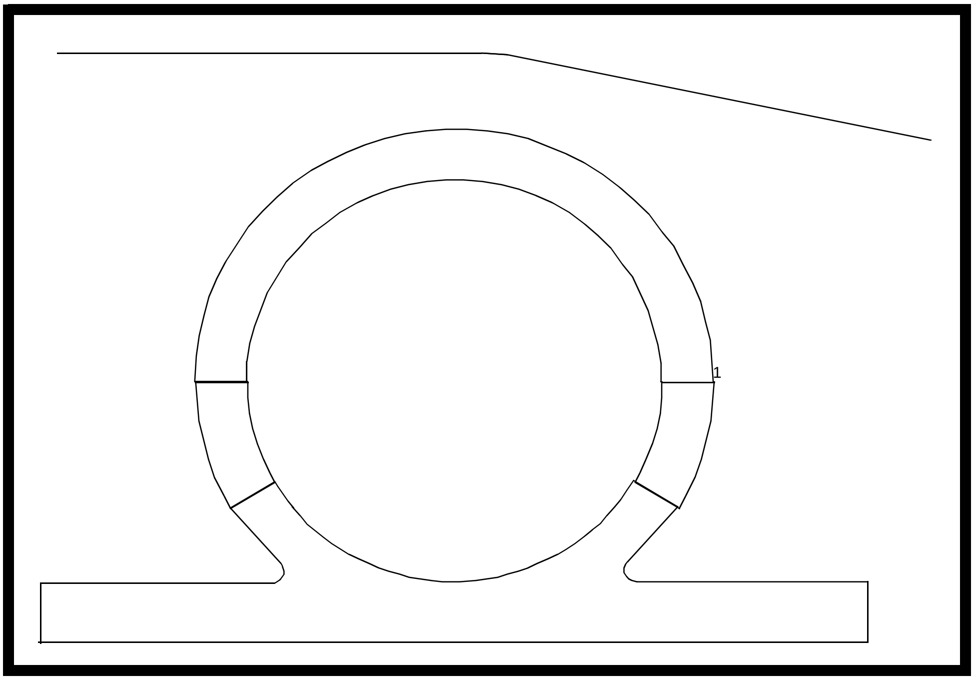
LESSON 13

Analysis of a Rubber Seal



Objectives:

- Large displacement analysis.
- Contact analysis using a rigid body contact model.
- Hyper-elastic material model.

Model Description:

In this Exercise we analyze a trunk 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 trunk door is considered very stiff relative to the rubber seal and can be modeled using a rigid body.

Suggested Exercise Steps:

- Build the seal geometry and mesh from a session file.
- Model the contact surfaces with LBC contact.
- Create the element properties.
- Create the Loads and BCs.
- Submit the job to analysis.
- Evaluate the results.

Exercise Procedure:

1. Open a new database. Name it **rubber_seal.db**.

File/New ...

Database Name:	**rubber_seal.db**
OK	
Analysis Code:	**MSC.Marc**
OK	

2. Read in the session file.

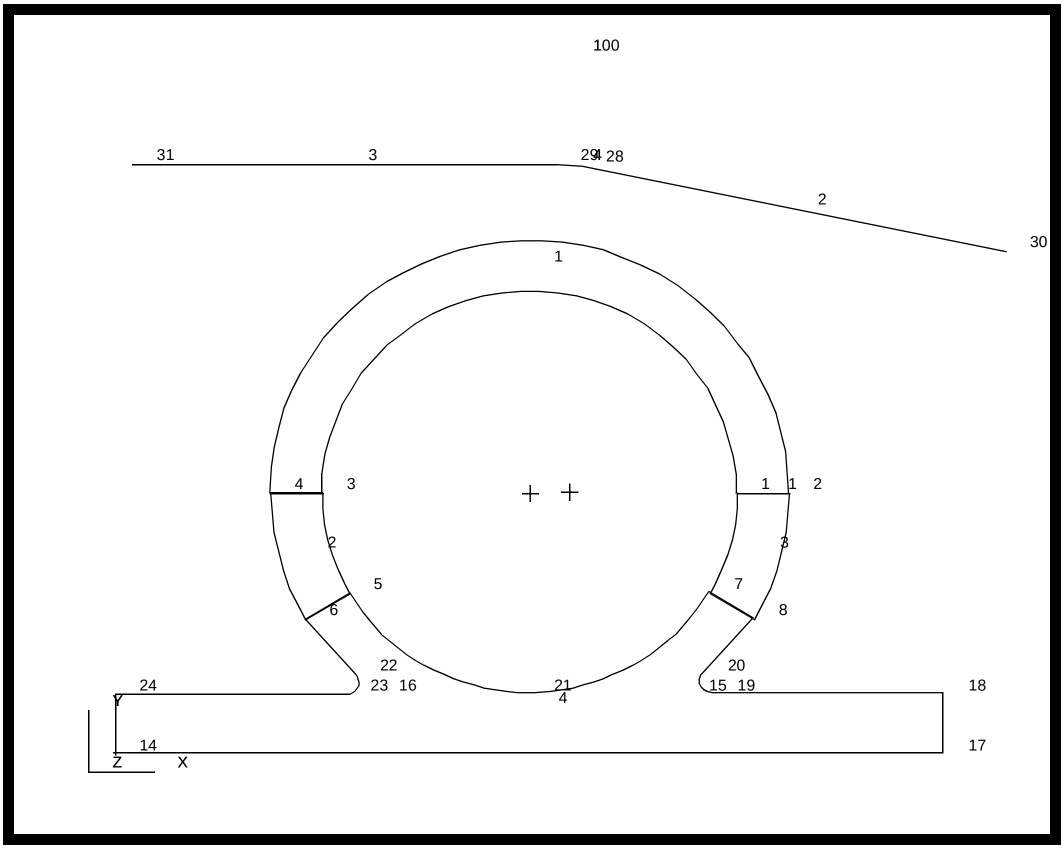
There is a session file that will create the geometry for this exercise

File/Session/Play...

Session File List:	**rubber_seal.ses**
Apply	

When the session file is done the viewport will contain all the geometry for the rubber seal and trunk rigid body. Additionally, 2 groups have been created, one containing the seal and the other containing the trunk.

Figure 13.1 - Geometry created by session file



Finally, post only the geometry for the seal.

Group/Post...

Groups to Post:	**trunk**

Apply

Cancel

3. Create a reference node for the rigid body.

■ **Elements**

Action:	**Create**
Object:	**Node**
Method:	**Edit**

■ **Associate With Geometry**

Node Location List:	see Figure 13.2

Apply

Now create the mesh seed for the rigid body.

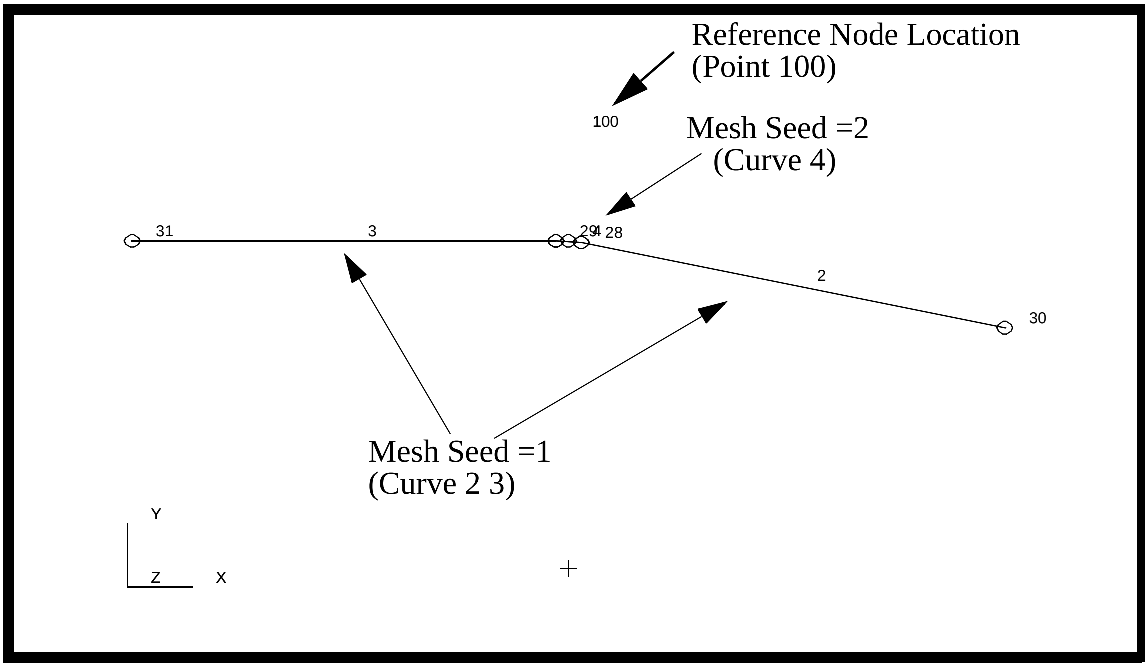
Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

■ **Number of Elements**

Number:	**1**
Curve List:	see Figure 9.2

Repeat this process with the *number* set to **2** and **Curve 4** in the *Curve List Box*. Your screen should appear like Figure 13.2:

Figure 13.2 - Element and node creation definition for trunk



Mesh the bar

Action:	**Create**

Object:	**Mesh**
Type:	**Curve**
Curve List:	select all posted curves

Apply

Equivalence the bar

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**

Apply

4. Post the **seal** group for meshing.

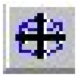
Group/Post...

Groups to Post:	**seal**

Apply

Cancel

Fit the model on the screen using the following toolbar icon:



Fit View

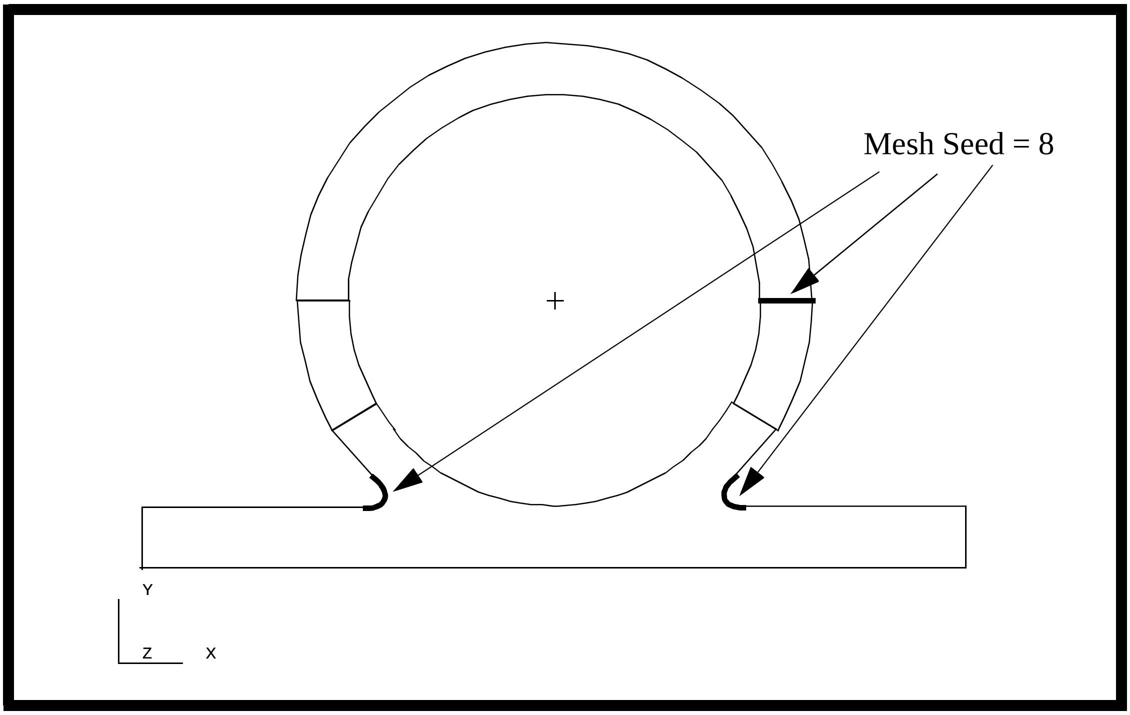
5. Create the mesh seed on the seal

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

■ **Number of Elements**

Number:	**8**
Curve List:	see Figure 13.3

Figure 13.3 - Mesh seed locations for seal



Mesh and Equivalence the seal, starting with the top.

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length:	**.015**
Mesher:	◆ **Isomesh**
Surface List:	select all green surfaces

Apply

Mesh the botom of the seal

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length:	**.015**
Mesher:	◆ **Paver**

Surface List: select magenta surface

Apply

6. Equivalence any duplicate nodes created during meshing.

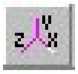
Action: **Equivalence**

Object: **All**

Method: **Tolerance Cube**

Apply

Change to **Iso 1 View** using this icon.



Iso 1 View

7. Since this is a 2-D solid model, all element normals must point in the positive Z direction. Verify the elements' normals, and correct those whose normals point the wrong direction.

Action: **Verify**

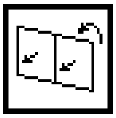
Object: **Element**

Test: **Normals**

Display Controls: ● **Draw Normal Vectors**

Apply

Test Control:



Reverse Elements

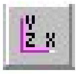
Guiding Element: Select an element pointing in the positive Z direction

Apply

Reset Graphics

 **Reset Graphics**

Change back to the **Front View**

 **Front View**

8. Define the rubber material.

The material constituitive model used in this analysis is an imcompressible Mooney Rivlin hyperelastic formulation.

■ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**rubber**

Input Properties...

Constitutive Model:	**Hyperelastic**
Model:	**Mooney Rivlin**
Strain Energy Function, C10:	**80**
Strain Energy Function, C01:	**20**

OK

Apply

9. Define the element properties.

In this step, you will be defining the element properties for the seal. The seal will be modeled using a 2-D Solid (Plane Strain) Hermann/ Reduced Integration element formulation. The rubber material will be assigned to this property. It should be noted, anytime a hyperelastic material is defined, it is required that it is used in conjunction with the Hermann element formulation.

■ **Properties**

Action:	**Create**
Dimension:	**2D**

Type:	**2D Solid**
Property Set Name:	**seal**
Options:	**◆ Plane Strain**
	◆ Hermann/Reduced Integration

Input Properties...

Material Name:	**rubber**
Thickness:	**1.0**

OK

Select Members	select all surfaces displayed

Add

Apply

Now that the modeling of the seal is complete, we need to model the contact surfaces.

10. Define the trunk door to seal contact *Load and Boundary Condition.*

For this model, we will assume the car door is perfectly rigid relative to the stiffness of the seal. The perfectly rigid surface is modeled using a contact load and boundry condition.

When you played the session file earlier, it created the geometry for the trunk door and seal and placed those entities into two seperate groups. You will create a group that contains all the entities.

Group/Create...

New Group Name:	**all**

■ Make Current

■ Unpost All Other Groups

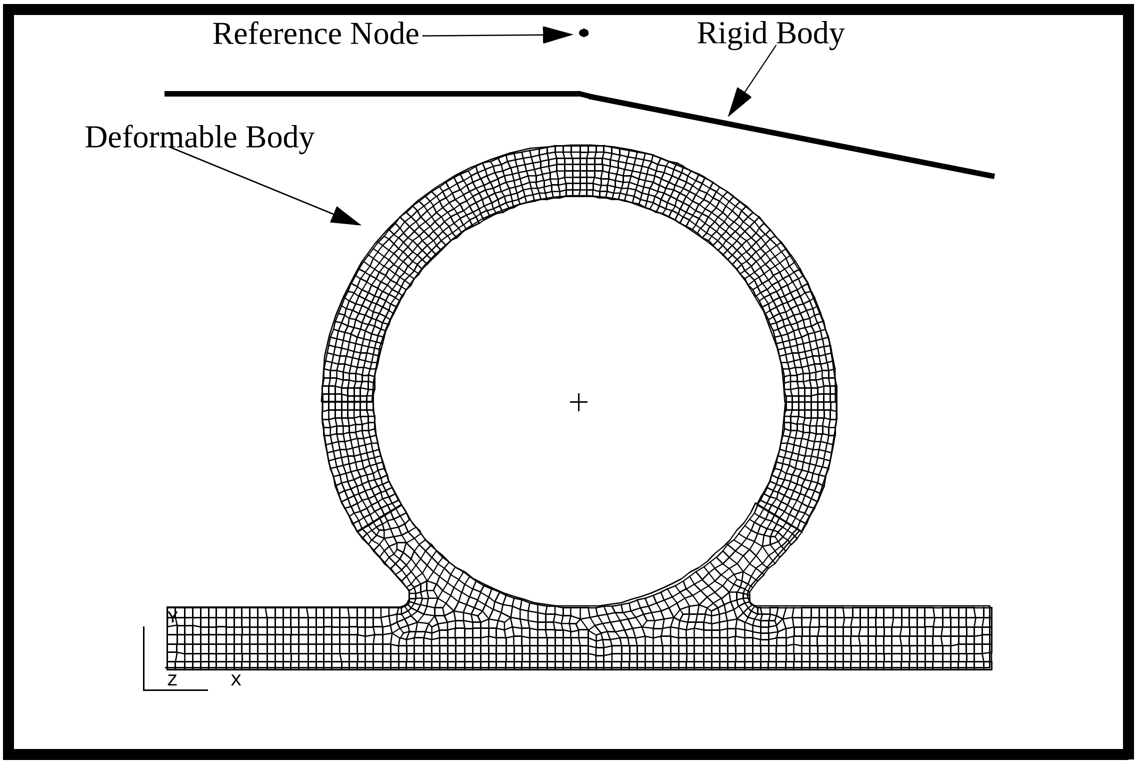
Group Contents:	**Add All Entities**

Apply

Cancel

Definition of the seal contact edges is critical. If the seal edges that come into contact with the door are incorrectly identified, the door will pass through the seal. This can be easily recognized in the post-processing phase, after the analysis. The model would then need to be adjusted and re-run until all contacting surfaces are correctly identified. See Figure 13.4 :

Figure 13.4 - Contact definitions for trunk and seal



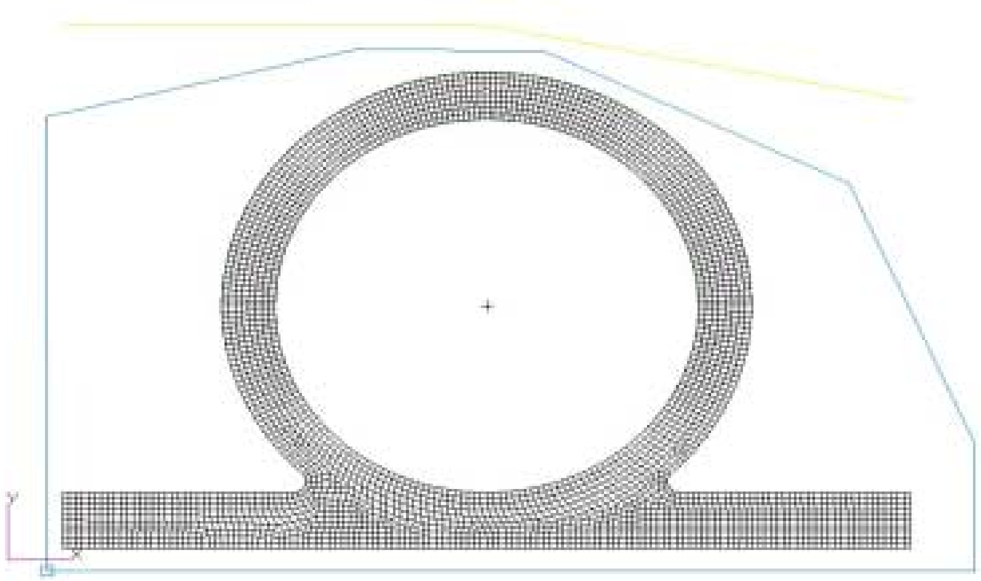
First define the deformable contact body.

■ **Load/BCs**

Action:	**Create**
Object:	**Contact**
Type:	**Element Uniform**
Option:	**Deformable Body**
New Set Name:	**contact_deform**
Target Element Type:	**2D**

Select Application Region...

Figure 13.5 - Contact definition for deformable seal contact



Geometry Filter:	● **Geometry**
Select Surfaces:	Pick area using <control> select to form a polygon (see Figure 13.5)

Add

OK

Apply

Now define the rigid contact body.

Action:	**Create**
Object:	**Contact**
Type:	**Element Uniform**
Option:	**Rigid Body**
New Set Name:	**contact_rigid**

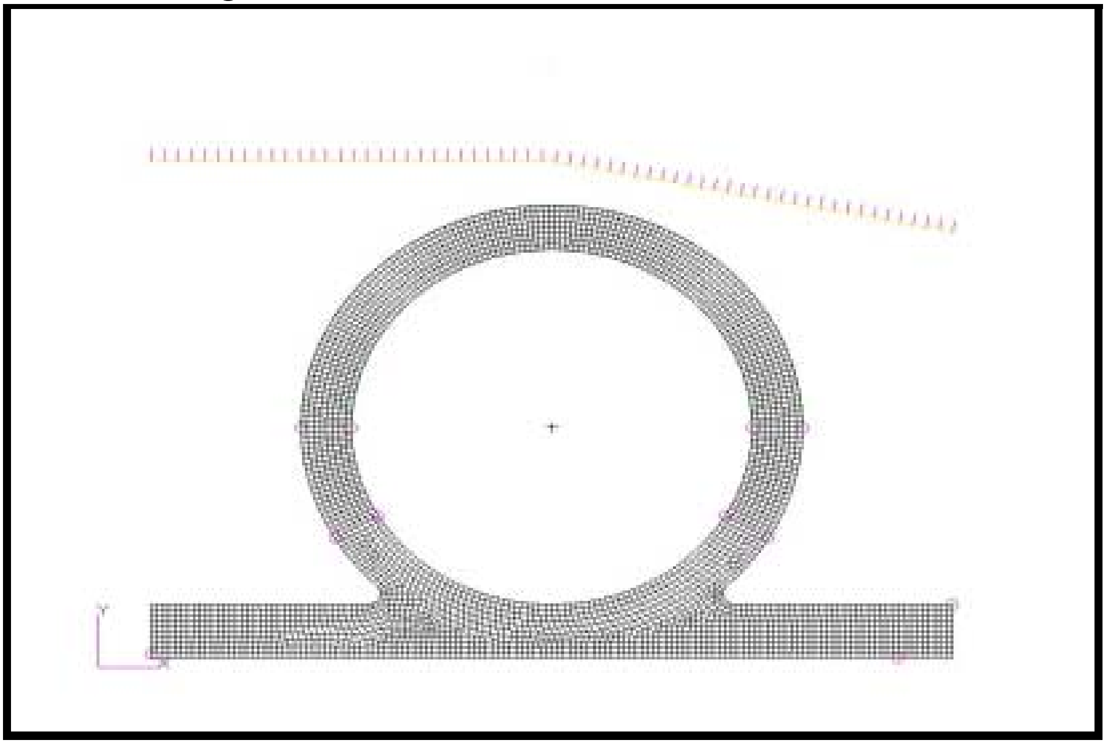
Target Element Type:	**1D**
Input Data...	

Make sure to flip the contact side. The side of the contact is always the opposite side the vectors are pointing.

	■ **Flip Contact Side**
Motion Control:	**Velocity**
Displacement (vector):	**<-0.06, -0.6, 0>**
OK	
Select Application Region...	
Geometry Filter:	● **Geometry**
Select Curves:	Choose all rigid body curves of bar, **Curve 2:4**
Add	
OK	
Apply	

The resulting contact definition is as shown in Figure 13.6:

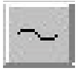
Figure 13.6 - Plot of contact definition markers



11. The following constraints will be used to fix the base of the model and control the movement of the rigid body. Use the figure above for Load and Boundry Condition application.

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**base_fixity**
Input Data...	
Translations:	**< 0, 0, >**
Rotations:	**< , , >**
OK	
Select Application Region...	
Geometry Filter:	● **Geometry**

Click on this icon to select the edge.



Curve or Edge

Select Geometric Entities: Select the base of the seal, **Surface 4.1**

Add

OK

Apply

12. Now you will prepare the model for analysis. We will use the default load case and output request for this analysis.

■ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**rubber_seal**

Load Step Creation...

Job Step Name:	**close_door**
Solution Type:	**Static**

Solution Parameters...

Linearity:	**NonLinear**
Nonlinear Geometric Effects:	**Large Displacement/ Large Strains**

Load Increment Parameters...

Arclength Method:	**None**
Total Time:	**1.0**

OK

Contact Table...

Parameters Defining Contact Between Bodies:	**Touch All**

OK

OK

Output Requests...

Select Element Results...

STRESS, COMPONENTS (311)

OK

OK

Apply

Cancel

13. Make sure only the **close_door** step is selected.

Load Step Selection...

Selected Job Steps: **close_door**

OK

Apply

The analysis job will take (on average) about 2 to 5 minutes to run. When the job is done there will be a results file titled **rubber_seal.t16** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at the ***rubber_seal.log*** and ***rubber_seal.sts*** files, as well as the ***rubber_seal.out*** file during or after the analysis. Also, you may use the **Analysis** application, **Monitor**, to ensure a successful run.

Action: **Monitor**

Object: **Job**

View Status File...

When the end of this file has the following line, you know that the job has completed successfully: Job ends with exit number : 3004

14. Read in the results

■ **Analysis**

Action: **Read Results**

Object: **Result Entities**

Method: **Attach**

Select Results File...

Selected Results File: **rubber_seal.t16**

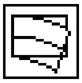
OK

Apply

15. Change the Display for postprocessing.

■ **Results**

Select the **Deform Attributes** icon



❐ **Show Undeformed Entities**

16. Now create a group for postprocessing.

Group/Create...

New Group Name: **fem**

■ **Make Current**

■ **Unpost All Other Groups**

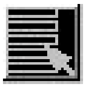
Group Contents: **Add All FEM**

Apply

Cancel

17. Create a deformed plot of the last analysis step.

Choose the **Select Results** icon



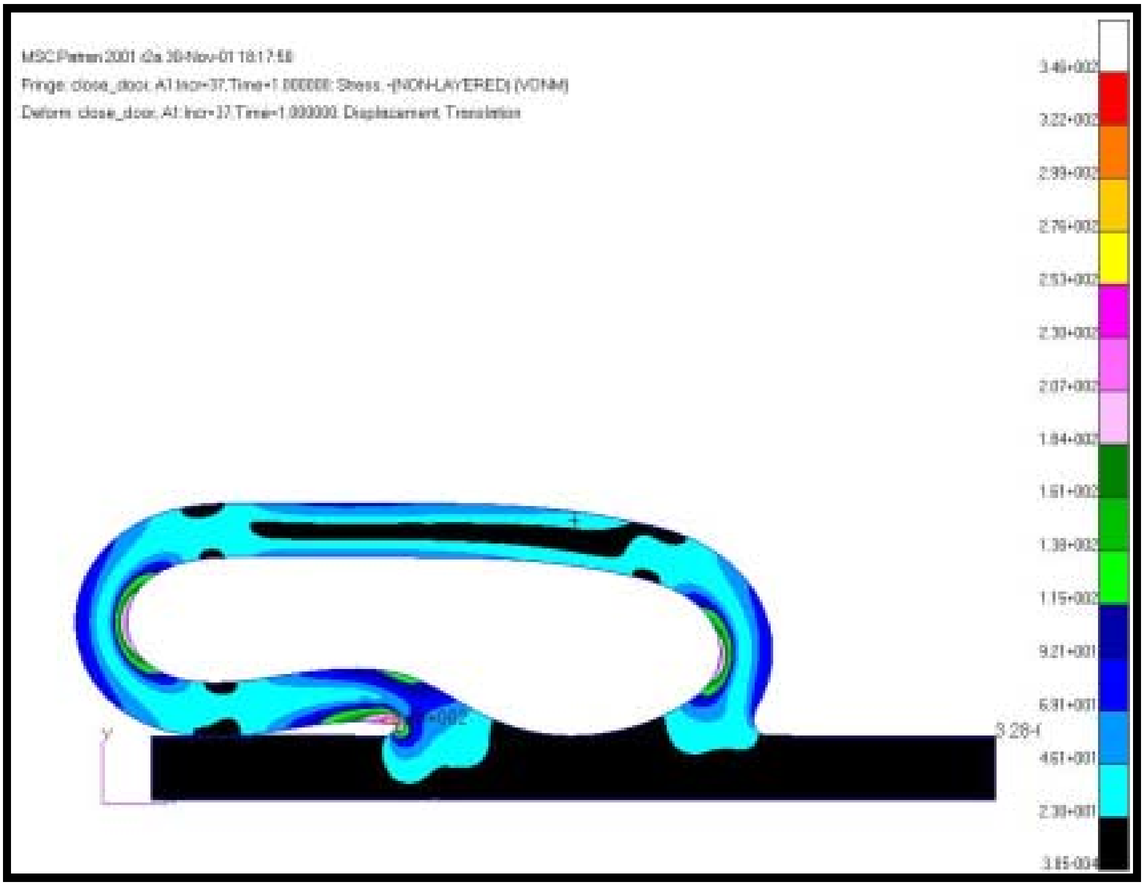
■ **Results**

Action:	**Create**
Object:	**Quick Plot**
Select Results Case:	select the last result case avalable
Select Fringe Results:	**Stress,**
Results Quantity:	**von Mises**
Select Deformation Result:	**Displacement, Translation**

Apply

You may wish to animate this using the *Results Animate* button. Your Model should look like Figure 13.7:

Figure 13.7 - Resulting deformation plot



Close the database and quit PATRAN.

This concludes the exercise