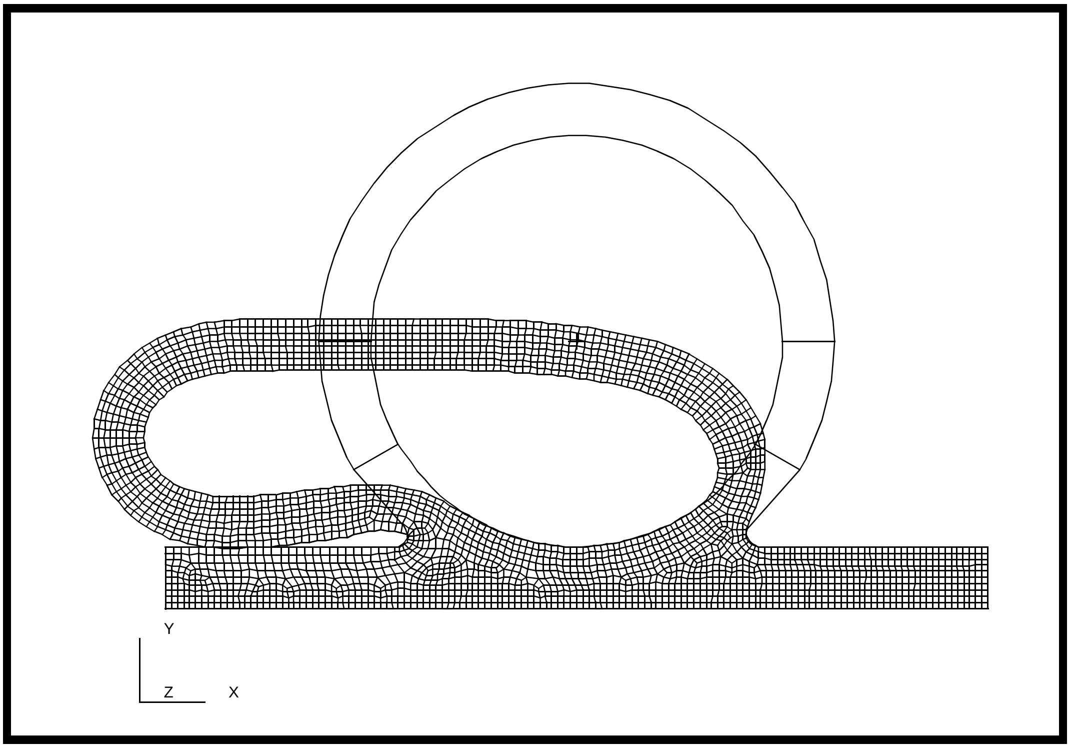
APPENDIX 5

Analysis of a Rubber Seal II



Objectives:

- Large displacement analysis
- Contact analysis using slide line and rigid body contact
- Hyper-elastic material model

Model Description:

In this Exercise, you will be increasing the trunk door displacement from exercise 9. The additional deflection will cause the rubber seal to contact itself. It will be necessary to model this contact constraint into the problem, otherwise the seal will pass through itself which can be observed in the results postprocessing. To properly handle this contact, you will be defining a general slide line contact condition. This requires the addition of bar elements to define the master (slide line) and the slave (ISL). The master and slave are defined using elements and assigning properties to them. Once you have defined the contact constraint and you have modified the trunk door displacement, you will submit the job using the same nonlinear step you have defined in exercise 9.

Suggested Exercise Steps:

- Model the contact surfaces with bar elements
- Create the element properties
- Modify the Loads and BCs
- Submit the job to analysis
- Evaluate the results

Exercise Procedure:

1. Open a the database named **rubber_seal.db**.

File/Open ...

Database Name: **rubber_seal.db**

OK

2. Define areas where the seal will contact itself.

A set of elements must be created at points where the seal contacts and/or slides across itself. You will use slide line/ISL elements for this type of contact. These elements are used to define a large sliding elastic-elastic body contact. For this contact pair, the master elements are the slide line elements and are modeled using Bar2 elements. The master surface is usually chosen as the surface with a coarser mesh or stiffer body. For this problem, the mesh density and the stiffness is the same, thus the choice will be somewhat arbitrary.

The ISL's represent the slave surface and need to be defined on the seal at any location in which the slide lines will come into contact with the seal. These elements are also modeled using Bar2 elements.

First, post only the seal.

Group/Post...

Select Groups to Post: **seal**

Apply

Create a group that will contain all the elements for the seal slide line/ ISL.

Group/Create...

New Group Name: **slide_seal**

■ **Make Current**

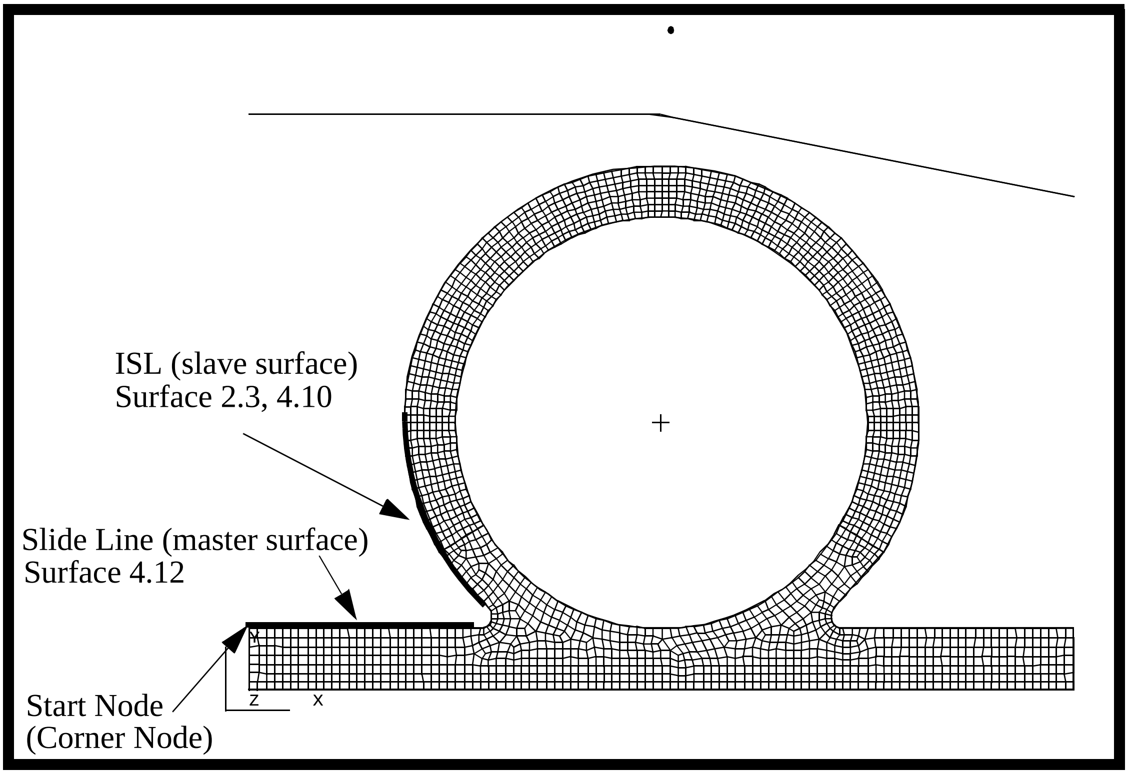
Apply

Cancel

Next, mesh the curves on the seal which will come-into contact with itself when the door closes (the door moving down and to the left in the figure below). Similar to rigid body definitions, the definition of the seal contact edges is important. If the seal edges come into contact with other portions of the seal that are incorrectly identified, the seal will pass through itself.

Figure A4.1 identifies surface edges that will contact the rigid surface of the door:

Figure A4.1 - Surfaces where seal contacts itself



◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Element Topology:	**Bar2**
Global Edge Length:	**.015**
Curve List:	see Figure A4.1

Apply

Equivalence the model.

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

Apply

3. Define the element properties where the seal will contact itself.

We need to define the orientation of the slide line. Specifically, we need to identify what is “inside” and what is “outside”. A node must be specified at one end of the slide line so that starting at that node and moving along the bar elements (this defines the S vector), the “outside” is defined as the left side of the rigid surface and “inside” is defined as the right. We can also write

Z = S x n

where Z is the positive out of plane 2D normal, n is the surface normal and S is a vector defined by the Surface Normal start node. To make the selection of the node easier, refer to the above diagram for the door ISL/slide line pair.

First, post the ISL/Slide Surface for the seal.

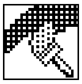
Group/Post...

Groups to Post:	**slide_seal**

Apply

Cancel

Retouch the graphics using the following toolbar icon:

 **Refresh Graphics**

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**ISL(in plane)**
Property Set Name:	**seal_isl**
Options:	**Planar**
	Elastic Slip Soft Contact

Input Properties...

ELSET name:	**slide**

Width:	**1.0**
OK	
Select Members:	Select the members corresponding to the slave surface (see Figure A4.1)

Click on this icon and select the elements for the slave surface (see Figure A4.1) Hold down control to select the elements.

Beam Element

Since the two groups of elements have a similar mesh densities and stiffness, the choice is arbitrary as to master and slave surface definition.

Add

Apply

Change the *Type* to **Slide Line**.

Action:	**Create**
Dimension:	**1D**
Type:	**Slide Line**
Property Set Name:	**seal_slideline**
Input Properties...	
ELSET name:	**slide**
Start Node:	see Figure A4.1
OK	
Select Members:	Select the members corresponding to the master surface (see Figure A4.1)

Add

Apply

4. Modify existing groups

Group/Modify...

Change Target Group...	**all**
All Entities:	**Add**
Change Target Group...	**fem**
Make Current	
All FEM:	**Add**
OK	

5. Modify the load and boundary conditions.

Now that the new model is complete, we need to modify the boundary conditions.

◆ **Load/BCs**

Action:	**Modify**
Object:	**Displacement**
Type:	**Nodal**
Select Set to Modify:	**close_door**
Modify Data...	
Translations:	**<-.07, -.7, >**
OK	
Apply	

6. Now you will prepare the model for analysis. In order to accommodate the large deformations, the time step of the analysis must be reduced.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**

Job Name: **seal_slide**

Step Creation ...

Available Job Steps: **close_door**

Solution Parameters ...

Minimum Delta-T: **1.0E-07**

OK

Apply

When asked if you wish to overwrite the existing step, respond **Yes**.

Yes

Cancel

Step Selection...

Selected Job Steps: **close_door**

Apply

Apply

The analysis job will take (on average) about 6 to 12 minutes to run. When the job is done there will be a results file titled **seal_slide.fil** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at **seal_slide.msg** and **seal_slide.sta** with the ***more*** command. Also, you may use ***ps -ef l grep afea*** and ***tail -lf seal_slide.sta*** to monitor the status.

7. Read in the results

◆ **Analysis**

Action: **Read Results**

Object: **Result Entities**

Method: **Translate**

Select Results File...

Selected Results File: **seal_slide.fil**

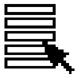
OK

Apply

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

◆ **Results**

Click on the **Select Results** icon



Action: **Create**

Object: **Quick Plot**

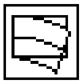
Select Result Cases: select the last increment

Select Fringe Results: **Stress, Components**

Results Quantity: **Von Mises**

Select Deformation Result: **Deformation, Displacements**

Change the **Display Properties** for results



Display/Results ...

Scale Factor: **1.0**

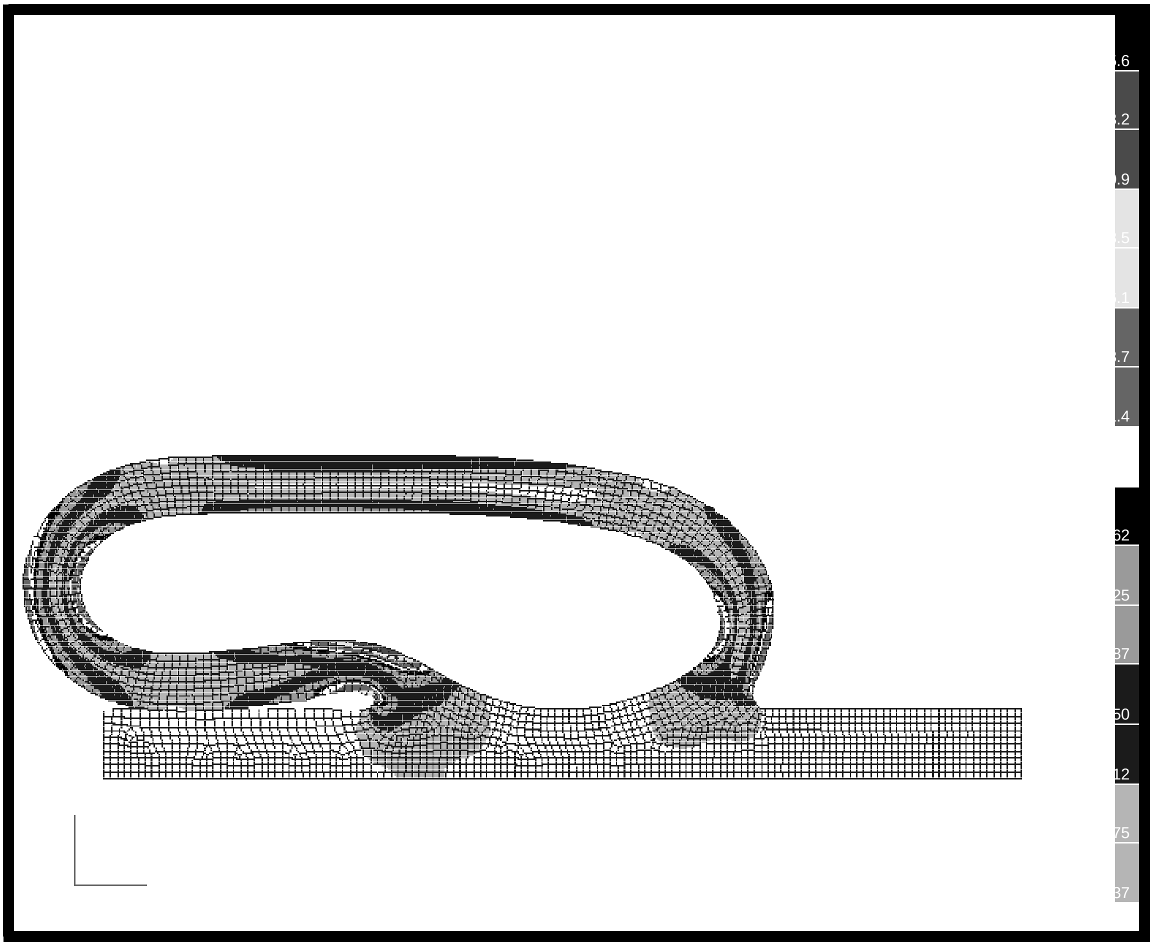
◆ **True Scale**

❒ **Show Undeformed Entities**

Apply

Your model should appear as shown in Figure A4.2:

Figure A4.2 - Resulting deformation of rubber seal



When done viewing the results, close the database and quit PATRAN.

This concludes the exercise.