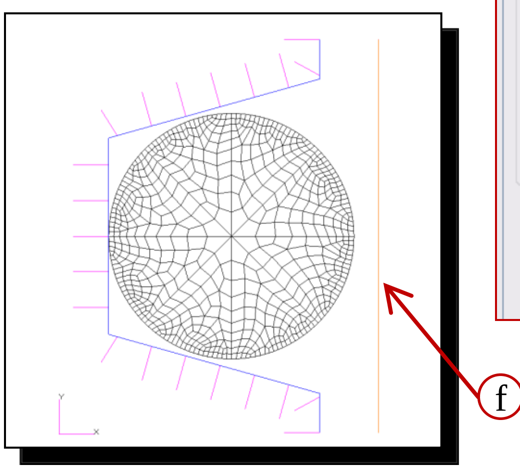


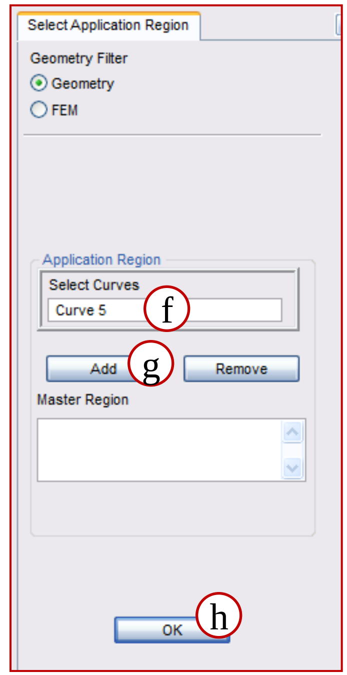
Step 5. Define the Rigid Contact Bodies

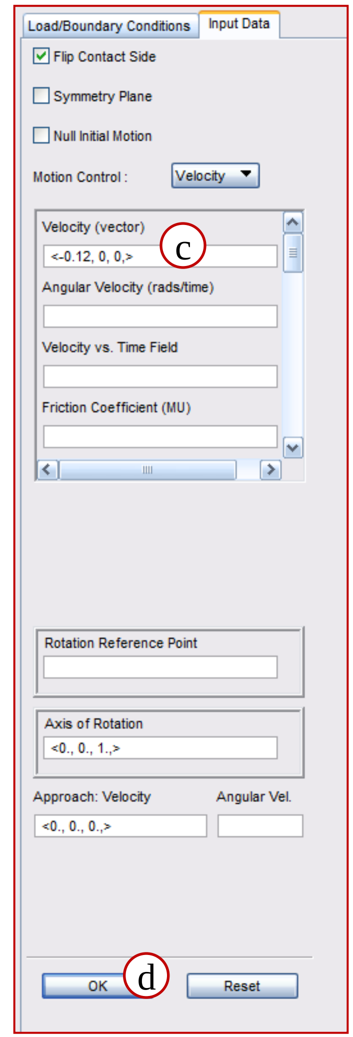


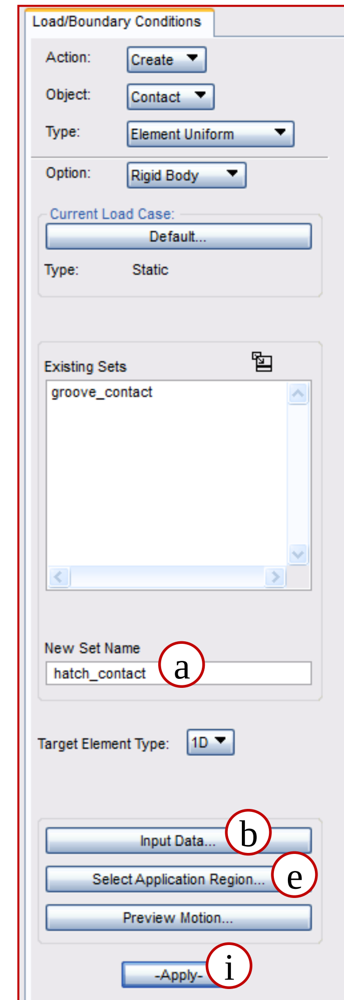
Define the hatch as a rigid body.

a. For the *New Set Name*, enter **hatch_contact**.
b. Click **Input Data**.
c. for *Velocity*, enter **<-0.12,0,0>**.
d. Click **OK**.
e. Click **Application Region**.
f. Click in the *Select Curves* text box and select the **hatch** curve on the screen or enter **Curve 5**.
g. Click **Add**.
h. Click **OK**.
i. Click **Apply**.







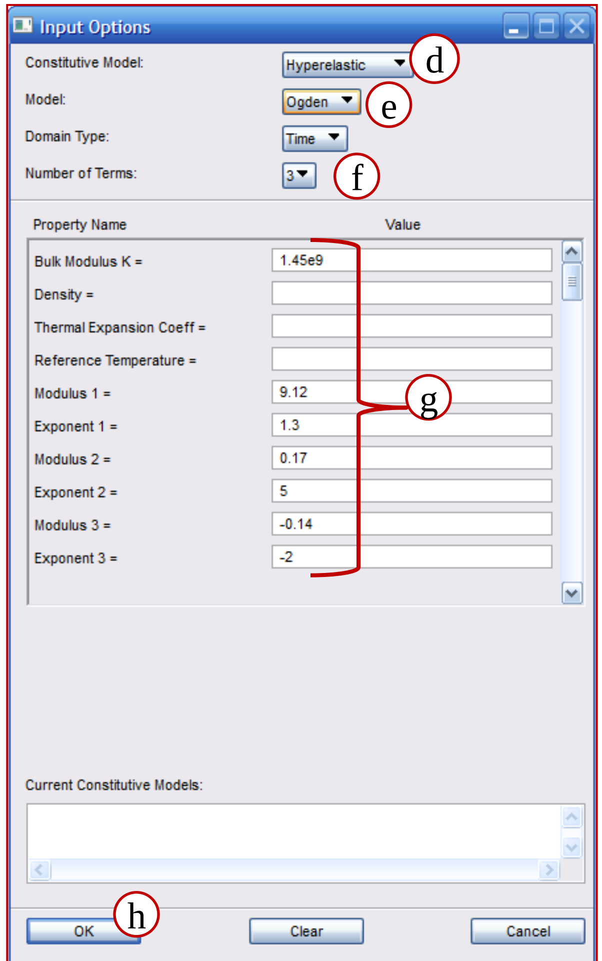


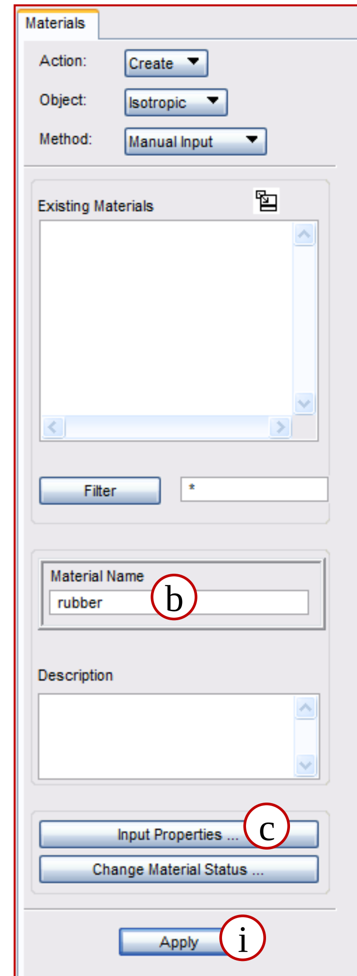
Note: Make sure the contact sides are correct as shown in the problem description page WS-4

Step 6. Define the Material

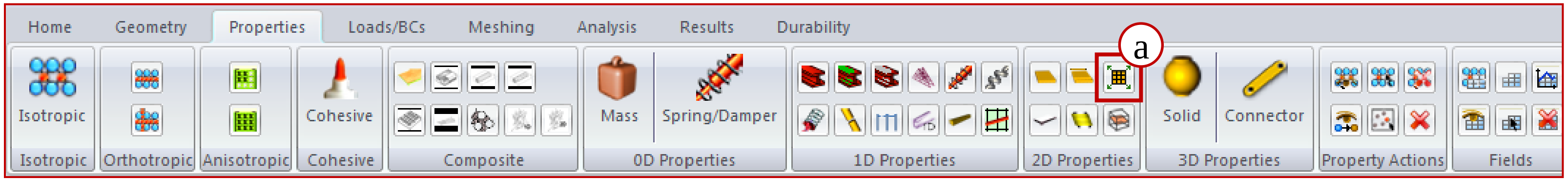
Define the material for the ring as isotropic, hyperelastic ogden with properties given on page WS-5.

a. Under the *Properties* tab, click **Isotropic** in the *Isotropic* group.
b. For the *Material Name*, enter ***rubber***.
c. Click **Input Properties**.
d. Select *Constitutive Model* to be **Hyperelastic**.
e. Select *Model* to be **Ogden**.
f. Select the *Number of Terms* to be **3**.
g. Enter the following values:
   - *Bulk Modulus* = **1.45e9**
   - *Modulus 1* = **9.12**
   - *Exponent 1* = **1.3**
   - *Modulus 2* = **0.17**
   - *Exponent 2* = **5**
   - *Modulus 3* = **-0.14**
   - *Exponent 3* = **-2**
h. Click **OK**.
i. Click **Apply**.



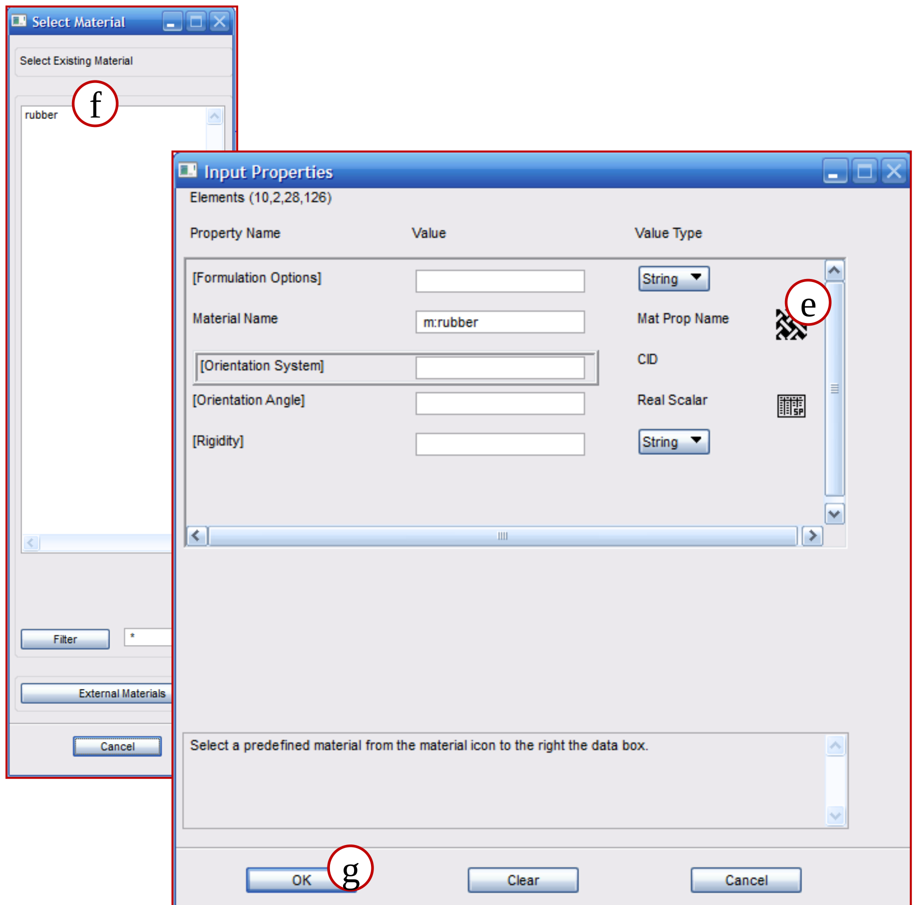


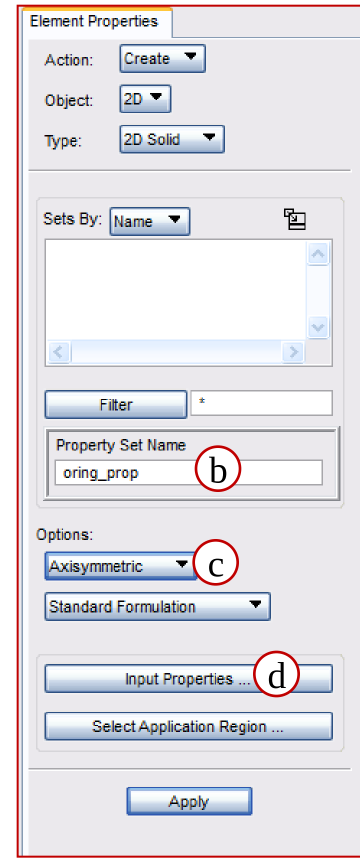
Step 7. Define Element Properties



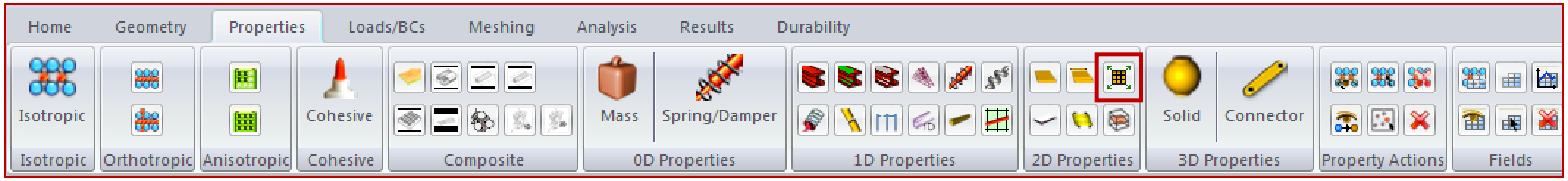
Define the o-ring properties as 2D solid, axisymmetric, and rubber.

a. Under the *Properties* tab, click **2D Solid** in the *2D Properties* group.
b. For the *Property Set Name*, enter **oring_prop**.
c. Set *Options* to be **Axisymmetric**.
d. Click **Input Properties**.
e. Click the **Material** icon.
f. From the *Select Existing material* list, select **rubber**.
g. Click **OK**.



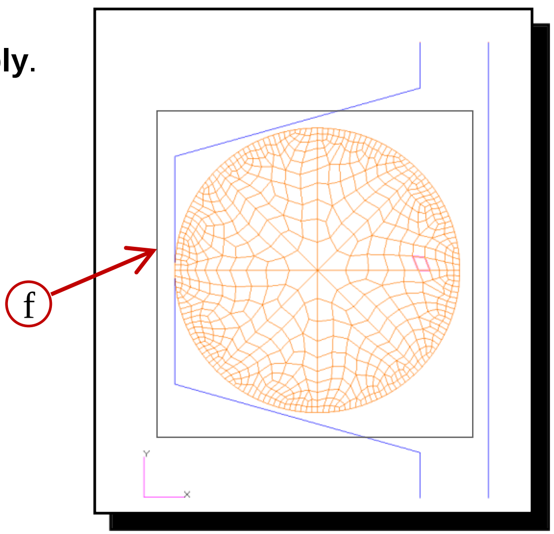


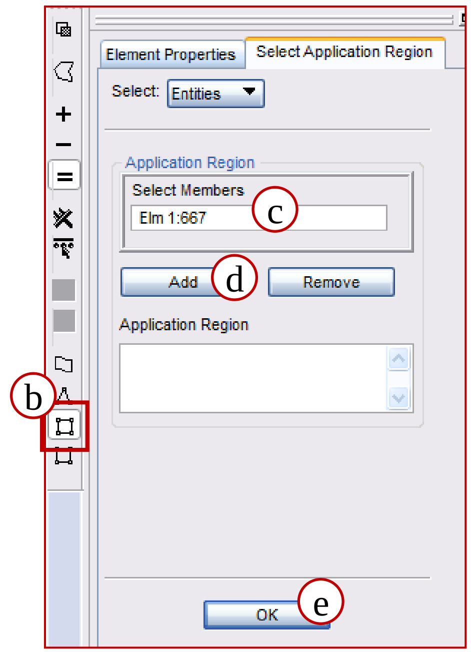
Step 7. Define Element Properties (Cont.)

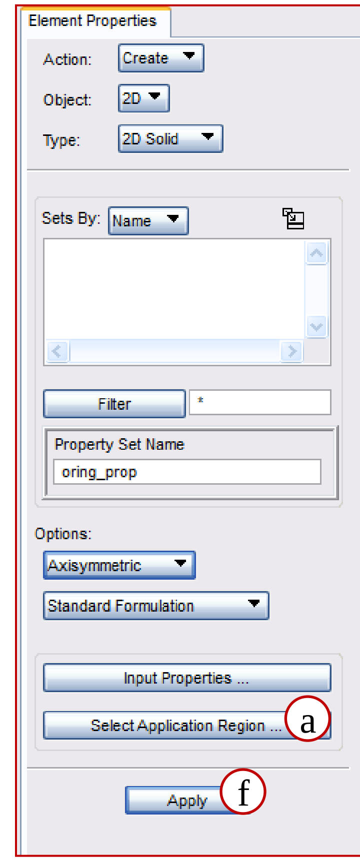


Define the o-ring properties as 2D solid, axisymmetric, and rubber continued.

a. Click **Select Application Region**.
b. Click the **Quad Element** icon
c. Click in the *Select Members* text box and select all the elements on the screen or enter **Elm 1:667**.
d. Click **Add**.
e. Click **OK**.
f. Click **Apply**.



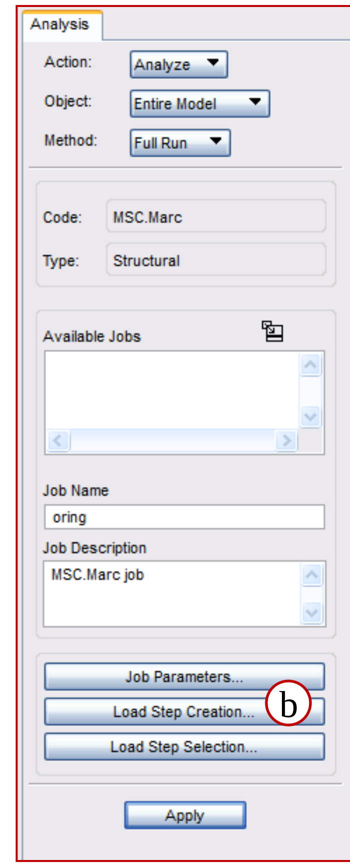




Step 8. Create Load Step and Set Solution Parameters

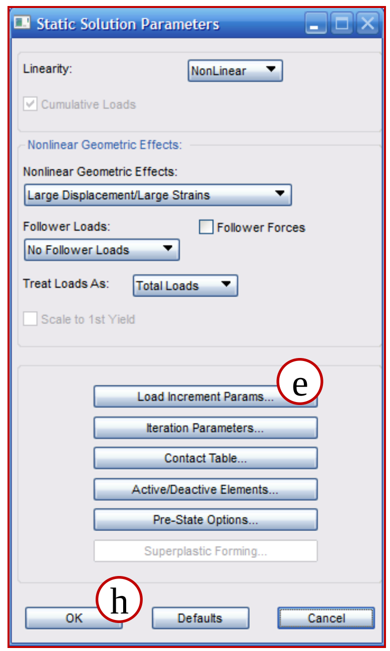
Create a load step name *Close_gap*.

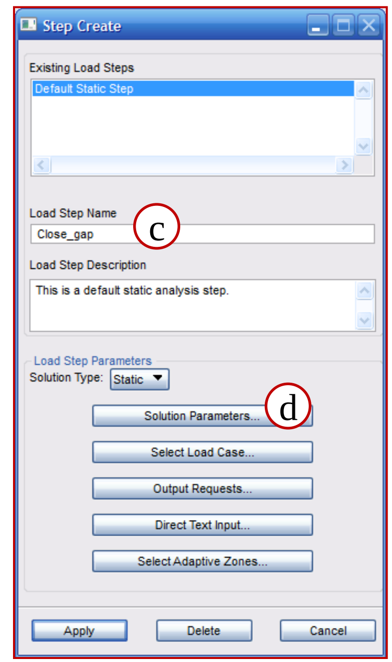
a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
b. Click **Load Step Creation**.
c. For the *Load Step Name*, enter **Close_gap**.

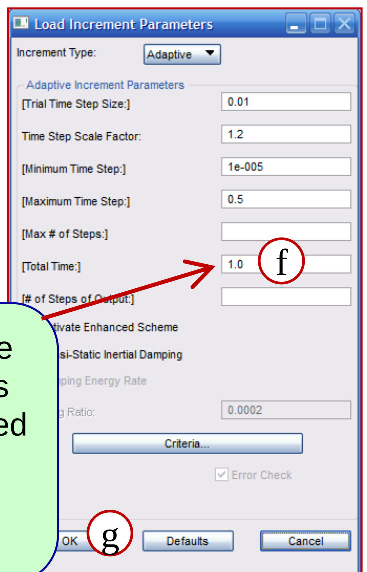


Set the solution parameter.

d. Click **Solution Parameters**.
e. Click **Load Increment Params**.
f. For *Total Time*, enter **1.0**.
g. Click **OK**.
h. Click **OK**.







Note: This is to make sure that the Hatch is moved to the specified position since the contact motion was specified by velocity.

Step 9. Request Output Results

Request output results as Contact Normal Stress, Contact Normal Force, Contact Status, and Displacement.

a. Click **Output Requests**.

b. Click **Select Nodal Results**.

c. Select the following from the *Available Result Types* list.
   - **Contact Normal Stress**
   - **Contact Normal Force**
   - **Contact Status**
   - **Displacement**

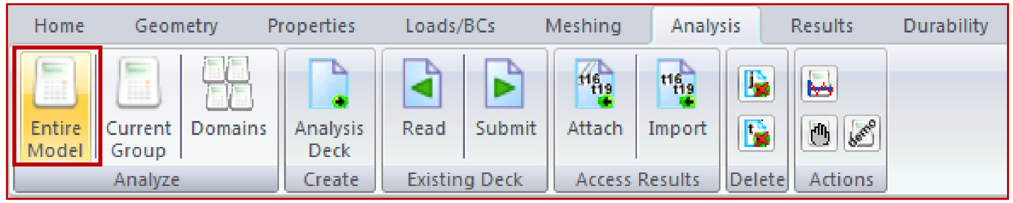
d. Check to make sure that the four result requests listed above are displayed in the *Selected Results types* box.

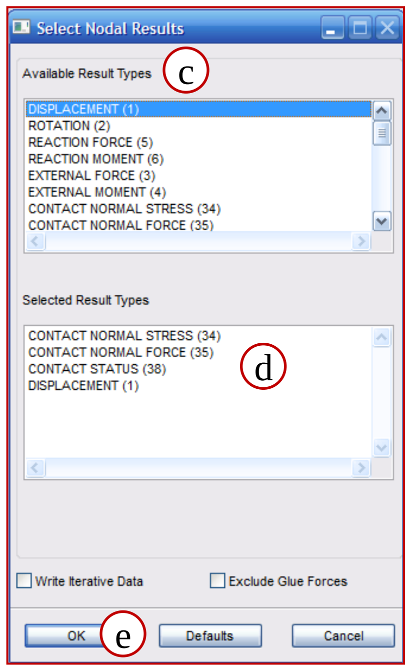
e. Click **OK**.

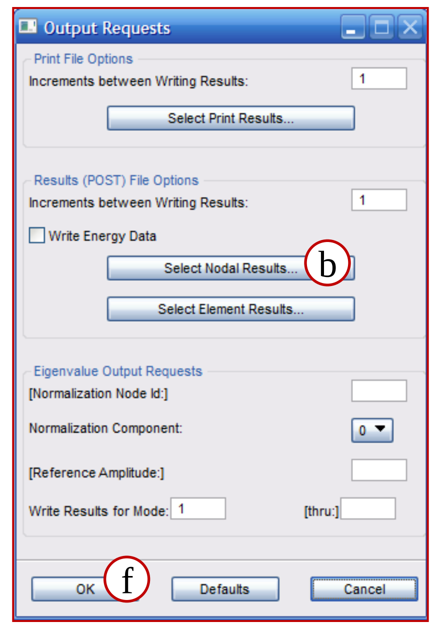
f. Click **OK**.

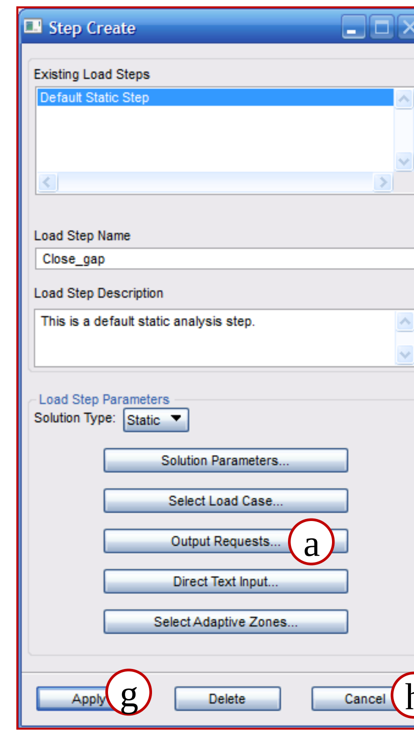
g. Click **Apply**.

h. Click **Cancel**.









Step 10. Select Load Step and Run the Analysis

Select the load step *Close_gap*.

a. Click **Load Step Selection**.

b. From the *Existing Load Steps* list, select **Close_gap**.

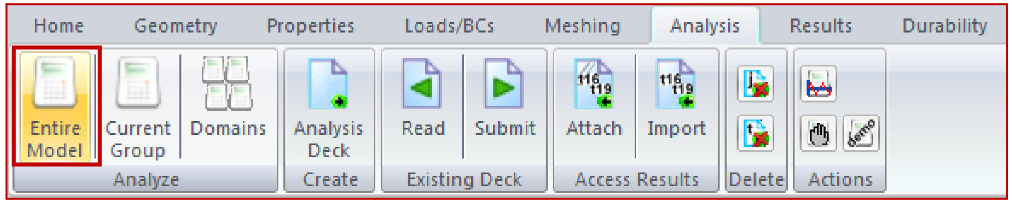
c. Under the *Selected Load Steps*, click on **Default Static Step** to de-select it.

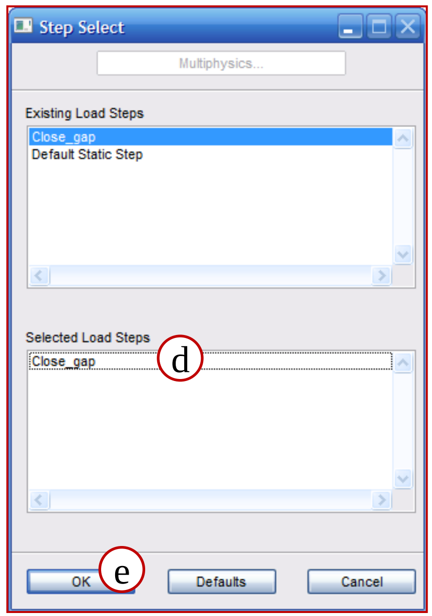
d. Check to make sure that the only load step listed under *Selected Load Steps* is **Close_gap**.

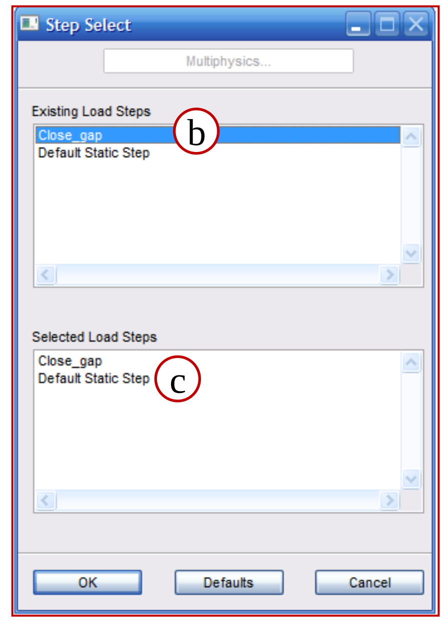
e. Click **OK**.

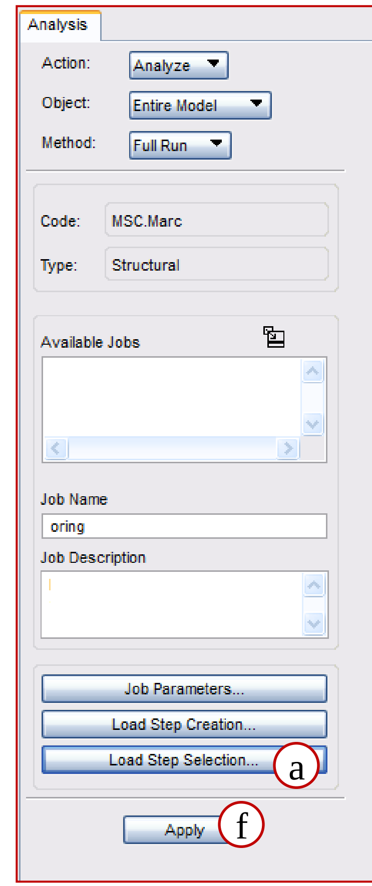
Run the analysis using Marc.

f. Click **Apply**.

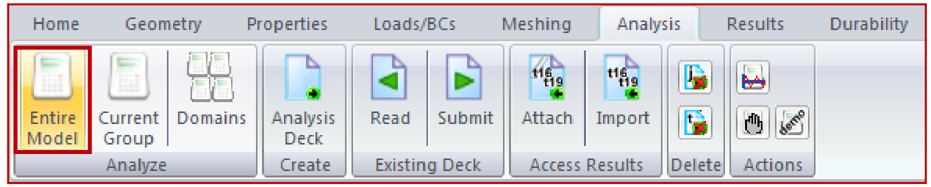








Step 11. Monitor the Job



Use the job monitoring form to monitor the job as it is running.

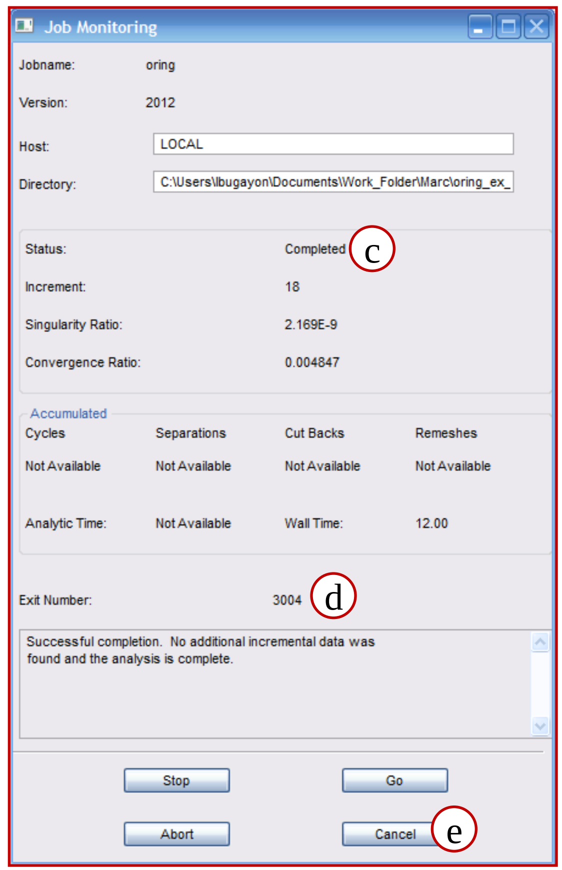
a. Change the *Action* to **Monitor** and the *Object* to **Job**.

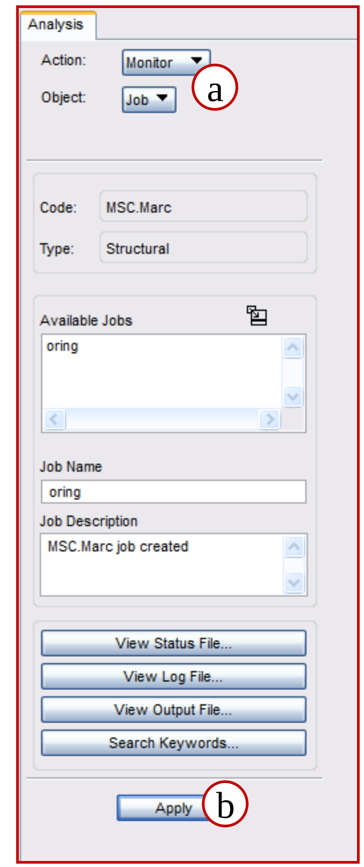
b. Click **Apply**.

c. The *Job Monitoring Form* will appear. When the job is complete the *status* will denote **Completed**.

d. For a job without errors the exit number should be **3004**

e. When done reviewing the form, click **cancel**.

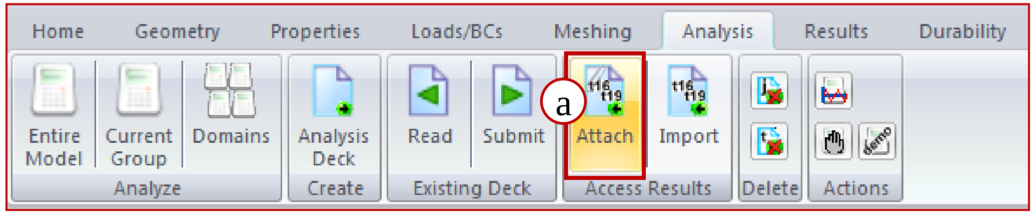


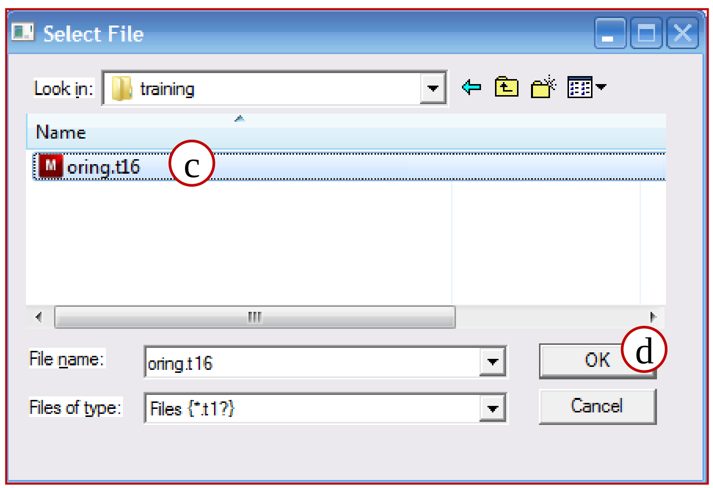


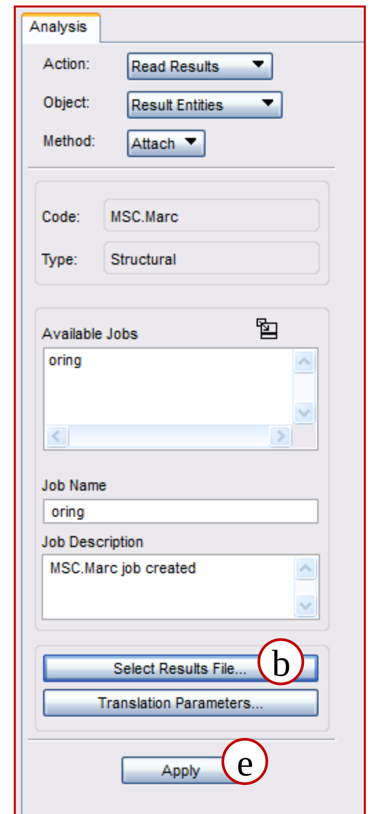
Step 12. Attach Results File

Attach the marc results file *oring.t16* to Patran.

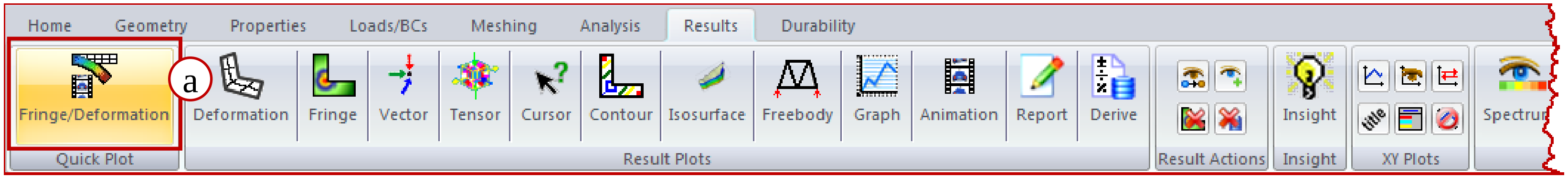
a. Under the *Analysis* tab, click **Attach** in the *Access Results* group.
b. Click **Select Results File**.
c. From the list select the result file **oring.t16**.
d. Click **OK**.
e. Click **Apply**.







Step 13. Plot Normal Contact Stresses



To plot normal contact stresses:

a. Under the *Results* tab, click **Fringe/Deformation** in the *Quick Plot* group.

b. From the *Select Result Cases* list, select the last increment.

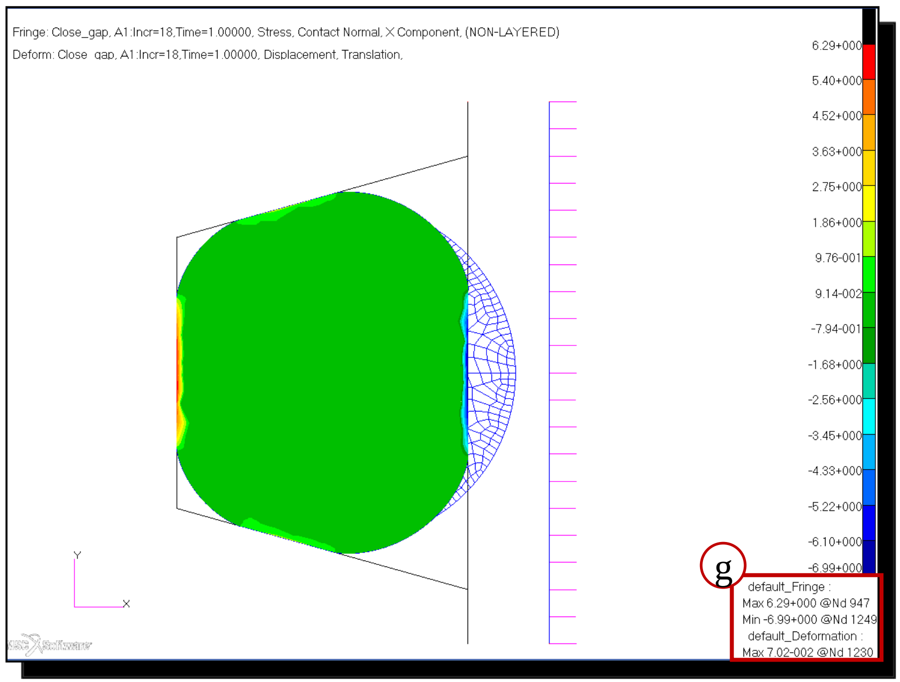
c. From the *Select Fringe Result* list, select **Stress, Contact Normal**.

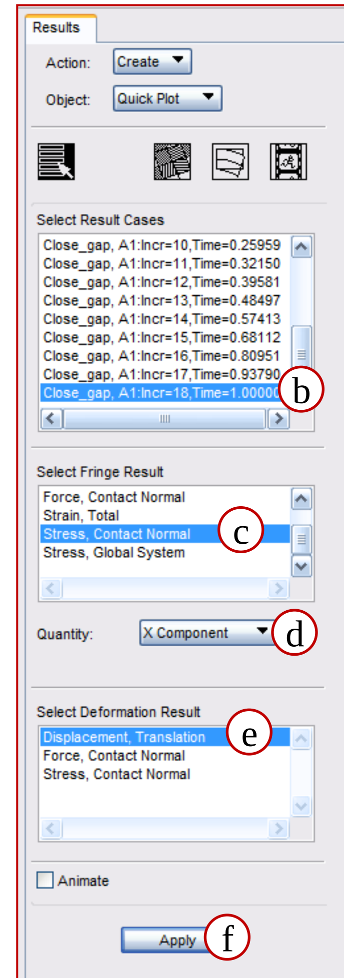
d. Set *Quantity* to be the **X Component**.

e. From the *Select Deformation Result* list, select **Displacement, Translation.**

f. Click **Apply.**

g. Notice that the *Maximum Contact Stress* is **6.29 PSI** at node **947**.

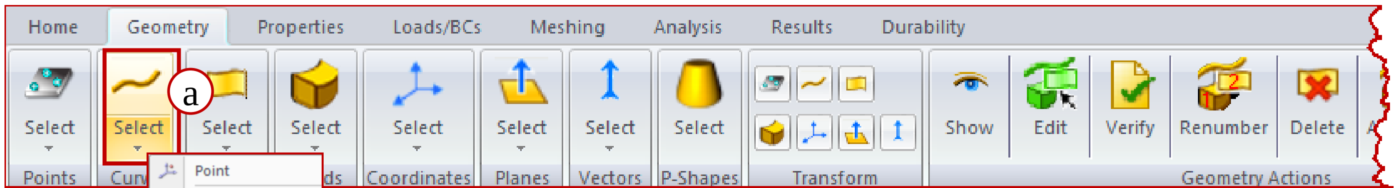


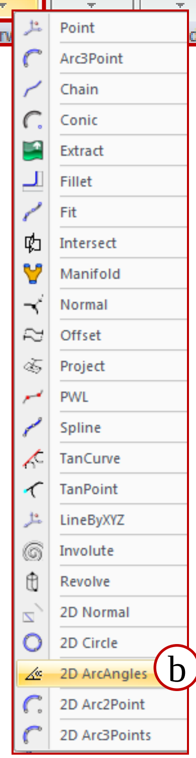


Step 14. Graph Contact Stresses Along the Surface of the Ring

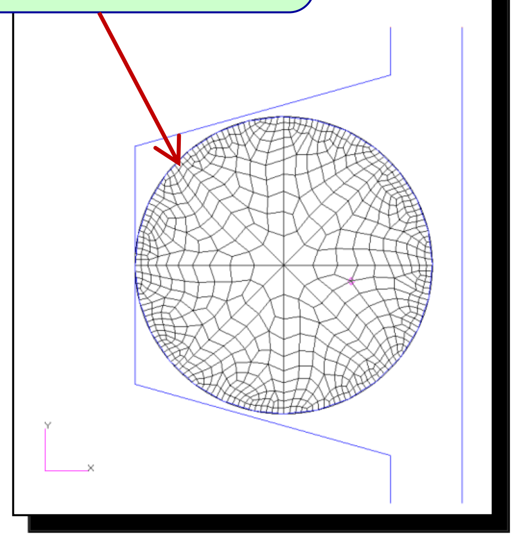
First create a curve to represent the surface of the ring. This curve will be used for the purpose of plotting.

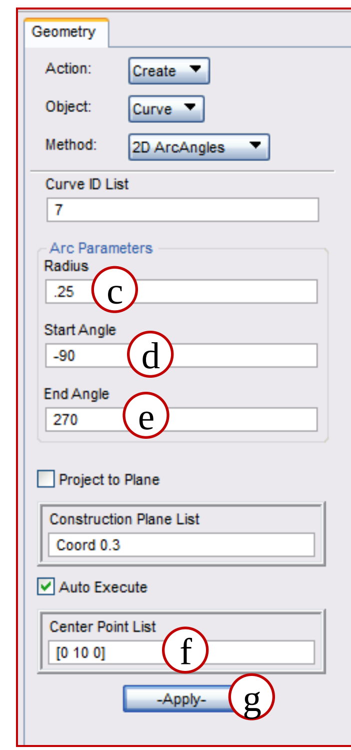
a. Under the *Geometry* tab, click **Select** in the *Curves* group.
b. From the drop down menu select **2D ArcAngles**.
c. For the *Radius*, enter **0.25**.
d. For the *Start Angle*, enter **-90**.
e. For the *End Angle*, enter **270**.
f. In *Center Point List* text box, enter **[0,10,0]**.
g. Click **Apply**.



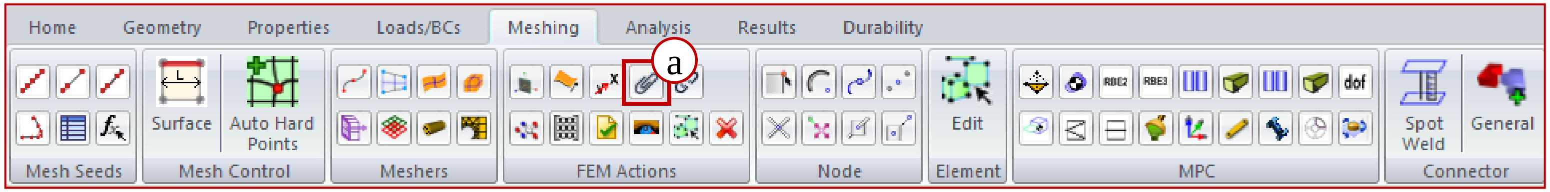


Created curve is on the circumference of the ring.



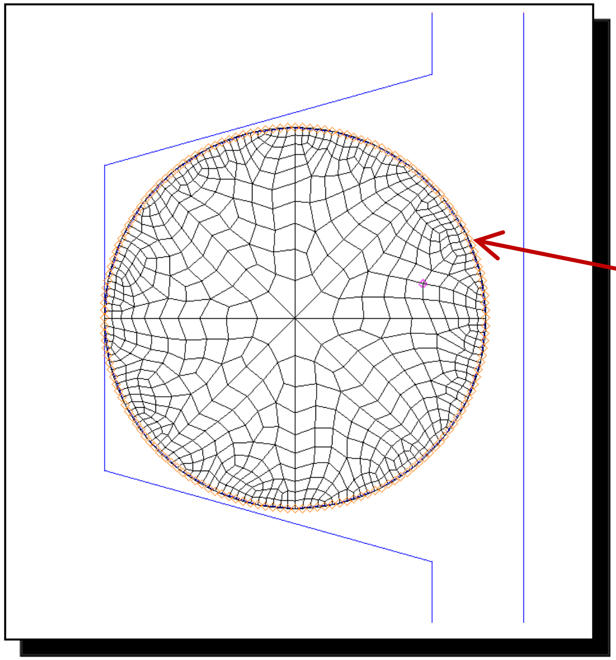


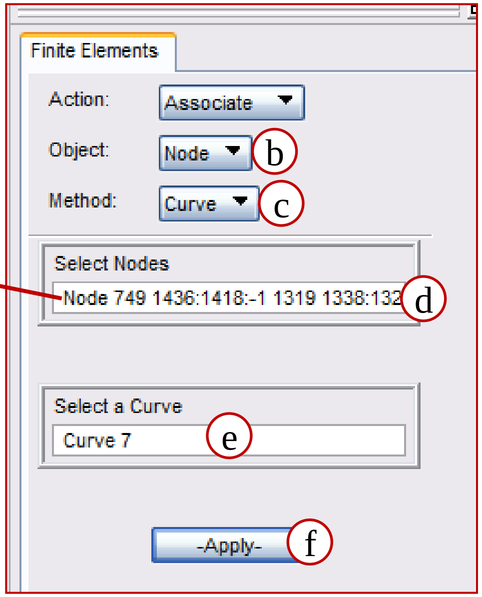
Step 14. Graph Contact Stresses Along the Surface of the Ring (Cont.)



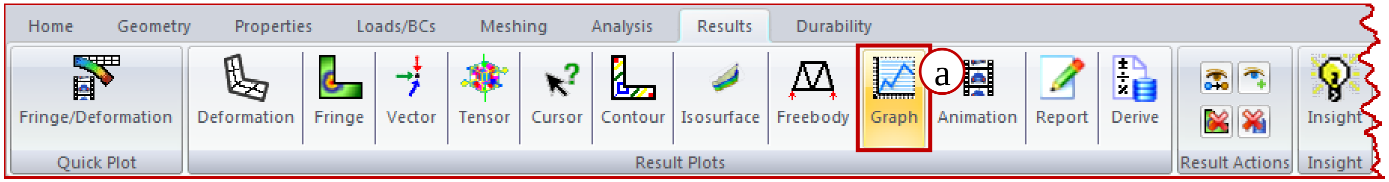
Next, associate nodes to the curve

a. Under the *Meshing* tab, click **Associate** in the *Fem Actions* group.
b. Set the *Object* to **Node**.
c. Set the *Method* to **Curve**.
d. Click in the *Select Nodes* text box and select all the nodes along the boundary of the ring.
e. Click in the *Select a Curve* and select the curve on the ring surface or enter **Curve 7**.
f. Click **Apply**





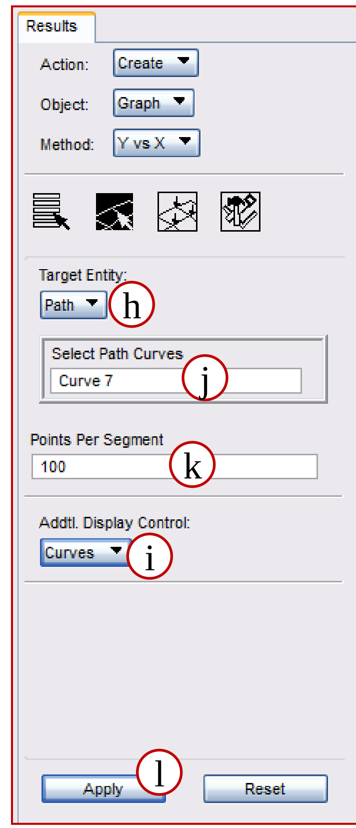
Step 14. Graph Contact Stresses Along the Surface of the Ring (Cont.)

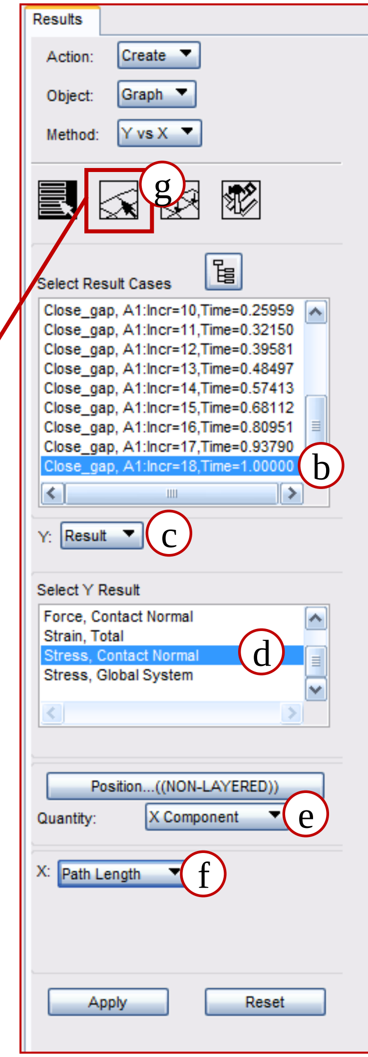


Using the curve previously created, graph contact stress along the surface of the ring.

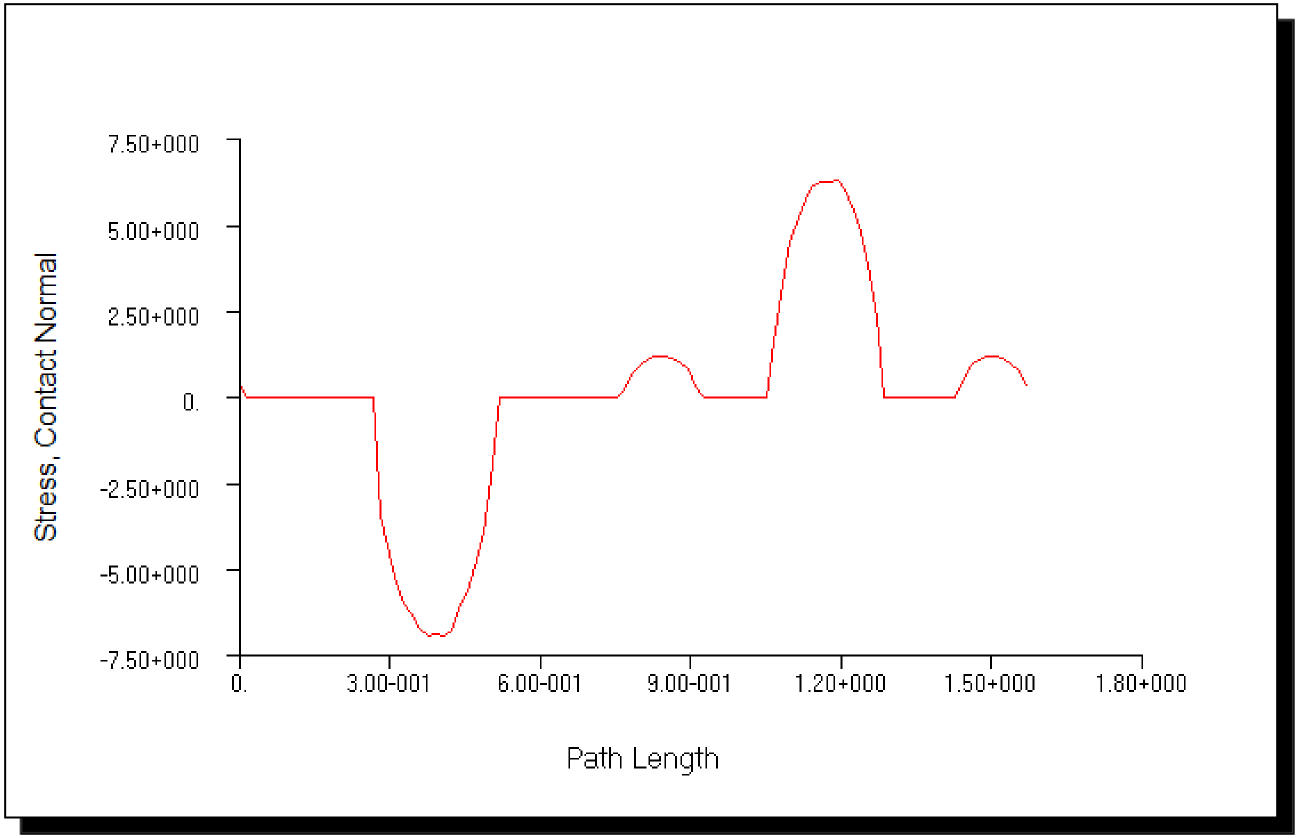
a. Under the *Results* tab, click **Graph** in the *Results Plots* group.
b. From the *Select Result Case* list, select the last increment.
c. For *Y* select **Result**.
d. From the *Select Y Result* list, select **Stress, Contact Normal**.
e. Set *Quantity* to be **X Component**
f. For *X* select **Path Length**.
g. Click the **Target Entities** icon.
h. Set *Target Entity* to be **Path.**
i. Set *Addtl. Display Control* to be **Curves**.
j. Click in the *Select Path Curves* text box and select the curve on the ring circumference from the screen or enter **Curve 7**.
k. For *Point Per Segment*, enter **100**.
l. Click **Apply**.

Graph of contact stresses along the surface of the ring shown on the next page.

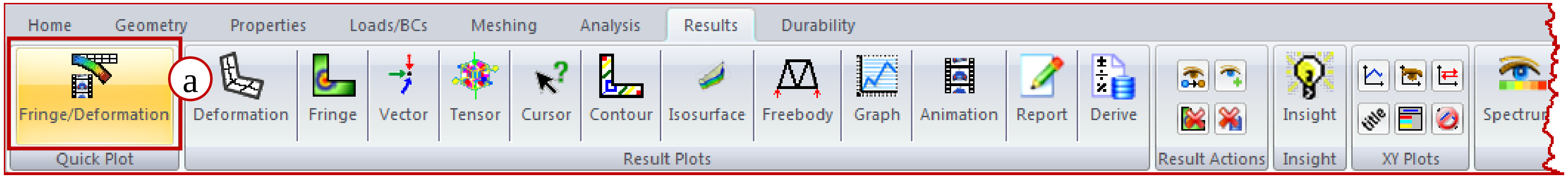




Step 14. Graph Contact Stresses Along the Surface of the Ring



Step 15. Plot Minimum Principal Stresses



To plot minimum principal stresses:

a. Under the *Results* tab, click **Fringe/Deformation** from the *Quick Plot* group.

b. From the *Select Result Case* list, select last increment.

c. From the *Select Fringe Result* list, select **Stress, Global System**.

d. Set *Quantity* to be **Min Principal**.

e. From the *Select Deformation Result* list, select **Displacement ,Translation**.

f. Click **Apply**.

g. Notice that the *Maximum Compressive Stress* is **7.0 PSI**