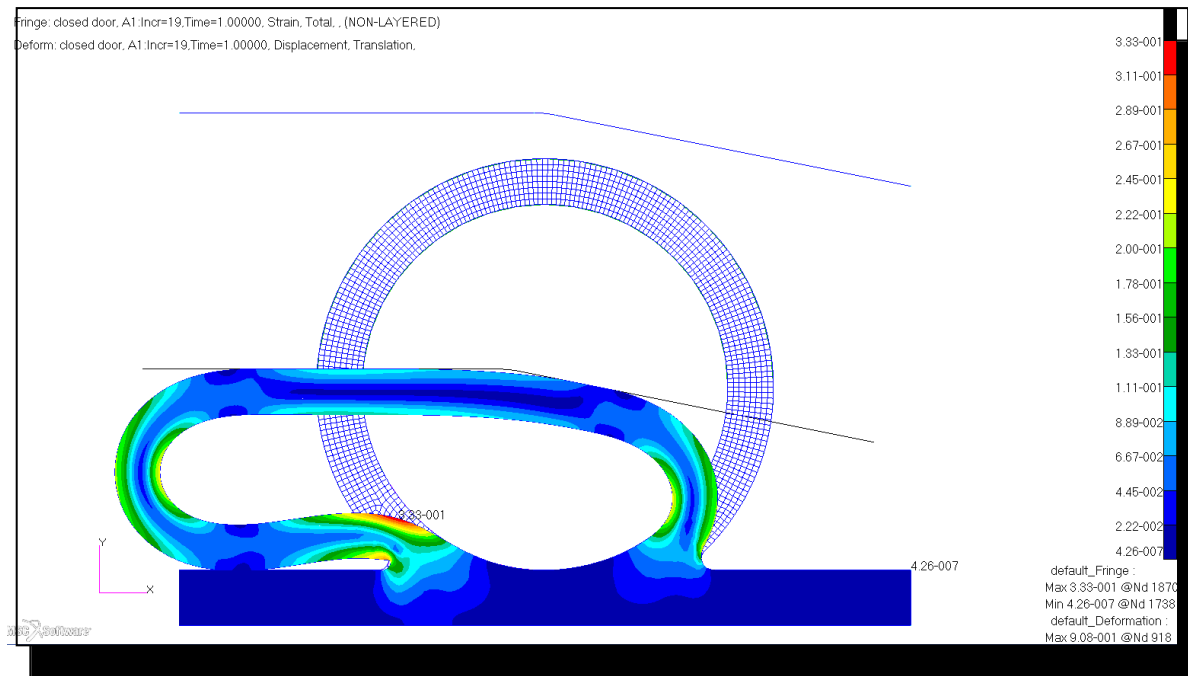


# WORKSHOP 4

## ANALYSIS OF A RUBBER SEAL



- **Objectives:**
  - To perform a Large Displacement/Large Strain analysis
    - Use contact analysis with rigid body contact
    - Use hyperelastic material properties.
- **Software Version:**
  - Marc 2012
  - Patran 2012
- **Files Required:**
  - rubber\_seal.ses
- **Problem Description:**
  - In this exercise you will analyze a door seal. The purpose of the analysis is to examine the stresses and deflections created during the closing of a door. The seal is made of a rubber material and therefore will be modeled using hyperelastic material properties. The door is considered very stiff relative to the rubber seal and can be modeled as a rigid body.

- **Suggested Exercise Steps:**

1. Create a new database called *Rubber\_Seal.db*.
2. Import the model by playing the provided session file, *rubber\_seal.ses*.
3. Mesh all surfaces with quad elements of 0.015 edge length.
4. Check the model for free edges.
5. Remove duplicate nodes.
6. Re-check the model for free edges.
7. Verify that all the element normals are pointing in the positive Z direction.
8. Change any incorrect normals to point in the positive Z direction.
9. Create a fixed boundary condition on the bottom of the rubber seal.
10. Define the deformable contact body as the rubber seal.
11. Define the rigid contact body as the door.
12. Preview the rigid body motion.
13. Create the rubber material with the following properties:
  - Constitutive model - Mooney-Rivlin, hyperelastic formulation
  - Strain Energy Function (C10) = 80
  - Strain Energy Function (C01) = 20

- **Suggested Steps (Cont.)**

14. Define the element properties:

- Options: 2D, Plane strain, Hybrid(Herrmann)/Red. Int.
- Include all Rubber Seal elements

15. Set the Job parameter to MSC Marc Version 2012.

16. Create the load step:

- Set the name to *closed door*
- Define the output requests as the following:
  - Strain, total components (301)
  - Stress components (global system ) (411)
  - Stress components (311)

17. Select the load step and run the analysis using Marc.

18. Monitor the job, make sure that the exit number is 3004.

19. Attach the Marc results file *rubber\_seal.t16*.

20. Plot the strain results.

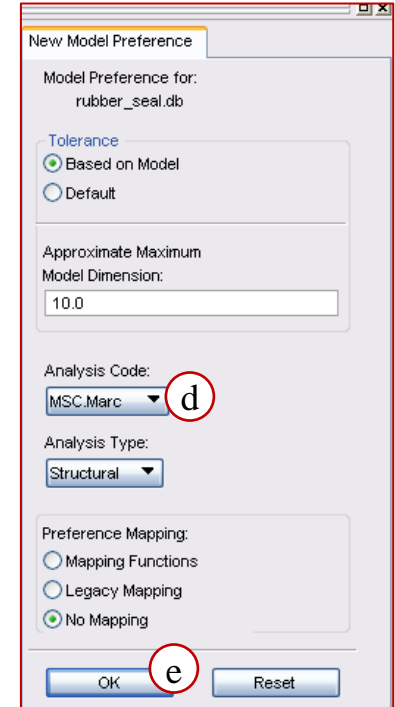
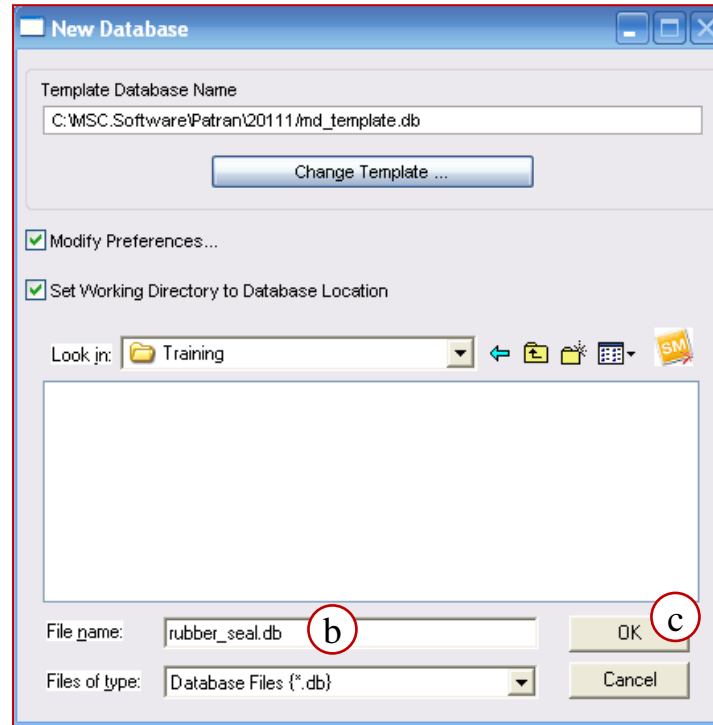
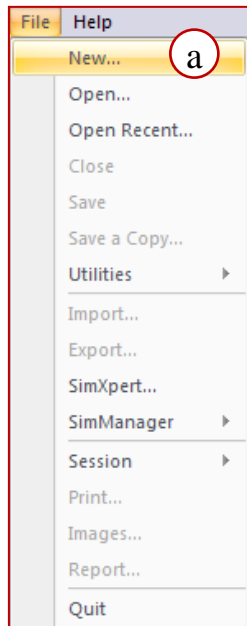
21. Create a graph of Load vs. Displacement.

22. Quit Patran

# Step 1. Create a New Database

Create a new database called *rubber\_seal.db*.

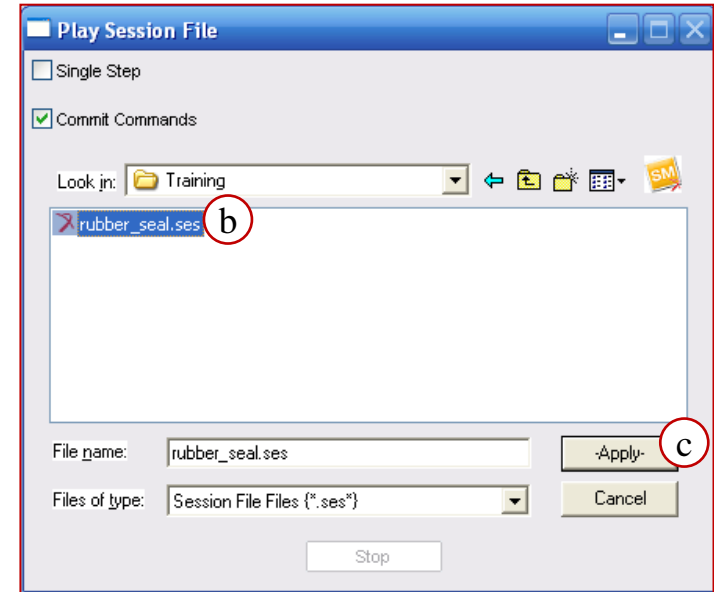
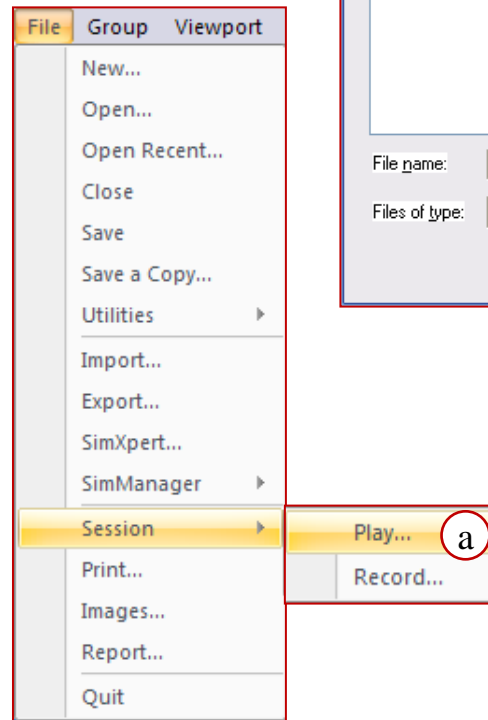
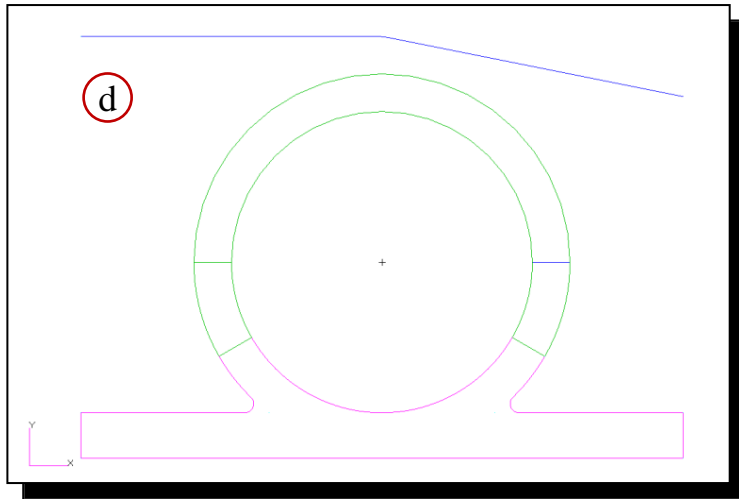
- a. Pull down **File > New**.
- b. Enter **rubber\_seal** as the *File name*.
- c. Click **OK**.
- d. Select **MSC.Marc** as the *Analysis Code*.
- e. Click **OK**.



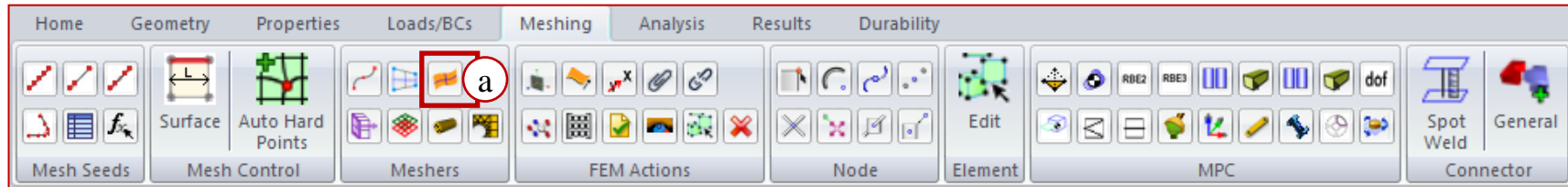
## Step 2. Run the Provided Session File

Import the model by playing the provided session file, *rubber\_seal.ses*.

- a. Pull down *File > Session > Play*.
- b. Select **rubber\_seal.ses** as the *File name*.
- c. Click **Apply**. Note: This action will run the session file. Please do not interrupt it.
- d. When finished the geometry should look like the diagram shown below.

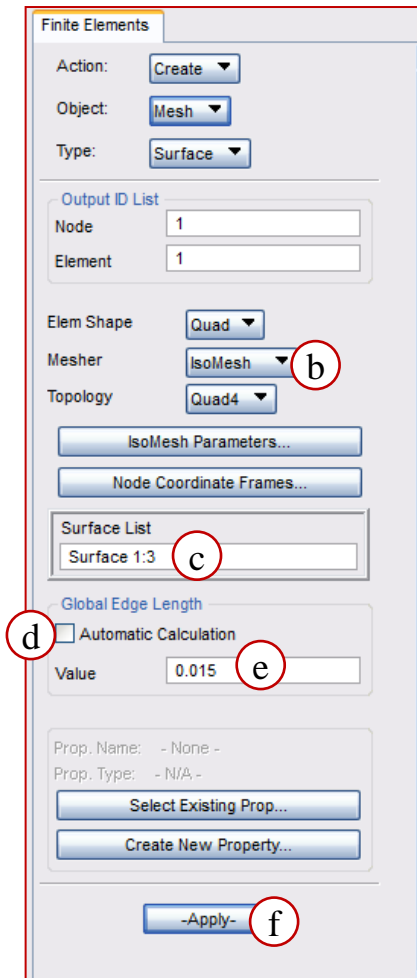
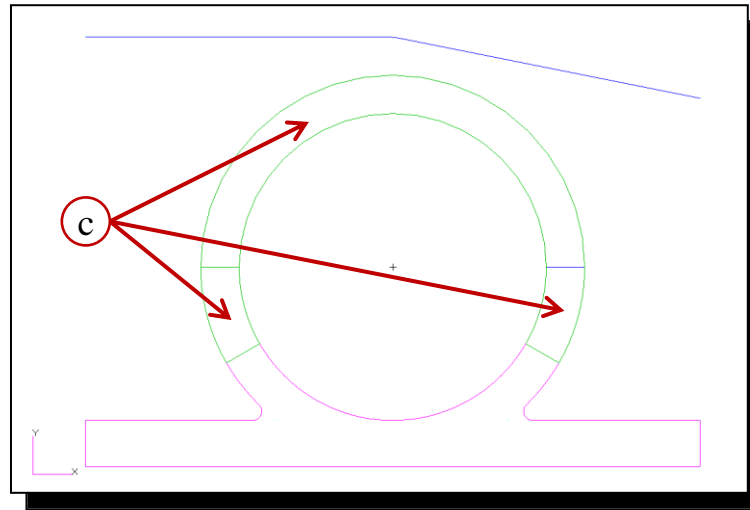


# Step 3. Mesh the Surfaces



Create the mesh on the top three surfaces with quad elements of 0.015 edge length.

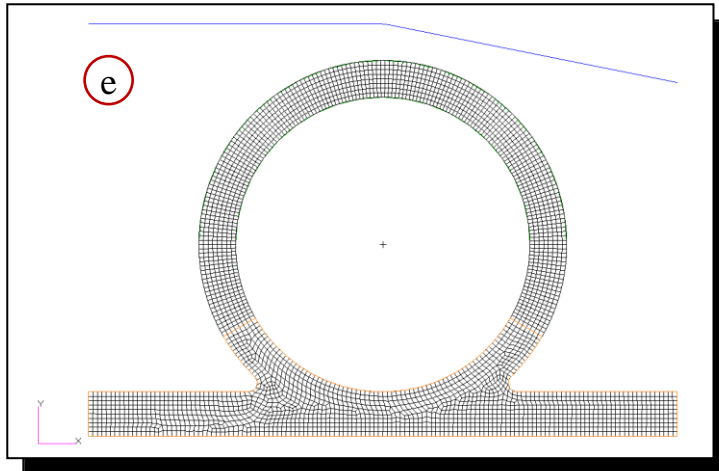
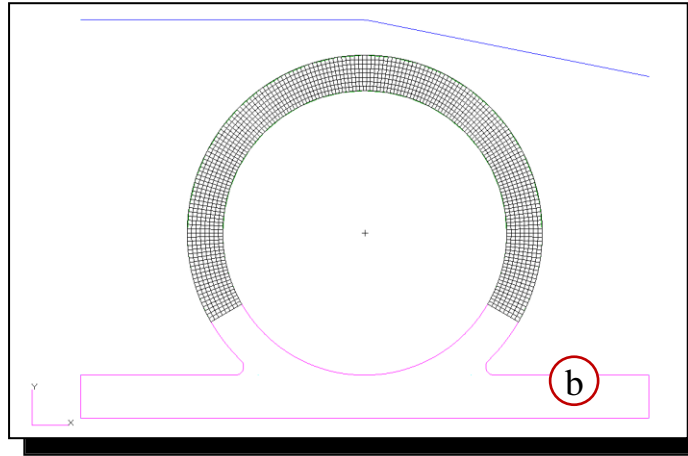
- Under the *Meshing* tab, click **Surface** in the *Meshers* group
- Pull down **IsoMesh** as the *Mesher*.
- Click in the *Surface List* text box and pick all three green surfaces (Surface 1:3). To select multiple surfaces, hold the SHIFT key down while picking.
- Uncheck **Automatic Calculation**.
- Enter **0.015** as the *Value*.
- Click **Apply**.



# Step 3. Mesh the Surfaces (Cont.)

Create the mesh on the bottom, multi-sided surface with quad elements of 0.015 edge length.

- Pull down **Paver** as the *Mesher*.
- Click in the *Surface List* text box and pick the bottom (magenta) surface (Surface 4).
- Verify that **0.015** is used for *Value*.
- Click **Apply**.
- The mesh should look like the one in the image below.



Finite Elements

Action: **Create**

Object: **Mesh**

Type: **Surface**

Output ID List

Node: 3451

Element: 3075

Elem Shape: **Quad**

Mesher: **Paver** a

Topology: **Quad4**

Paver/Hybrid Parameters...

Node Coordinate Frames...

Surface List

Surface 4 b

Global Edge Length

☐ Automatic Calculation

Value: 0.015 c

Prop. Name: - None -

Prop. Type: - N/A -

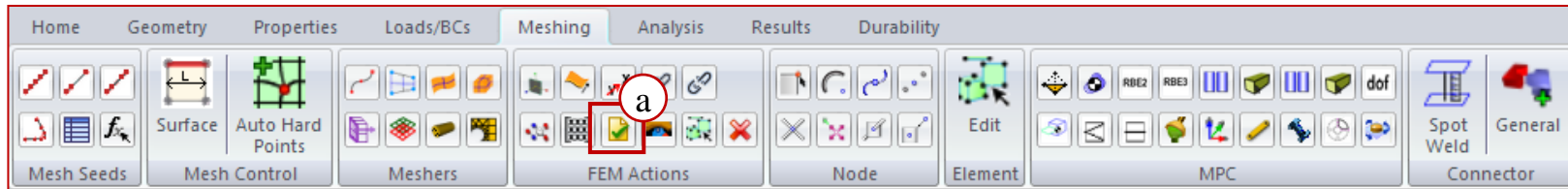
Select Existing Prop...

Create New Property...

-Apply- d



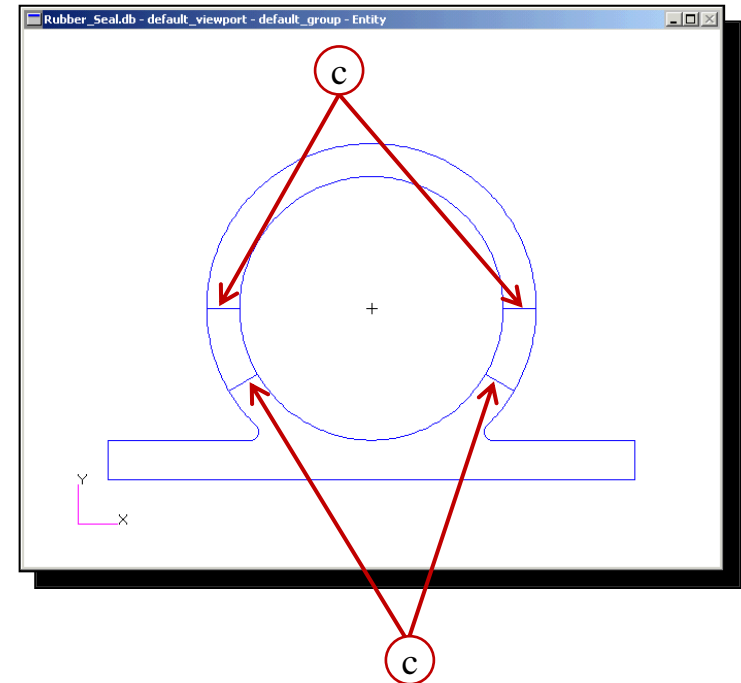
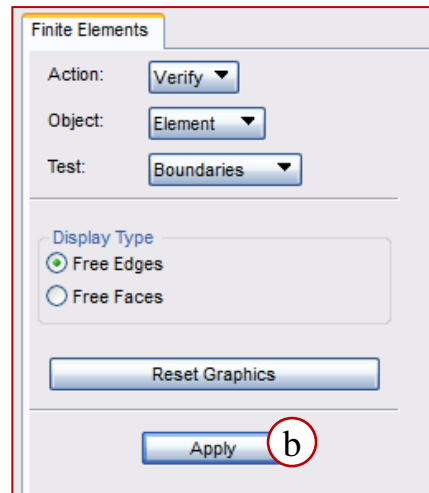
# Step 4. Check the Model for Free Edges



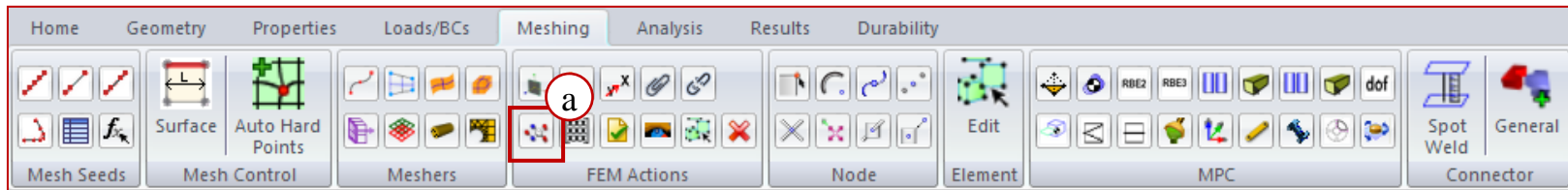
To check the model for free edges:

- a. Click **Verify** in the *FEM Actions* group.
- b. Click **Apply**.
- c. Note that 4 free edges were found.

This action will show where there are free element edges. Free element edges are essentially 'cracks' in the model where duplicate nodes exist. Patran will not automatically remove duplicate nodes when meshing congruent surfaces like the ones in this model. Step 5 will show the **Equivalence** action to remove duplicate nodes.

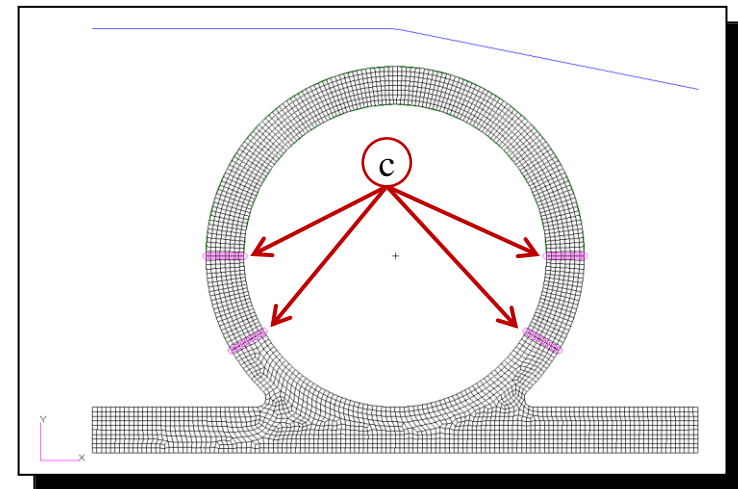
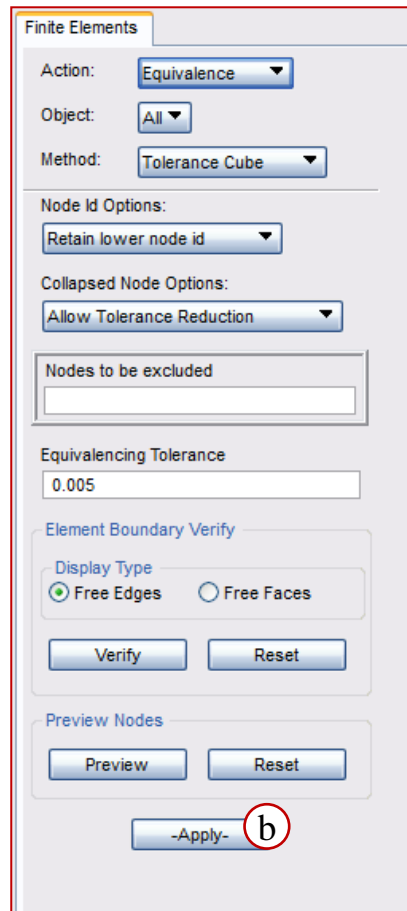


# Step 5. Remove the Duplicate Nodes



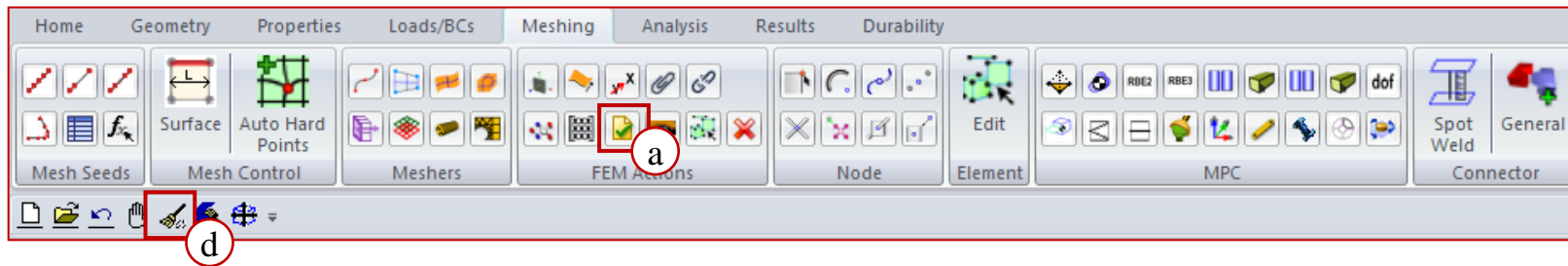
To remove duplicate nodes:

- a. Click **Equivalence** in the *FEM Actions* group
- b. Click **Apply**.
- c. The merged nodes are circled in magenta.



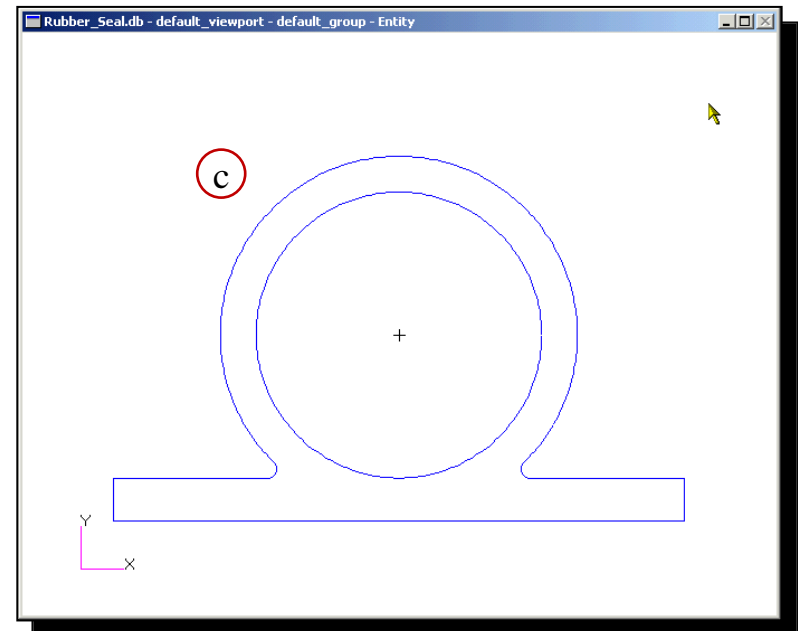
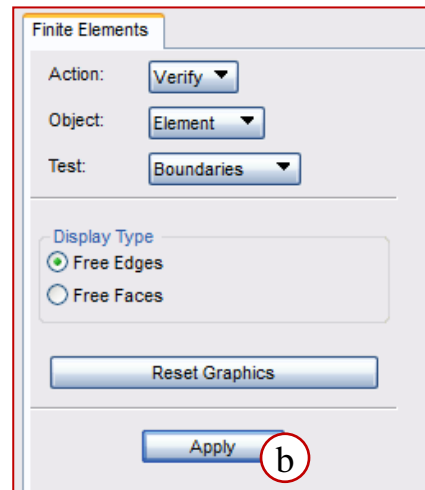
Equivalence any duplicate nodes created during meshing.  
This process will merge all co-located nodes.

# Step 6. Re-Check the Model for Free Edges



To re-check for free edges:

- Click **Verify** in the *FEM Actions* group.
- Click **Apply**.
- Notice that after the equivalence operation the interior free edges are gone.
- Click **Reset Graphics**.



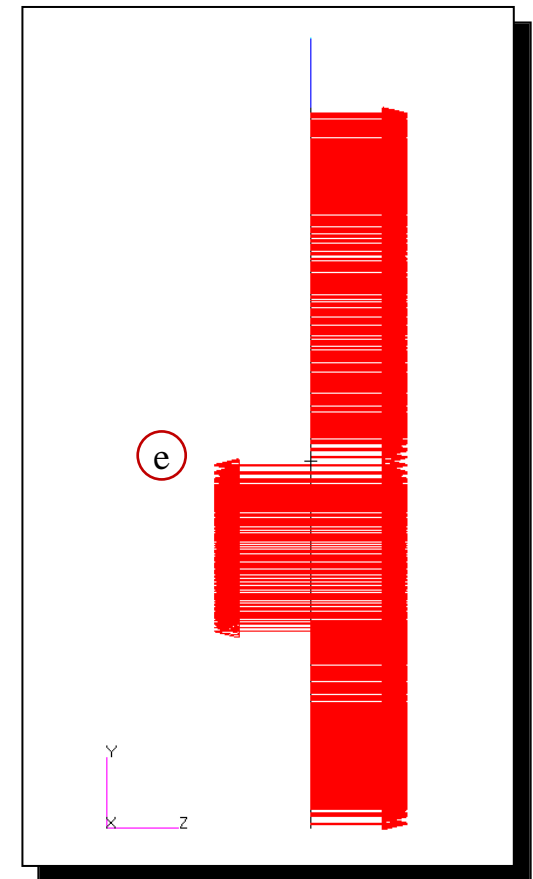
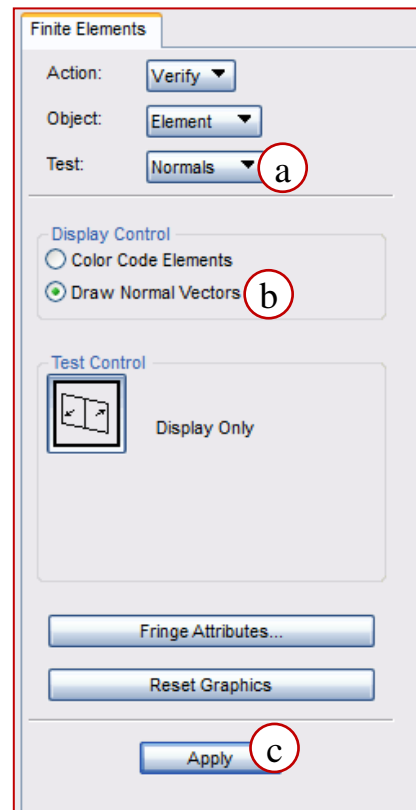
# Step 7. Verify the Element Normals



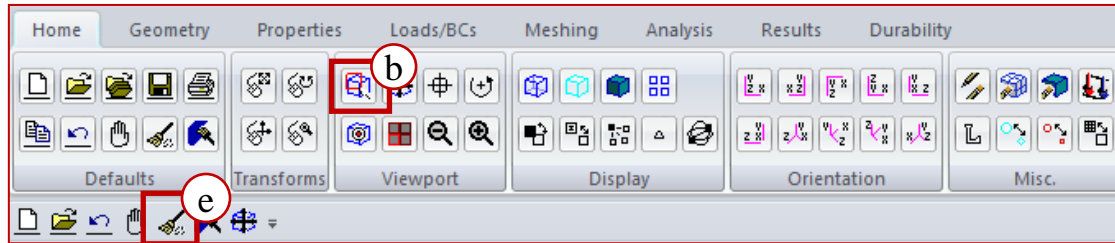
Verify that all the element normals are pointing in the positive Z direction.

- Pull down *Test* to **Normals**.
- Select **Draw Normal Vectors**.
- Click **Apply**.
- Under the *Home* tab, click **Left Side View** in the *Orientation* group.
- Notice that not all the element normals are pointed in the positive Z direction.

Since this is a 2-D solid model, all element normals must point in the positive Z-direction. Verify the element normals. The normals that point in the wrong direction will need to be corrected in the next step.

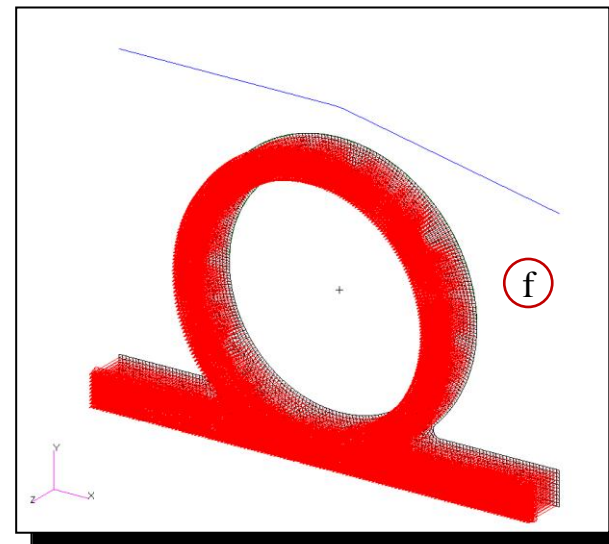
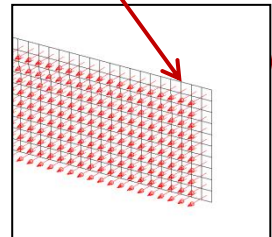
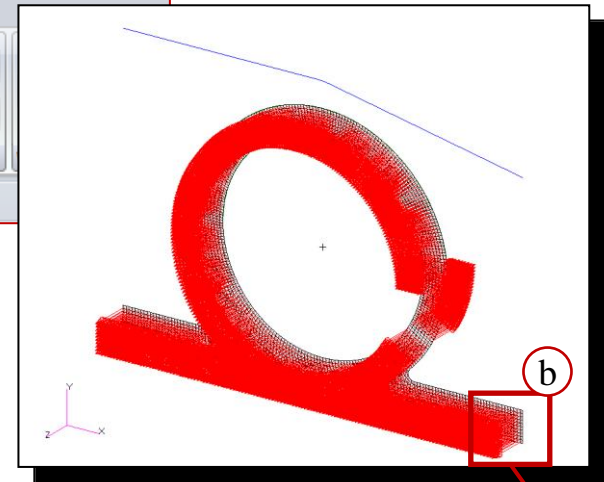
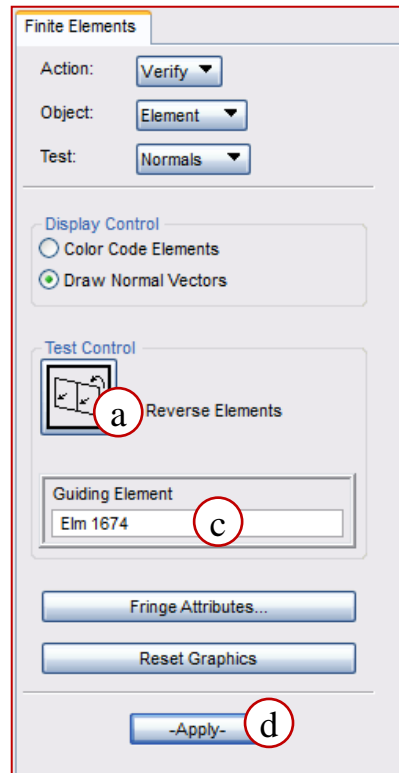


# Step 8. Reverse Incorrect Normals

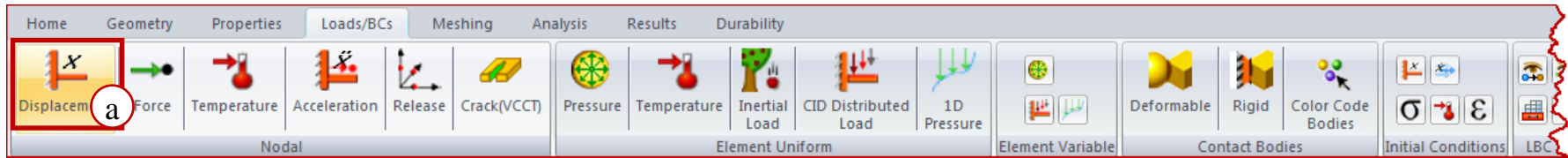


Change the incorrect normals to point in the positive Z direction:

- Click the **Display Only** icon to toggle to **Reverse Elements** mode.
- Zoom in on the right edge of the rubber seal.
- Click in the *Guiding Element* text box and pick any element from the portion where you zoomed in, (since all these element normals are pointing in the correct direction.)
- Click **Apply**.
- Click **Reset Graphics**.
- All the element normals should now be pointing in the positive Z direction.

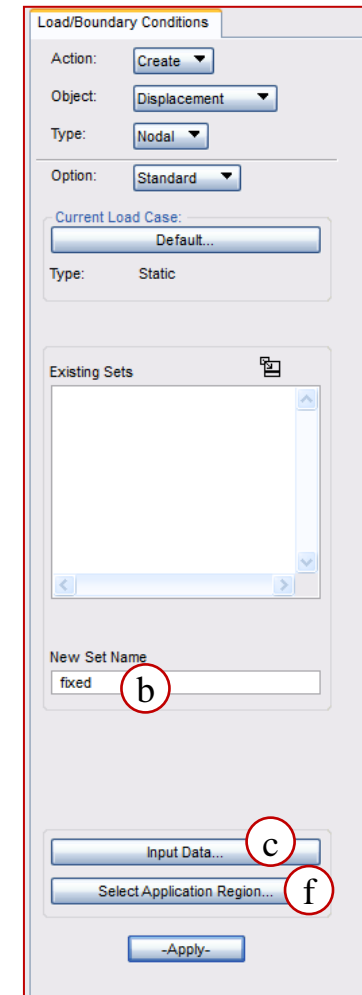
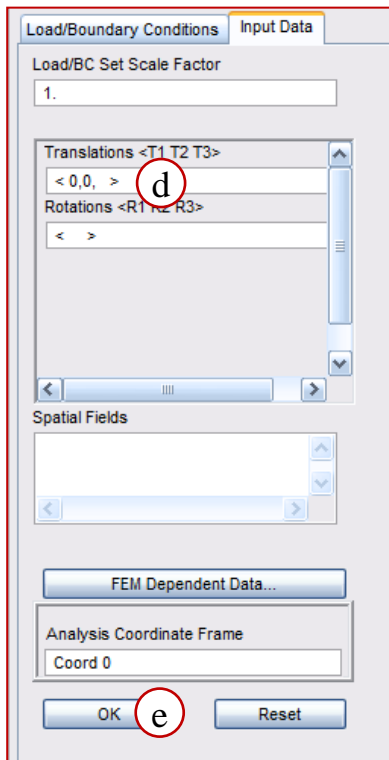


# Step 9. Create Fixed Boundary Condition



Create a fixed boundary condition on the bottom of the rubber seal:

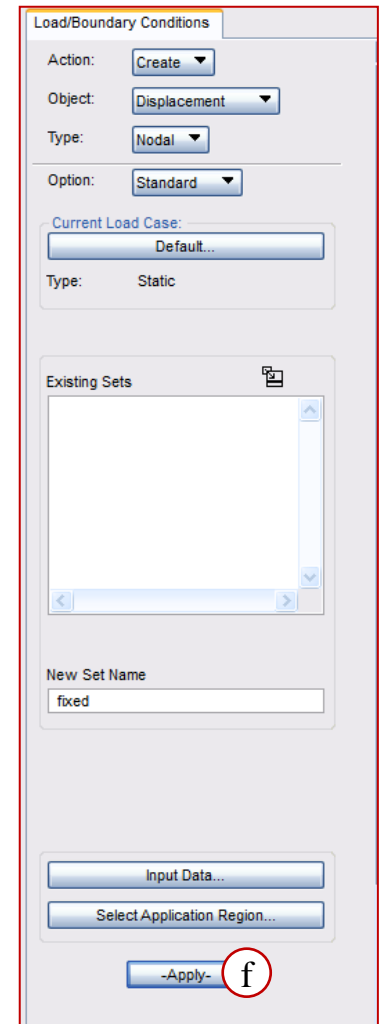
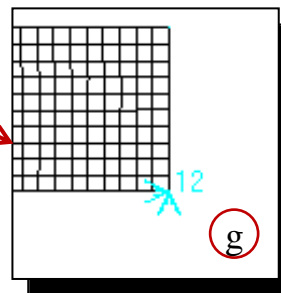
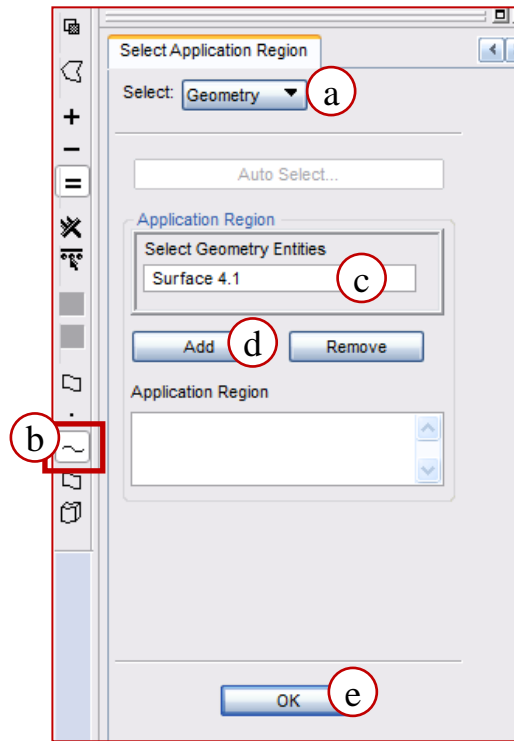
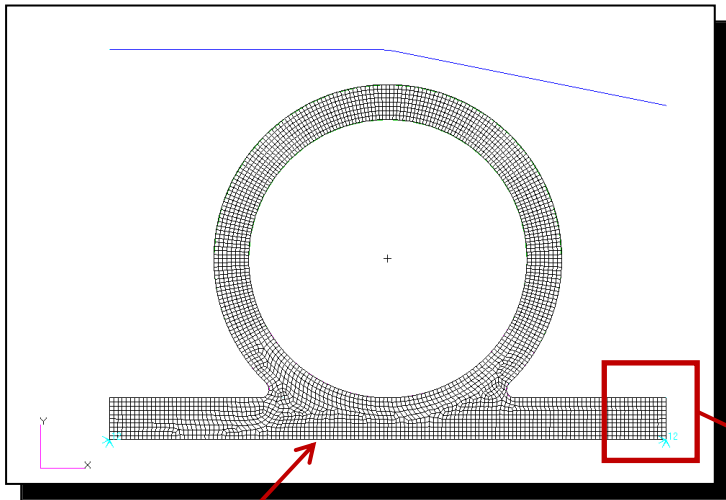
- Under the *Loads/BCs* tab, click **Displacement** in the *Nodal* group.
- Enter **fixed** as the *New Set Name*.
- Click **Input Data**.
- Enter **<0,0, >** for the *Translation*.
- Click **OK**.
- Click **Select Application Region**.



# Step 9. Create Fixed Boundary Condition (Cont.)

Create a fixed boundary condition on the bottom of the rubber seal continued.

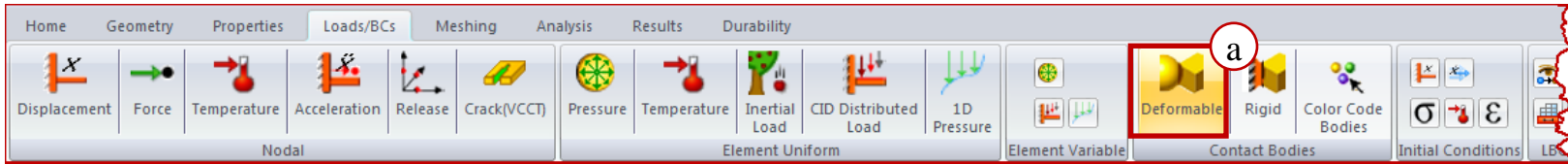
- Set *Select* to **Geometry**.
- Select **Curve or Edge** on the *Select Menu*.
- Click in the *Select Geometries Entities* text box and select the bottom edge of the seal.
- Click **Add**.
- Click **OK**.
- Click **Apply**.
- After the fixed boundary condition is applied, the model appears as shown.





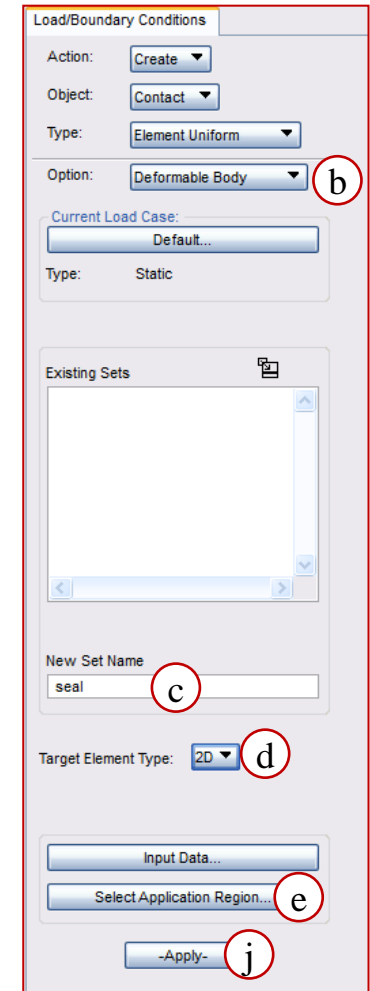
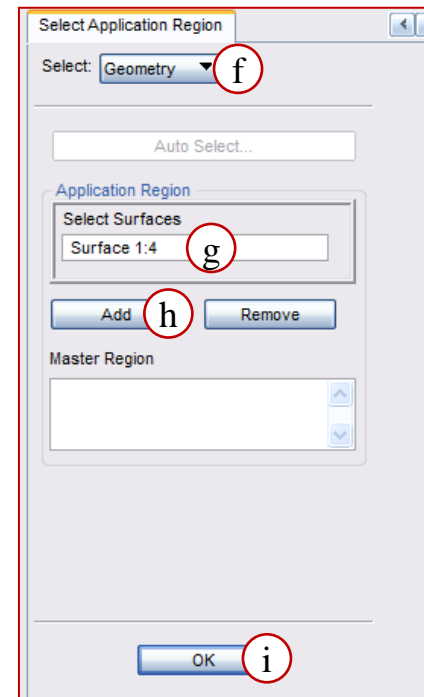
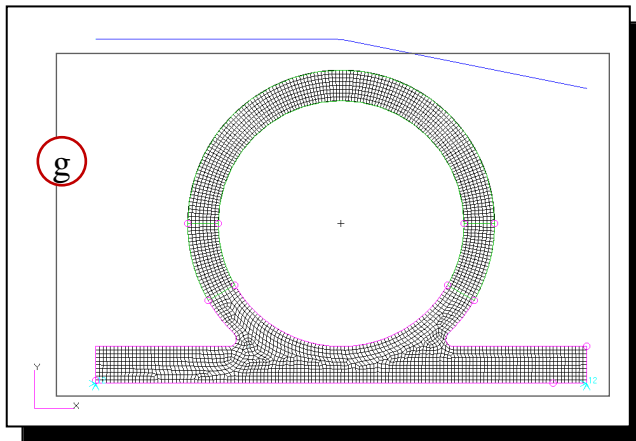


# Step 10. Define the Deformable Contact Body

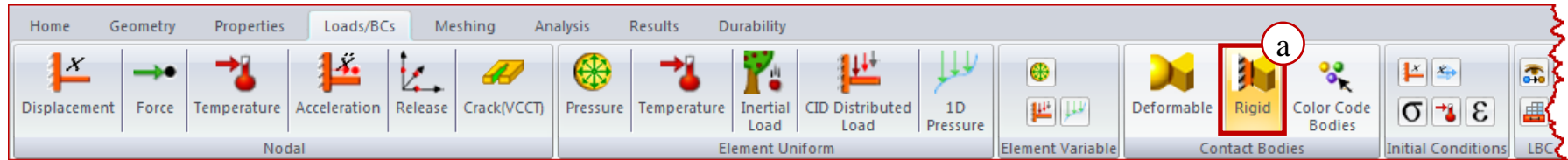


Define the seal as the deformable contact body:

- a. Click **Deformable** in the *Contact Bodies* group.
- b. Confirm **Deformable Body** as the *Option*.
- c. Enter **seal** as the *New Set Name*.
- d. Pull down **2D** as the *Target Element Type*.
- e. Click **Select Application Region**.
- f. Set *Select* to **Geometry**.
- g. Click in the *Select Surfaces* text box and select all four surfaces.
- h. Click **Add**.
- i. Click **OK**.
- j. Click **Apply**.

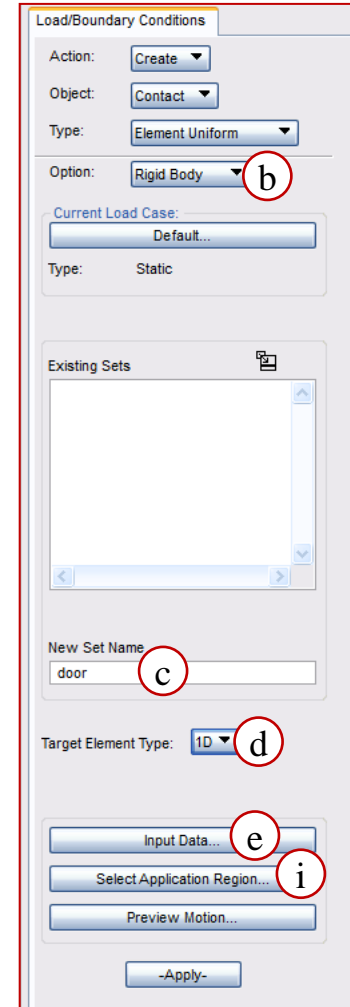
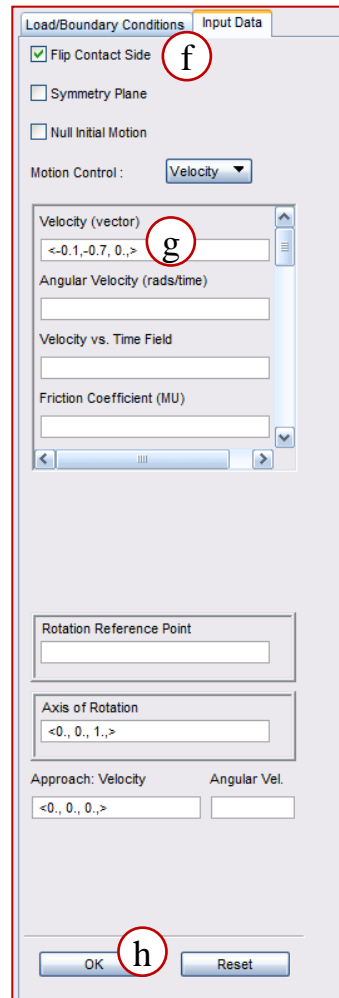


# Step 11. Define the Rigid Contact Body



Define the door as the rigid contact body.

- a. Click **Rigid** in the *Contact bodies* group.
- b. Confirm **Rigid Body** as the *Option*.
- c. Enter **door** as the *New Set Name*.
- d. Pull down **1D** as the *Target Element Type*.
- e. Click **Input Data**.
- f. Check **Flip Contact Side** (Flipping the Contact Side is needed, because of the orientations of the curves representing the door)
- g. Enter **<-0.1, -0.7, 0.>** for *Velocity*.
- h. Click **OK**.
- i. Click **Select Application Region**.

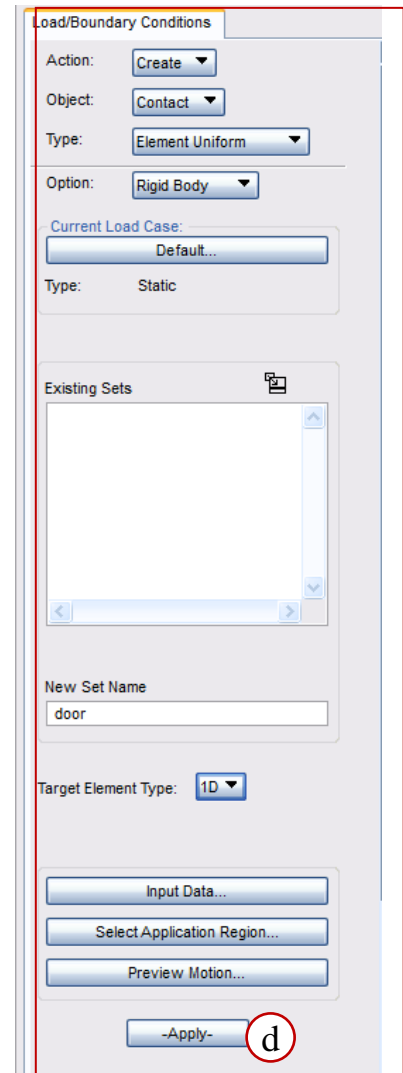
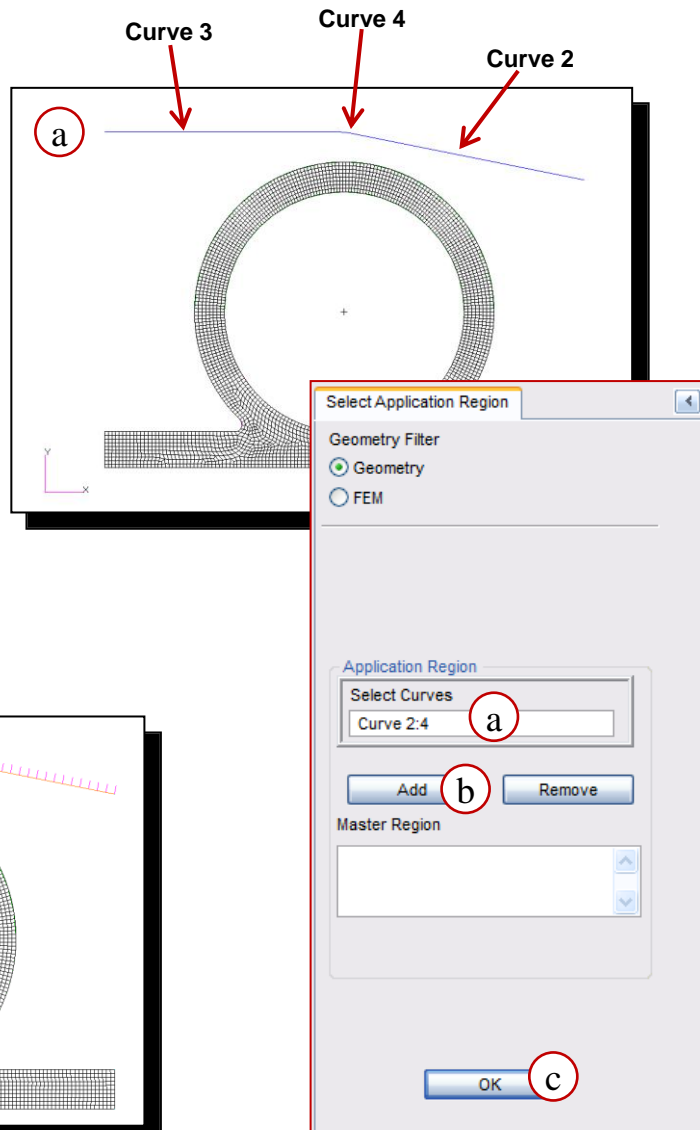


# Step 11. Define the Rigid Contact Body (Cont.)

Define the door as the rigid contact body continued.

- Click in the *Select Curves* text box and select all three curves defining the rigid surface (Curve 2:4).
- Click **Add**.
- Click **OK**.
- Click **Apply**.
- You will now see the contact tick marks (pink) along the three curves.

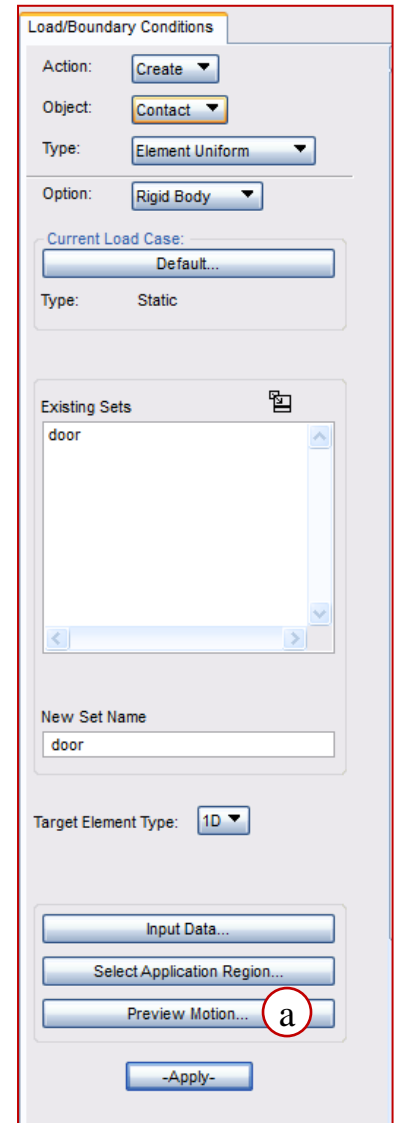
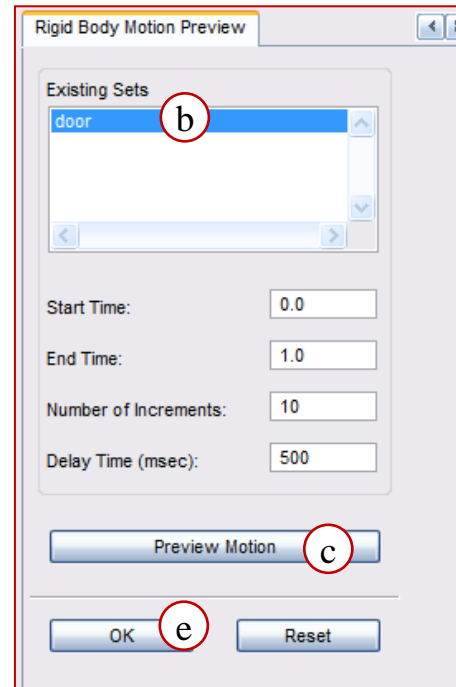
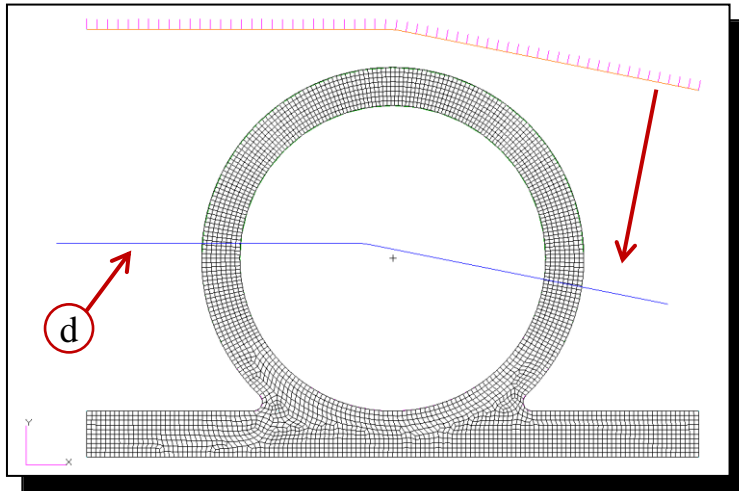
The rigid contact markers point toward the inside of the rigid body. Think of them as tick marks representing a wall. If the tick marks are reversed, Modify the Contact Body and flip the contact side on the Modify Data Form.



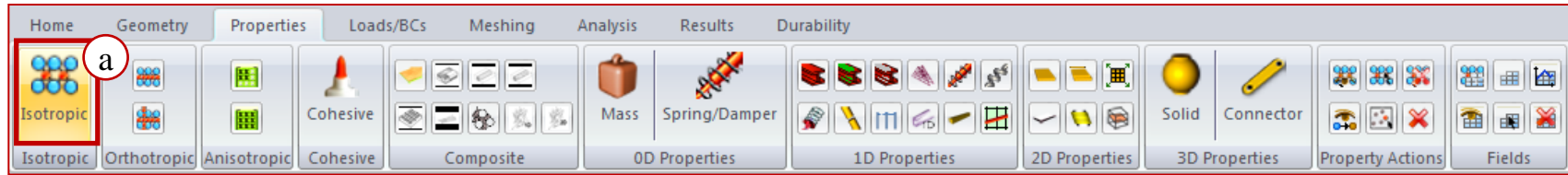
# Step 12. Preview the Rigid Body Motion

To preview the rigid body motion:

- a. Click **Preview Motion**.
- b. Under *Existing Sets*, select **door**.
- c. Click **Preview Motion**.
- d. The Rigid Body should move in the prescribed motion that was entered on the Input Data form. The ending position of the rigid body is shown below.
- e. Click **OK** when finished examining the motion.



# Step 13. Create the Rubber Material



To create the rubber material:

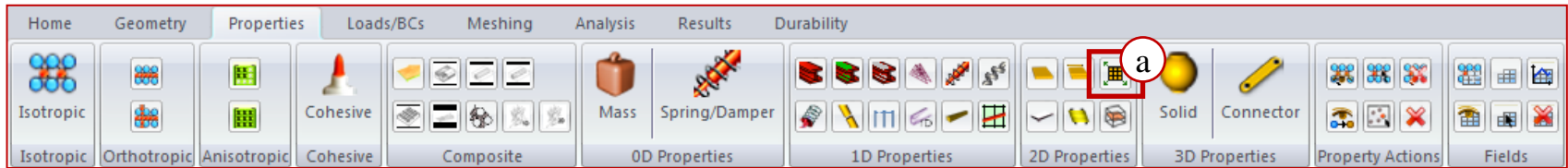
- Under the *Properties* tab, click **Isotropic** in the *Isotropic* group.
- Enter **rubber** as the *Material Name*.
- Click **Input Properties**.
- Pull down **Hyperelastic** for the *Constitutive Model*.
- Pull down **Mooney-Rivlin** for the *Model*.
- Enter **80** as the *Strain Energy Function, C10*.
- Enter **20** as the *Strain Energy Function, C01*.
- Click **OK**.
- Click **Apply**.

The material's constitutive model used in this analysis is an Incompressible Mooney-Rivlin hyperelastic formulation. Make sure that the analysis code is set for MSC Marc under references-Analysis.

The 'Input Options' dialog box is shown. It has fields for Constitutive Model (Hyperelastic, labeled 'd'), Model (Mooney-Rivlin, labeled 'e'), Domain Type (Time), and Number of Terms (1). Below these are input fields for Property Name and Value. The Strain Energy Function, C10 is set to 80 (labeled 'f'), and the Strain Energy Function, C01 is set to 20 (labeled 'g'). Other fields include Density, Thermal Expansion Coeff, Bulk Modulus, and Reference Temperature. At the bottom, there are buttons for OK (labeled 'h'), Clear, and Cancel.

The 'Materials' dialog box is shown. It has fields for Action (Create), Object (Isotropic), and Method (Manual Input). Below these is a list of Existing Materials. The Material Name is set to rubber (labeled 'b'). At the bottom, there are buttons for Input Properties (labeled 'c'), Change Material Status, and Apply (labeled 'i').

# Step 14. Define the Element Properties

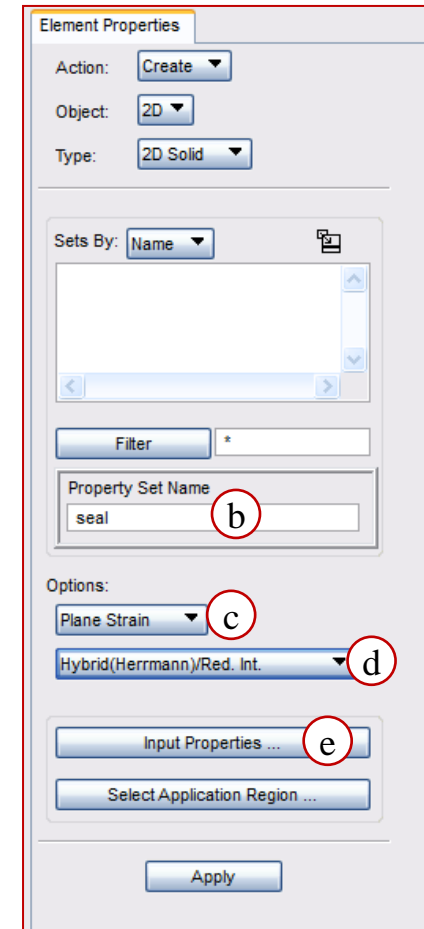
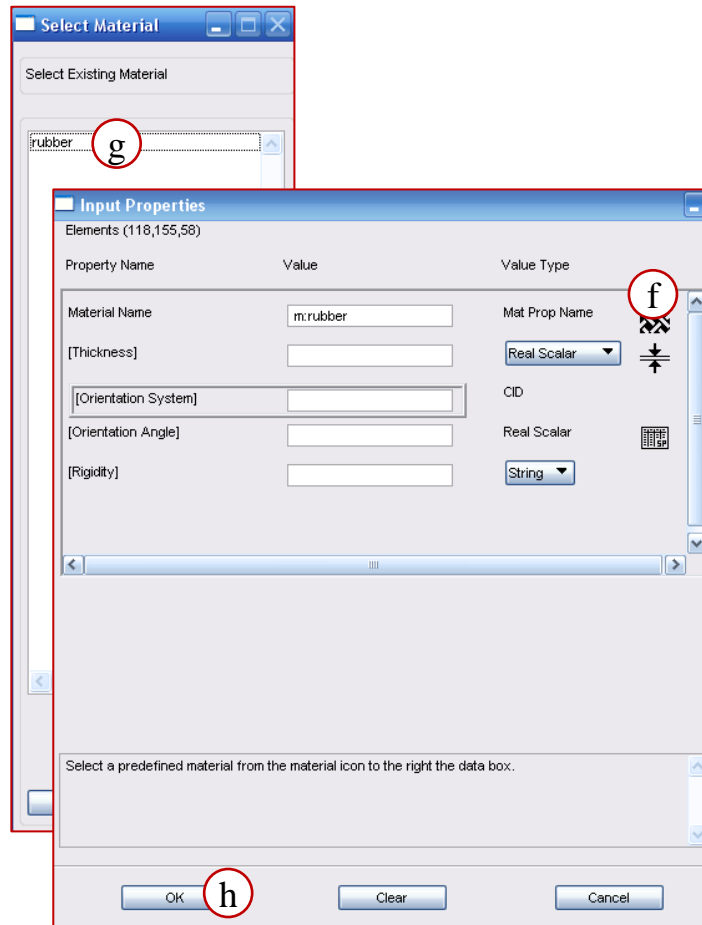


To define the element properties:

- Click **2D Solid** in the *2D Properties* group.
- Enter **Seal** as the *Property Set Name*.
- Pull down **Plane Strain** as the *Option*.
- Pull down **Hybrid(Herrmann)/Red. Int.** as the option.
- Click **Input Properties**.
- Click on the **Mat Prop Name** icon.
- Choose **rubber** under the *Select Existing Material*.
- Click **OK**.

In this step, you will be defining the element properties for the rubber seal. The seal will be modeled using a 2-D Solid (Plane Strain) Herrmann element/reduced integration formulation. The rubber material will be assigned to this property.

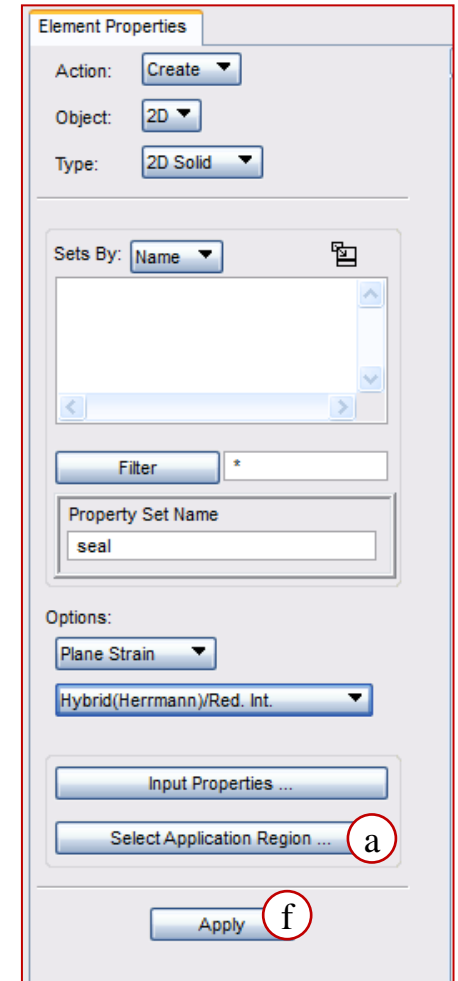
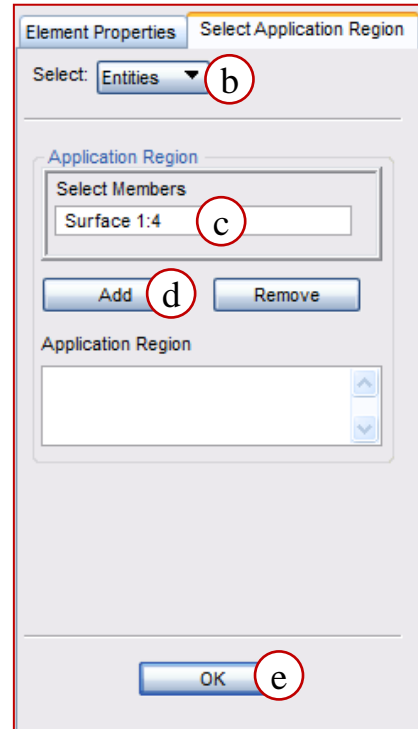
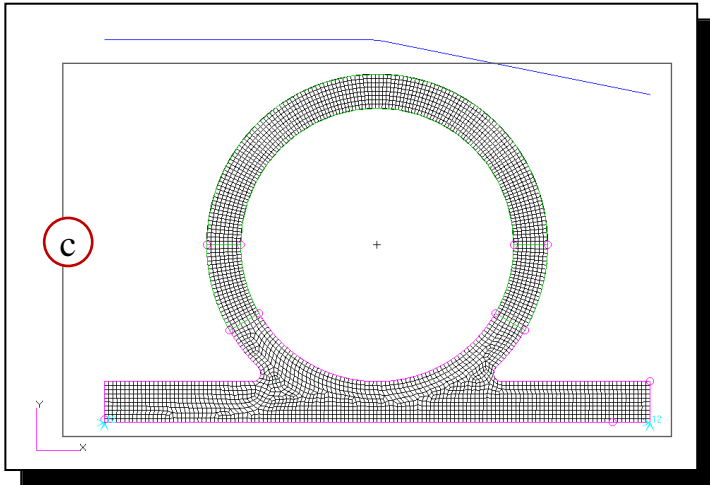
**Incompressible hyperelastic materials require the use of the Herrmann formulation.**



# Step 14. Define the Element Properties (Cont.)

Define the element properties continued.

- a. Click **Select Application Region**.
- b. Set *Select* to **Entities**.
- c. Click in the *Select Members* text box and select all four surfaces.
- d. Click **Add**.
- e. Click **OK**.
- f. Click **Apply**.

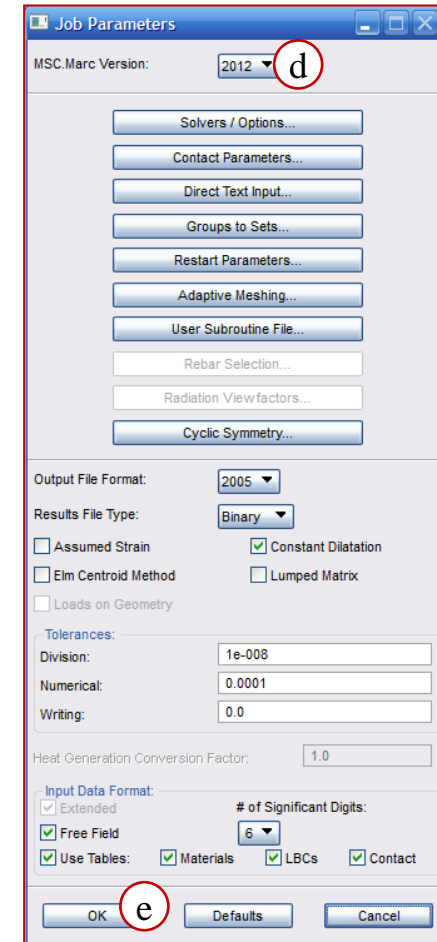
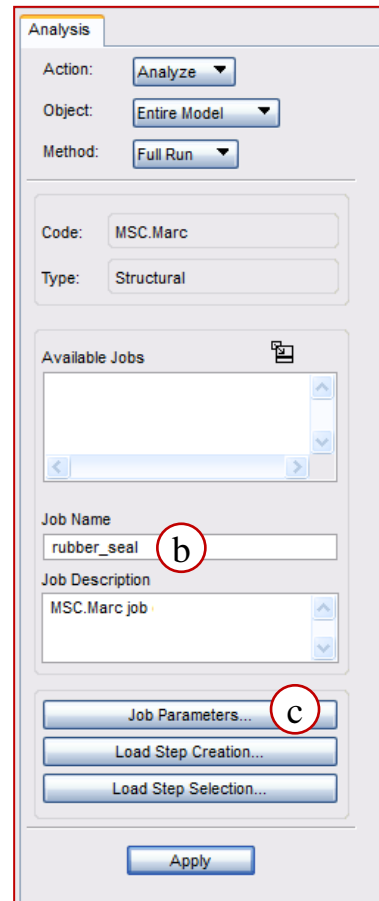


# Step 15. Set Job Parameters



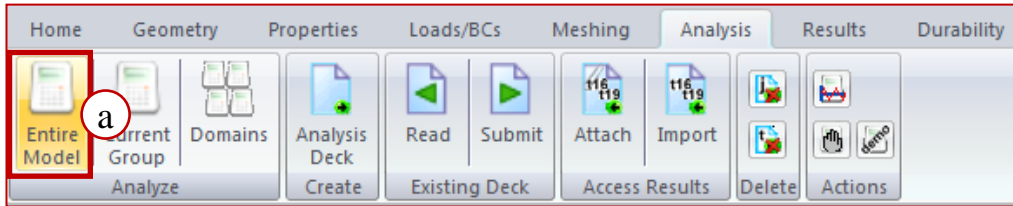
Set job parameter to MSC Marc Version 2012.

- Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
- Enter **rubber\_Seal** as the *Job Name*.
- Click **Job Parameters**.
- Pull down *MSC.Marc Version* to **2012**.
- Click **OK**.



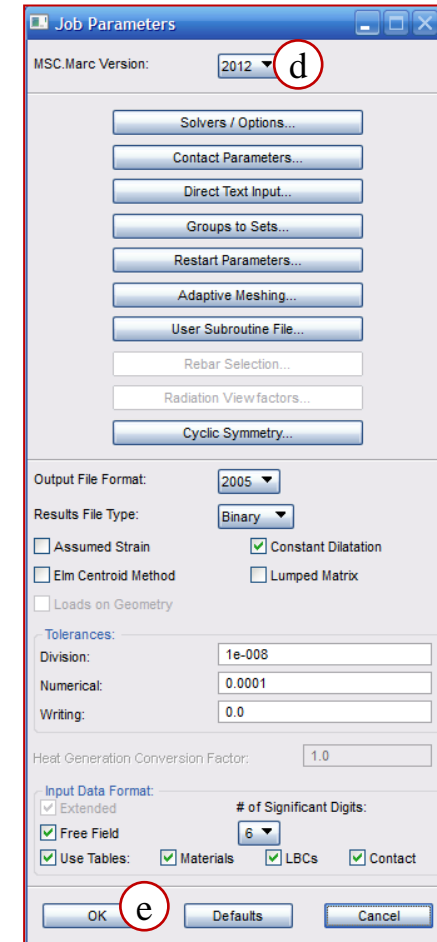
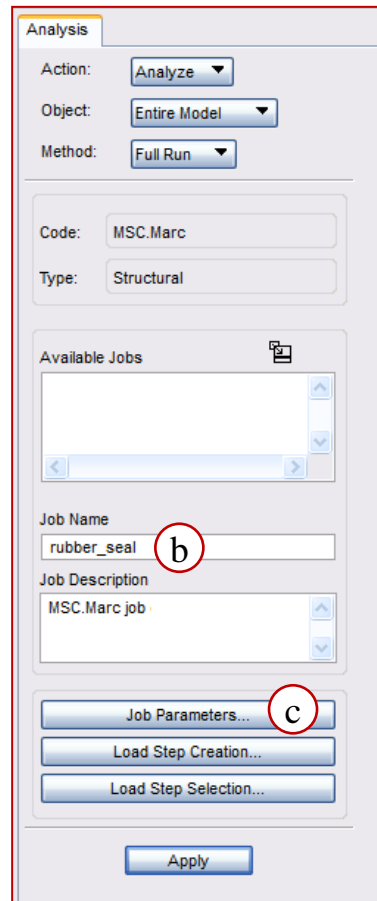


# Step 15. Set Job Parameters



Set job parameter to MSC Marc Version 2012.

- a. Under the *Analysis* tab, click **Entire Model** in the *Analyze* group.
- b. Enter **rubber\_Seal** as the *Job Name*.
- c. Click **Job Parameters**.
- d. Pull down *MSC.Marc Version* to **2012**.
- e. Click **OK**.

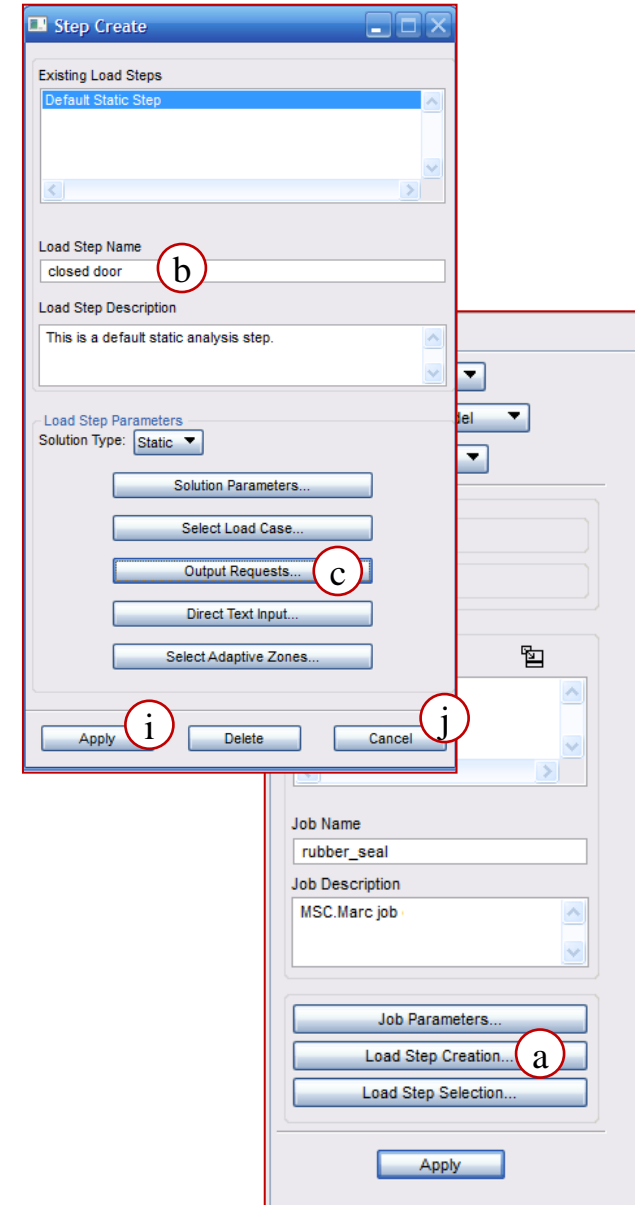
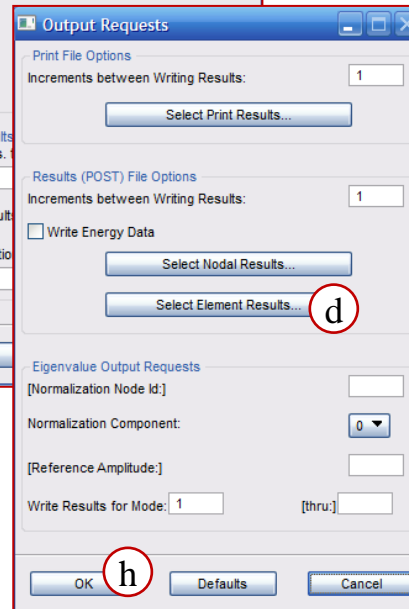
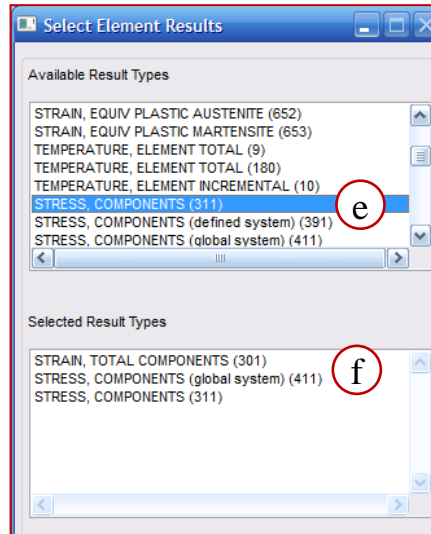


# Step 16. Create a Load Step

Create a load step called *closed door* and define output requests.

- a. Click **Load Step Creation**.
- b. Enter **close door** as the *Load Step Name*.
- c. Click **Output Requests**.
- d. Click **Select Element Results**.
- e. Select **Stress Components (311)** under *Available Request Types*.
- f. Notice that **Stress, Components (global system) (411)** and **Strain, Total components (301)** already appear under the *Select Result Types* list.
- g. Click **OK**.
- h. Click **OK**.
- i. Click **Apply**.
- j. Click **Cancel**.

The default Stress quantity that will be written to the output file will not work with the Herrmann Elements for Rubber Materials. We have to switch the stress result quantity with one that will give us stress results to look at during post-processing. If we keep the default settings, the stress results will all be equal to zero during post-processing.



# Step 17. Select Load Case and Run the Analysis

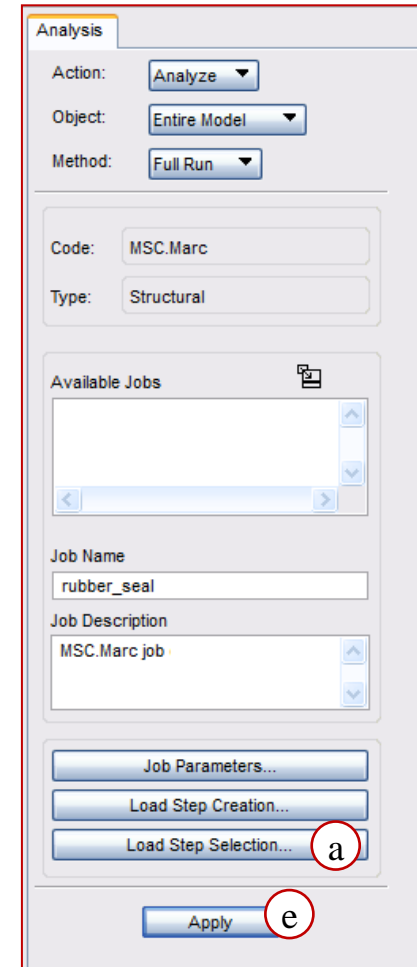
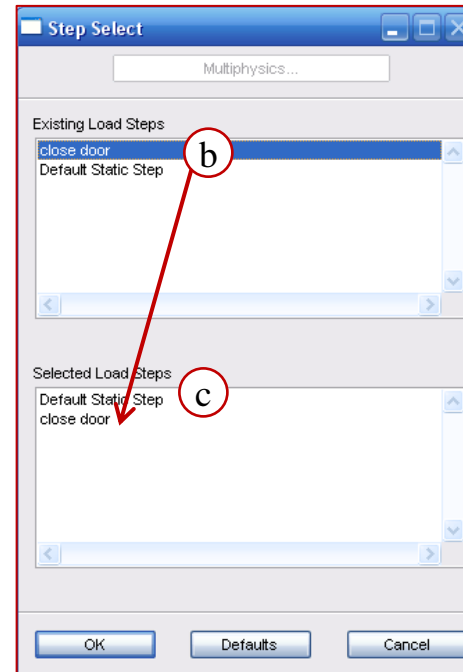
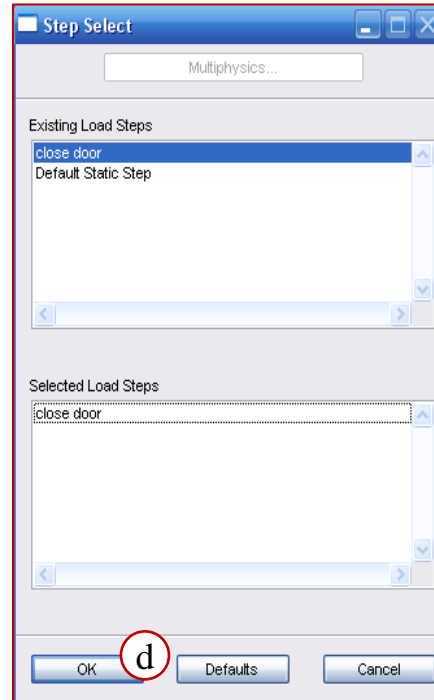
To select the load case.

- a. Click **Load Step Selection**.
- b. Select **close door** under *Existing Load Steps*. It will be added to the *Selected Load Steps* list below.
- c. Deselect **Default Static Step** by clicking on it under the *Selected Load Steps* list. The *Default Static Step* will be removed from the *Selected Load Step* list.
- d. Click **OK**.

To run the analysis using Marc.

- e. Click **Apply**.

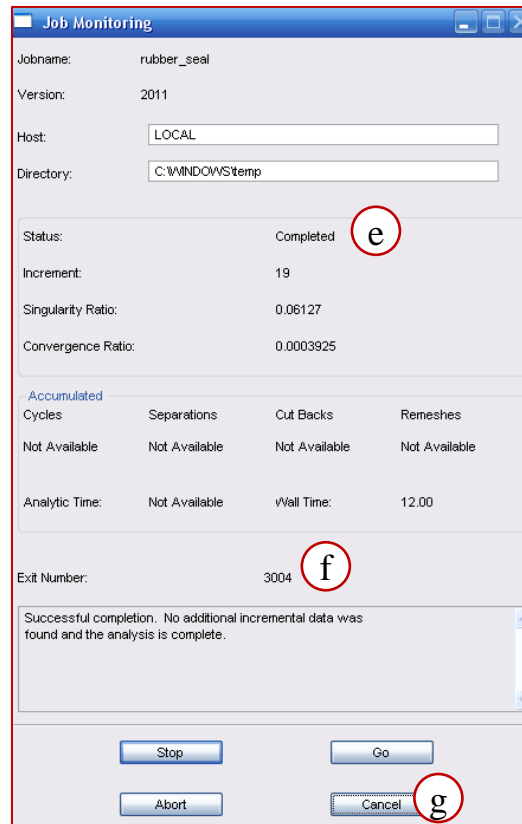
The **Step Select** form should look like this when you close it.



# Step 18. Monitor the Job

Monitor the linear job:

- a. Change the *Action* to **Monitor**.
- b. Change the *Object* to **Job**.
- c. Select **rubber\_seal** under *Available Jobs*.
- d. Click **Apply**.
- e. The Job Monitoring form will automatically appear. The *status* will denote **Complete** when the job is finished.
- f. For a job with no errors the *Exit Number* should read **3004**.
- g. Click **Cancel** when the job is complete.



The Job Monitoring dialog box displays the following information:

Jobname: rubber\_seal  
Version: 2011  
Host: LOCAL  
Directory: C:\WINDOWS\temp

Status: Completed (e)  
Increment: 19  
Singularity Ratio: 0.06127  
Convergence Ratio: 0.0003925

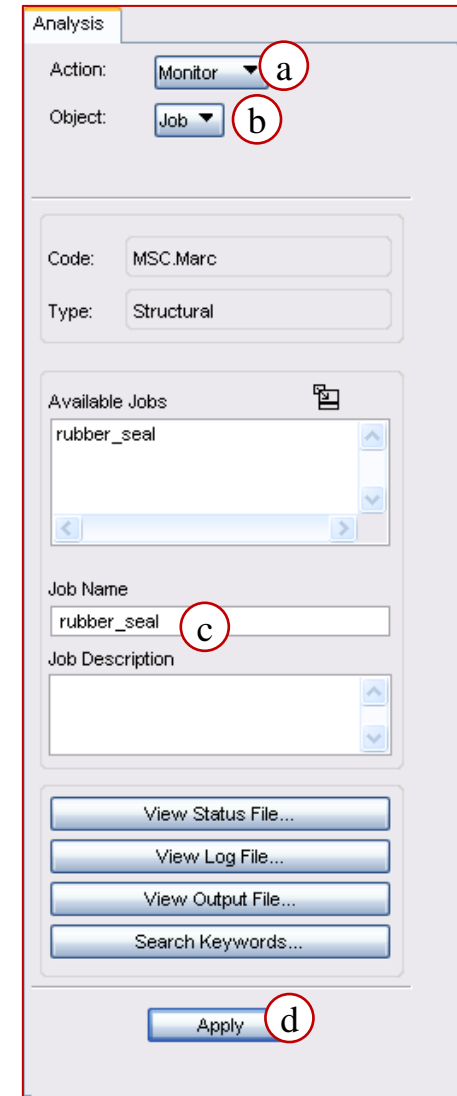
Accumulated			
Cycles	Separations	Cut Backs	Remeshes
Not Available	Not Available	Not Available	Not Available

Analytic Time: Not Available    Wall Time: 12.00

Exit Number: 3004 (f)

Successful completion. No additional incremental data was found and the analysis is complete.

Buttons: Stop, Go, Abort, Cancel (g)



The Analysis dialog box displays the following information:

Action: Monitor (a)  
Object: Job (b)

Code: MSC.Marc  
Type: Structural

Available Jobs: rubber\_seal (c)

Job Name: rubber\_seal (c)  
Job Description:

Buttons: View Status File..., View Log File..., View Output File..., Search Keywords..., Apply (d)

# Step 18. Monitor the Job (Cont.)

Monitor the job continued.

- Click **View Status File...**
- Close the *rubber\_seals.sts* Status File Window when done reviewing it.

**rubber\_seal.sts - Notepad**

File Edit Format View Help

case #	inc #	cycl #	sepa #	cut #	cycl #	separ #	cut #	rmesh #	time of the inc	total time of the job	displ-mag min	displ-mag max
---of the inc--- -----of the analysis-----												
0	0	0	0	0	0	0	0	0	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
1	1	1	0	0	1	0	0	0	1.0000E-02	1.0000E-02	0.0000E+00	0.0000E+00
1	2	1	0	0	2	0	0	0	1.2000E-02	2.2000E-02	0.0000E+00	0.0000E+00
1	3	1	0	0	3	0	0	0	1.4400E-02	3.6400E-02	0.0000E+00	0.0000E+00
1	4	1	0	0	4	0	0	0	1.7280E-02	5.3680E-02	0.0000E+00	0.0000E+00
1	5	1	0	0	5	0	0	0	2.0736E-02	7.4416E-02	0.0000E+00	0.0000E+00
1	6	1	0	0	6	0	0	0	2.4883E-02	9.9299E-02	0.0000E+00	0.0000E+00
1	7	1	0	0	7	0	0	0	2.9860E-02	1.2916E-01	0.0000E+00	0.0000E+00
1	8	2	0	0	9	0	0	0	3.5832E-02	1.6499E-01	0.0000E+00	7.2565E-03
1	9	2	0	0	11	0	0	0	4.2998E-02	2.0799E-01	0.0000E+00	4.4692E-02
1	10	2	0	0	13	0	0	0	5.1598E-02	2.5959E-01	0.0000E+00	9.1371E-02
1	11	3	1	0	16	1	0	0	6.1917E-02	3.2150E-01	0.0000E+00	1.3658E-01
1	12	4	2	0	20	3	0	0	7.4301E-02	3.9581E-01	0.0000E+00	1.9924E-01
1	13	7	4	0	27	7	0	0	8.9161E-02	4.8497E-01	0.0000E+00	2.9146E-01
1	14	7	3	0	34	10	0	0	8.9161E-02	5.7413E-01	0.0000E+00	4.0267E-01
1	15	8	4	0	42	14	0	0	8.9161E-02	6.6329E-01	0.0000E+00	5.1850E-01
1	16	8	4	0	50	18	0	0	8.9161E-02	7.5245E-01	0.0000E+00	6.2905E-01
1	17	8	4	0	58	22	0	0	8.9161E-02	8.4161E-01	0.0000E+00	7.3381E-01
1	18	8	4	0	66	26	0	0	8.9161E-02	9.3077E-01	0.0000E+00	8.3368E-01
1	19	11	4	0	77	30	0	0	6.9229E-02	1.0000E+00	0.0000E+00	9.0793E-01

Job ends with exit number : 3004  
total wall time: 28.25

Analysis

Action: **Monitor**

Object: **Job**

Code: MSC.Marc

Type: Structural

Available Jobs

rubber\_seal

Job Name

rubber\_seal

Job Description

**View Status File...**

View Log File...

View Output File...

Search Keywords...

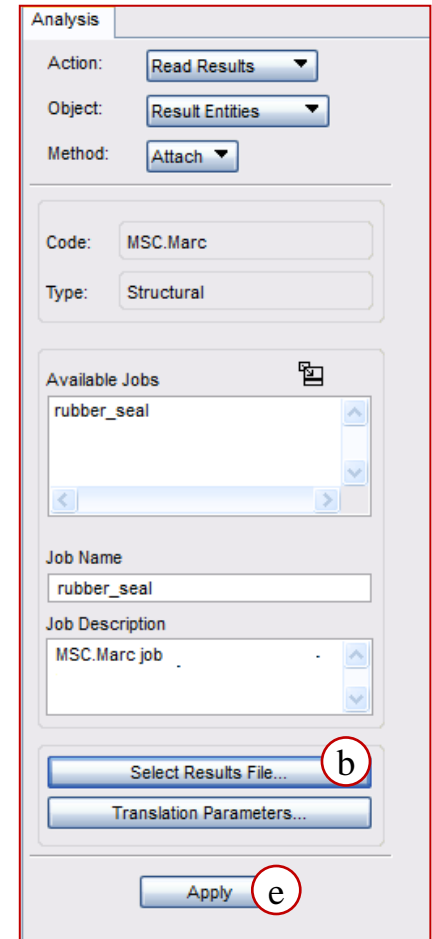
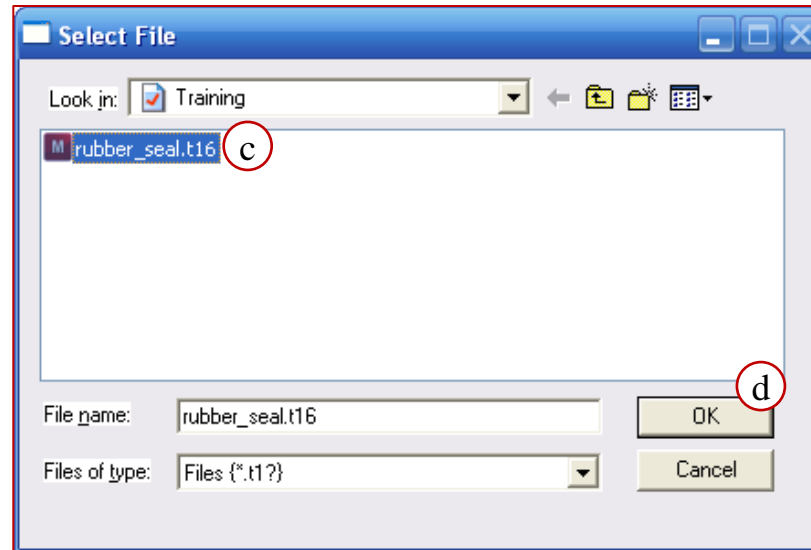
Apply

# Step 19. Attach the Results File

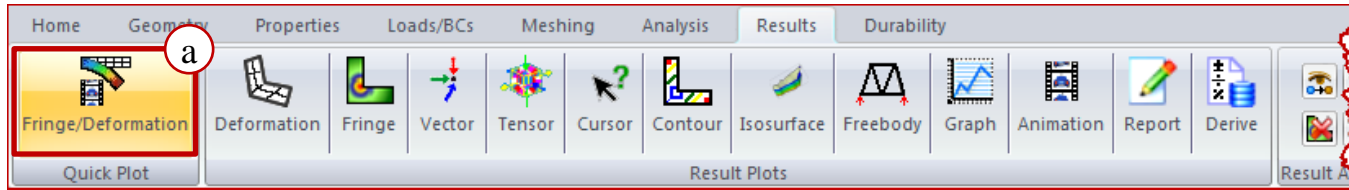


Attach the Marc results file *rubber\_seal.t16*.

- a. Click **Attach** in the Access Results group.
- b. Click **Select Results File**.
- c. Select the file **rubber\_seal.t16**
- d. Click **OK**.
- e. Click **Apply**.

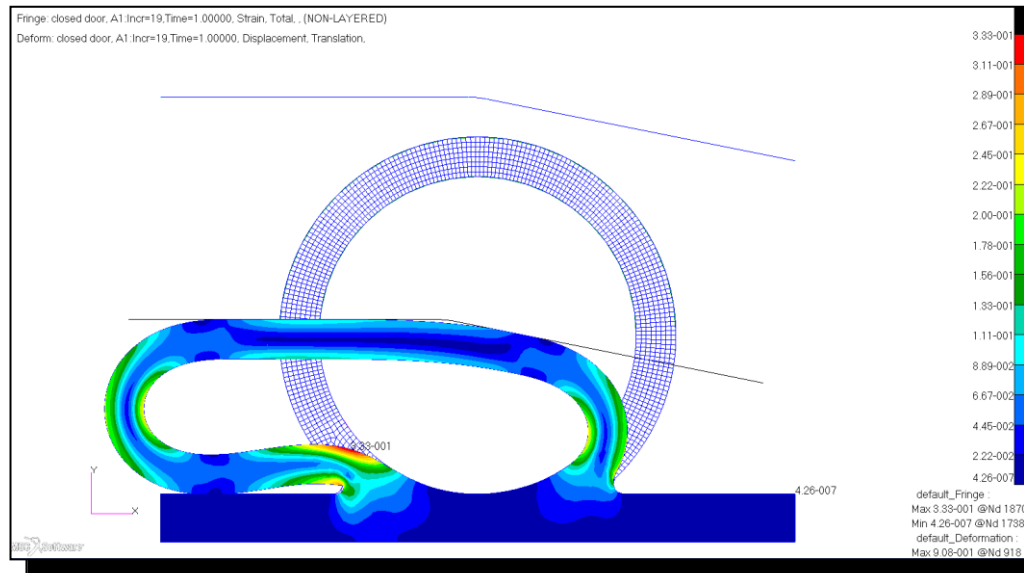


# Step 20. Plot the Strain Results

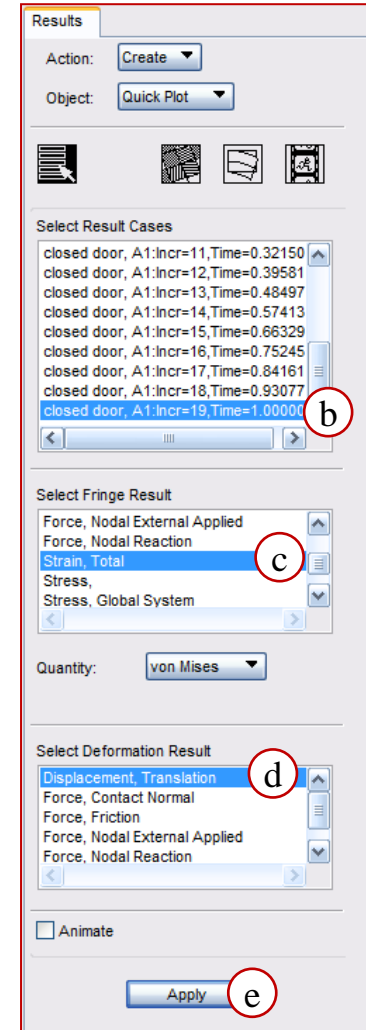


Plot the strain results.

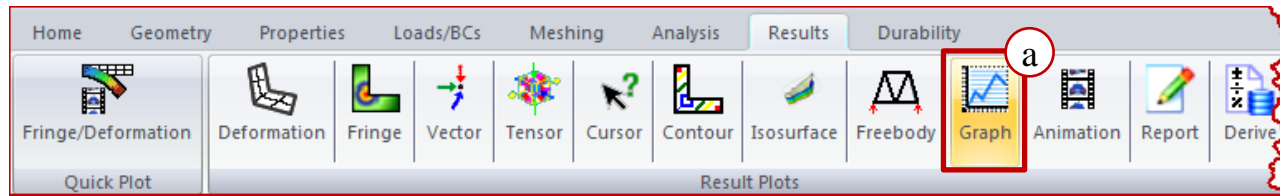
- Under the *Results* tab, click **Fringe/Deformation** in the *Quick plot* group.
- Select the last result case from the *Select Result Cases* list.
- Select **Strain, Total** as the *Select Fringe Result*.
- Select **Displacement, Translation** as the *Select Deformation Result*.
- Click **Apply**.



Note: When post-processing nonlinear Results, a quick way to see an animation of the increments is to select all of the result cases in the Quick Plot form. Patran will plot the desired results of each load case with a few second delay between each one

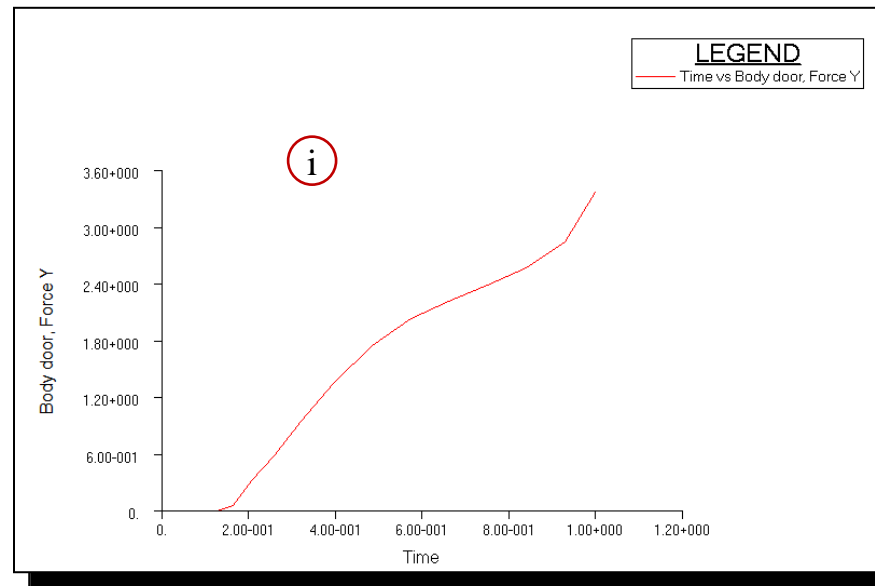


# Step 21. Create a Graph of Load vs. Displacement

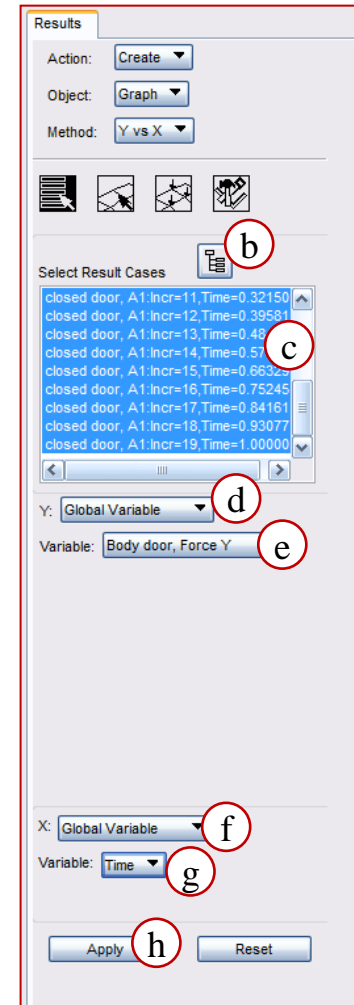


Create a graph of Load vs. Displacement:

- a. Click **Graph** in the *Result Plots* group.
- b. Click the **Results Tree** icon to view all of the result cases.
- c. Select all of the result cases.
- d. Pull down **Global Variable** for Y.
- e. Pull down **Body door, Force Y** for *Variable*.
- f. Pull down **Global Variable** for X.
- g. Pull down **Time** for *Variable*.
- h. Click **Apply**.
- i. The plot shows the total body force required to compress the rubber seal.



One nice feature of Marc rigid bodies is that they automatically sum all nodal loads from the interaction with the deformable body. As a result, we can obtain the total force required to compress the seal without having to manually sum up the individual nodal reaction loads.





# Step 22. Quit Patran

