Hyperelastic Seal

Introduction

In this model you study the force-deflection relation of a car door seal made from a soft rubber material. The model uses a hyperelastic material model together with formulations that can account for the large deformations and contact conditions.

You find the theory for hyperelastic materials in the section Hyperelastic Material Theory in the *Nonlinear Structural Materials Module User's Guide*.

Model Definition

The seal is compressed between a stationary plane surface and an indenting cylinder. It is of special interest to investigate the effect of air confined within the seal. Figure 1 shows the undeformed geometry of the seal and the contacting surfaces.

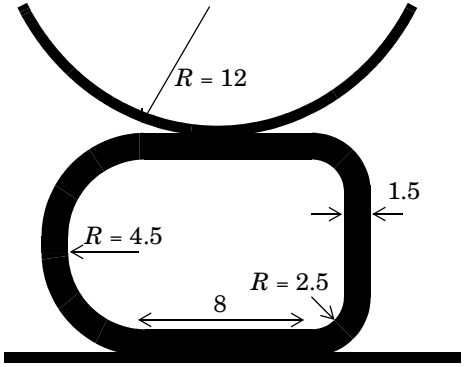


Figure 1: Model geometry.

The model describes a cross section of the seal assuming plane strain. The contacting surfaces are rigid when compared to the seal.

When computing the pressure from the air compressed inside the seal, the current cross-section area is required. A useful method for computing an area is by using Gauss' theorem, and converting to a contour integral:

$$A = \int 1 dA = \int \left(\nabla \cdot \begin{bmatrix} x \\ 0 \end{bmatrix} \right) dA = \oint x \hat{n}_x dl \quad (1)$$

You need to compute the integral in the deformed geometry, which is the default.

MATERIAL PROPERTIES

- The rubber is hyperelastic and is modeled as a Mooney-Rivlin material with $C_{10} = 0.37$ MPa and $C_{01} = 0.11$ MPa. The material is almost incompressible, so the bulk modulus is set to 10^4 MPa. A mixed formulation is automatically used for this material model.
- The compression of the confined air is assumed to be adiabatic, giving the pressure-density relation

$$\frac{p}{p_0} = \left(\frac{\rho}{\rho_0} \right)^{\gamma} = \left(\frac{A_0}{A} \right)^{\gamma} \quad (2)$$

Here the cross-section area is denoted by A, with the undeformed value $A_0 = 123.63$ mm^2. The constant γ has the value 1.4 and $p_0 = 0.1$ MPa is the standard air pressure. The load acting on the interior of the seal is then

$$\Delta p = p - p_0 = p_0 \left(\left(\frac{A_0}{A} \right)^{\gamma} - 1 \right) \quad (3)$$

CONSTRAINTS AND LOADS

- The lower straight part of the seal is glued to the car body, which is modeled with a spring foundation condition. The spring foundation uses glue stiffness as spring coefficient.
- One contact pair between the cylinder and the seal.
- One contact pair between the stationary plate and the seal.
- The rigid cylinder is lowered using the parameter of the parametric solver as the negative y displacement. It starts with a gap of 0 mm and is lowered 4 mm.

 ©2013 COMSOL

Results and Discussion

Figure 2 shows the deformed shape at the lowest cylinder position—corresponding to an indentation of 4 mm—without internal pressure. The deformation scale is 1:1, that is, a true shape. The plot shows a detachment region of significant size.

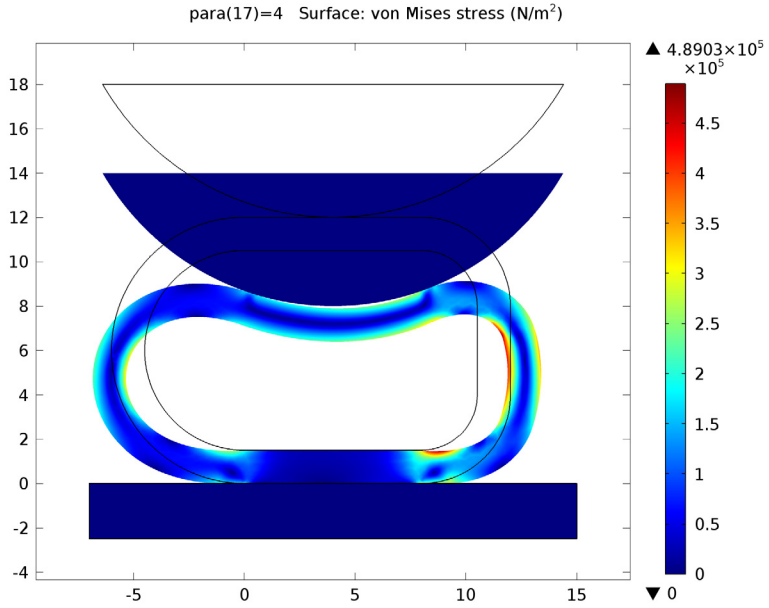


Figure 2: Seal deformation at 4 mm indentation without internal pressure.

Figure 3 shows the corresponding contact pressure plot. The detachment region appears first at indentation just over 2.5 mm and grows as the indentation increases further. The actual contact areas are reduced to two spots at the sides.

Such a significant change in the contact pressure distribution indicates that the computations must be performed using a fine mesh together with sufficiently small steps in the parametric analysis with respect to the indentation value.

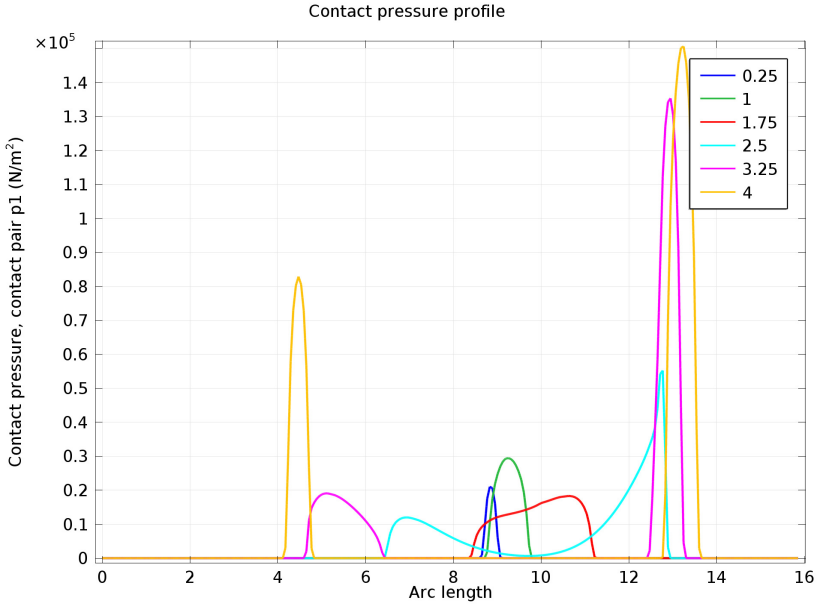


Figure 3: The contact pressure distribution over the area between the seal and cylinder for different indentations without internal pressure.

Figure 4 shows the result of the computations with the internal pressure taken into account. The the seal profile appears inflated.

©2013 COMSOL

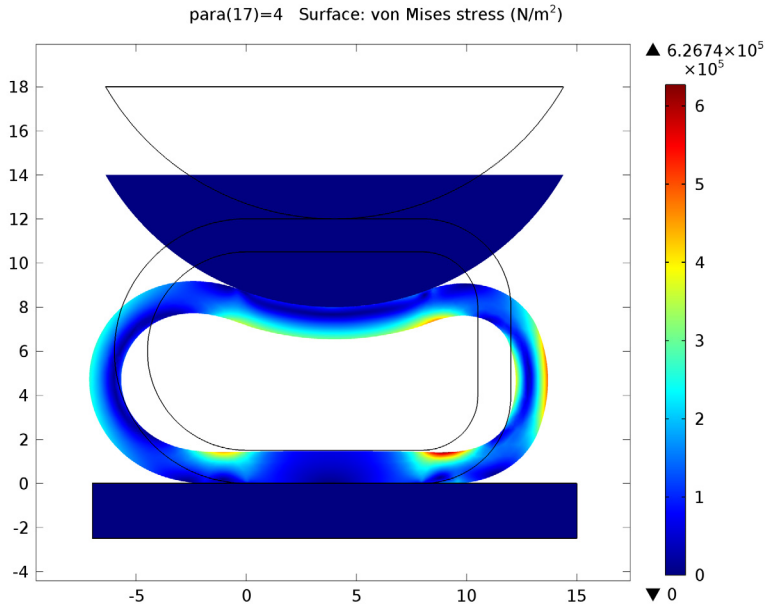


Figure 4: Seal deformation at 4 mm indentation with the internal pressure.

The contact pressure plot in Figure 5 confirms that the detachment region never appears even though the contact pressure has a pronounced minimum in the middle part.

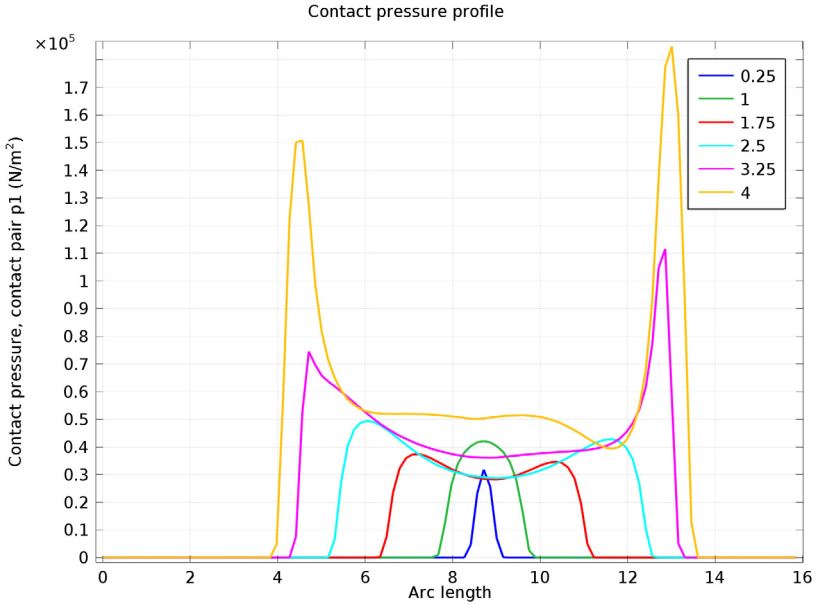


Figure 5: The contact pressure distribution for different indentations with the internal pressure taken into account.

Figure 6 contains a plot of the force per unit length versus compression (displacement of the rigid cylinder) with and without the internal pressure taken into account.

©2013 COMSOL

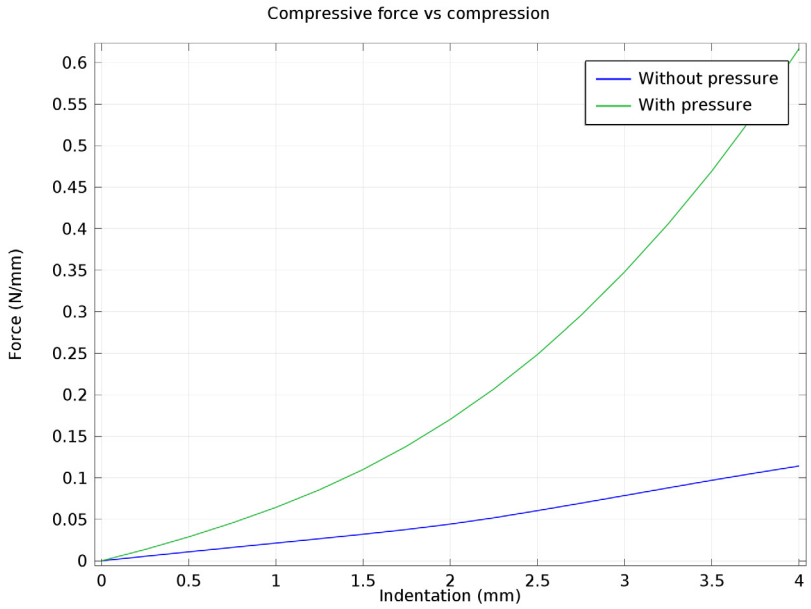


Figure 6: Force per unit length versus compression with and without internal pressure.

Notice that the forces needed to compress the seal can be up to one order of magnitude larger when the effect of the air is taken into account.

In reality, a car door seal contains small holes through which the air can escape as long as the compression is not too fast. Thus the computed values are the limits corresponding to very slow and very fast compression, respectively.

Model Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/hyperelastic_seal`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

7 Click **Finish**.

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Circle 1

1 Right-click **Model 1>Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `6`.

4 Locate the **Position** section. In the **y** edit field, type `6`.

5 Locate the **Rotation Angle** section. In the **Rotation** edit field, type `45`.

6 Click the **Build Selected** button.

Circle 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `4`.

4 Locate the **Position** section. In the **x** edit field, type `8`.

5 In the **y** edit field, type `4`.

6 Locate the **Rotation Angle** section. In the **Rotation** edit field, type `45`.

7 Click the **Build Selected** button.

8 Click the **Zoom Extents** button on the Graphics toolbar.

Circle 3

1 Right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `4`.

4 Locate the **Position** section. In the **x** edit field, type `8`.

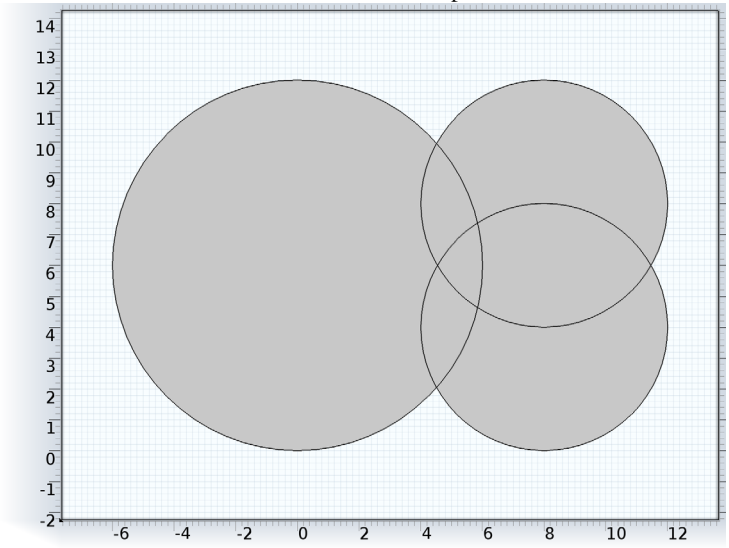
5 In the **y** edit field, type `8`.

©2013 COMSOL

6 Locate the **Rotation Angle** section. In the **Rotation** edit field, type 45.

7 Click the **Build All** button.

8 Click the **Zoom Extents** button on the Graphics toolbar.



Rectangle 1

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type 8.

4 In the **Height** edit field, type 12.

5 Click the **Build Selected** button.

Rectangle 2

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

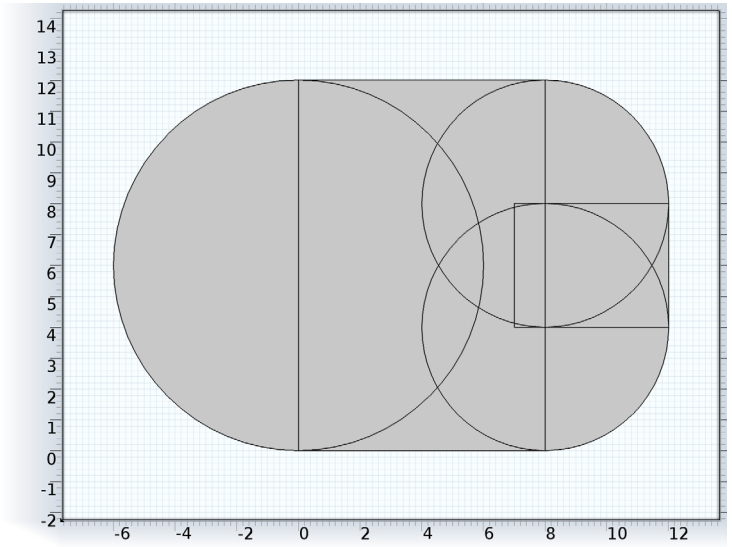
3 In the **Width** edit field, type 5.

4 In the **Height** edit field, type 4.

5 Locate the **Position** section. In the **x** edit field, type 7.

6 In the **y** edit field, type 4.

7 Click the **Build Selected** button.



Union 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Union**.

2 Click in the **Graphics** window, press Ctrl+A to highlight all objects, and then right-click to confirm the selection.

3 In the **Union** settings window, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

5 Click the **Build Selected** button.

Circle 4

1 Right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `4.5`.

4 Locate the **Position** section. In the **y** edit field, type `6`.

5 Click the **Build Selected** button.

Circle 5

1 Right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `2.5`.

©2013 COMSOL

4 Locate the **Position** section. In the **x** edit field, type `8`.

5 In the **y** edit field, type `4`.

6 Click the **Build Selected** button.

Circle 6

1 Right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `2.5`.

4 Locate the **Position** section. In the **x** edit field, type `8`.

5 In the **y** edit field, type `8`.

6 Click the **Build Selected** button.

Rectangle 3

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `8`.

4 In the **Height** edit field, type `9`.

5 Locate the **Position** section. In the **y** edit field, type `1.5`.

6 Click the **Build Selected** button.

Rectangle 4

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `3.5`.

4 In the **Height** edit field, type `4`.

5 Locate the **Position** section. In the **x** edit field, type `7`.

6 In the **y** edit field, type `4`.

7 Click the **Build Selected** button.

Difference 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.

2 In the **Difference** settings window, locate the **Difference** section.

3 Under **Objects to add**, click **Activate Selection**.

4 Select the object **uni1** only.

5 Under **Objects to subtract**, click **Activate Selection**.

6 Select the objects **c4**, **c5**, **c6**, **r3**, and **r4** only.

7 Click the **Build Selected** button.

Rectangle 5

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `22`.

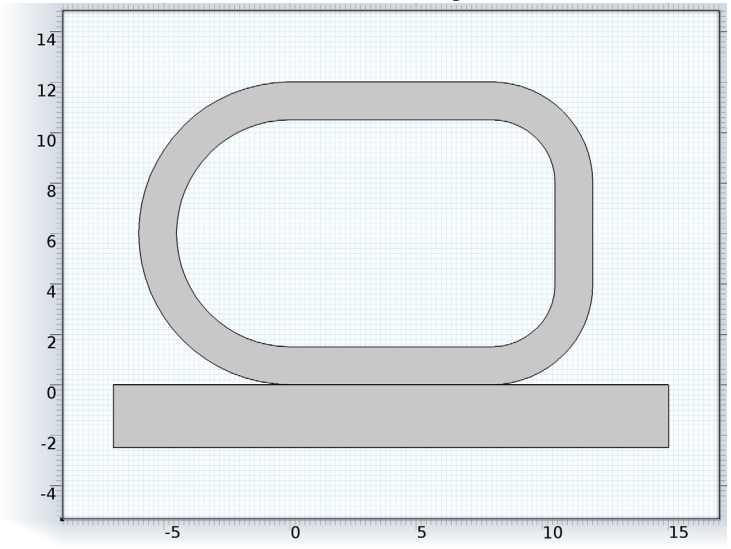
4 In the **Height** edit field, type `2.5`.

5 Locate the **Position** section. In the **x** edit field, type `-7`.

6 In the **y** edit field, type `-2.5`.

7 Click the **Build Selected** button.

8 Click the **Zoom Extents** button on the Graphics toolbar.



Circle 7

1 Right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `12`.

4 Locate the **Position** section. In the **x** edit field, type `4`.

5 In the **y** edit field, type `24`.

6 Click the **Build Selected** button.

©2013 COMSOL

7 Click the **Zoom Extents** button on the Graphics toolbar.

Rectangle 6

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type 22.

4 In the **Height** edit field, type 6.

5 Locate the **Position** section. In the **x** edit field, type -7.

6 In the **y** edit field, type 12.

7 Click the **Build Selected** button.

Intersection 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Intersection**.

2 Select the objects **r6** and **c7** only.

3 Click the **Build Selected** button.

4 Click the **Zoom Extents** button on the Graphics toolbar.

Form Union

1 In the **Model Builder** window, under **Model 1>Geometry 1** click **Form Union**.

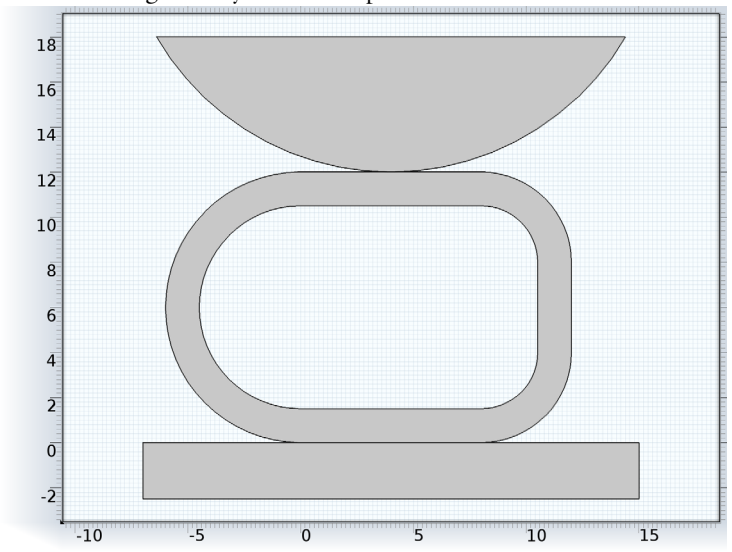
2 In the **Finalize** settings window, locate the **Finalize** section.

3 From the **Finalization method** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Click the **Build Selected** button.

The model geometry is now complete.



GLOBAL DEFINITIONS

Add a parameter that you can use to gradually increase the vertical displacement.

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
para	0	Vertical displacement parameter

DEFINITIONS

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

 ©2013 COMSOL

4 Select Boundaries 9, 10, 12, 14, 15, 20, and 21 only.

5 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Inner Boundary` in the **New name** edit field.

7 Click **OK**.

8 Right-click **Definitions** and choose **Pairs>Contact Pair**.

9 In the **Pair** settings window, locate the **General** section.

10 In the **Pair name** edit field, type `upper`.

11 Select Boundaries 6 and 7 only.

12 Click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundaries 11, 18, and 22 only.

13 Right-click **Definitions** and choose **Pairs>Contact Pair**.

14 In the **Pair** settings window, locate the **General** section.

15 In the **Pair name** edit field, type `lower`.

16 Select Boundary 3 only.

17 Click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundaries 17 and 19 only.

SOLID MECHANICS

1 In the **Model Builder** window, under **Model 1** click **Solid Mechanics**.

2 In the **Solid Mechanics** settings window, locate the **Thickness** section.

3 In the d edit field, type `50[mm]`.

In the plane strain approximation, this setting only affects total force computations.

Hyperelastic Material 1

1 Right-click **Model 1>Solid Mechanics** and choose **Hyperelastic Material**.

2 In the **Hyperelastic Material** settings window, locate the **Hyperelastic Material** section.

3 From the **Material model** list, choose **Mooney-Rivlin, two parameters**.

4 Select Domain 3 only.

5 In the κ edit field, type `1e4[MPa]`.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **More>Fixed Constraint**.

2 Select Domain 1 only.

Prescribed Displacement 1

1 Right-click **Solid Mechanics** and choose **More>Prescribed Displacement**.

2 Select Domain 2 only.

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box. Leave the value in the u_0 edit field at 0

5 Select the **Prescribed in y direction** check box.

6 In the u_{0y} edit field, type `-para*1[mm]`.

You add an elastic spring foundation at the seal bottom to model the glue layer.

Spring Foundation 1

1 Right-click **Solid Mechanics** and choose **Spring Foundation**.

2 Select Boundary 8 only.

3 In the **Spring Foundation** settings window, locate the **Spring** section.

4 In the $\mathbf{k}_A$ table, enter the following settings:

1e10	x
2e10	y

Contact 1

1 Right-click **Solid Mechanics** and choose **Pairs>Contact**.

2 In the **Contact** settings window, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 1**.

Friction 1

1 Right-click **Model 1>Solid Mechanics>Contact 1** and choose **Friction**.

2 In the **Friction** settings window, locate the **Friction** section.

3 In the μ_{stat} edit field, type `0.2`.

Contact 2

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Pairs>Contact**.

2 In the **Contact** settings window, locate the **Pair Selection** section.

3 In the **Pairs** list, select **Contact Pair 2**.

 ©2013 COMSOL

Friction 1

1 Right-click **Model 1>Solid Mechanics>Contact 2** and choose **Friction**.

2 In the **Friction** settings window, locate the **Friction** section.

3 In the μ_{stat} edit field, type `0.2`.

DEFINITIONS

Integration 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `AreaInt`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Inner Boundary**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Inner Boundary**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression
`EnclosedArea`	`AreaInt(-x*solid.nx)`
`int_p`	`0.1[MPa]*((123.63[mm^2]/EnclosedArea)^1.4-1)`

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 Select Domain 3 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value
Density	rho	1100[kg/m^3]
Model parameters	C10	0.37[MPa]
Model parameters	C01	0.11[MPa]

Because all displacements on the rigid parts are prescribed, the choice of material is insignificant. For the sake of clarity, select steel. It is also possible to use the rubber material for the entire model.

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Structural steel**.

3 Click **Add Material to Model**.

Structural steel

1 In the **Model Builder** window, under **Model 1>Materials** click **Structural steel**.

2 Select Domains 1 and 2 only.

MESH 1

Free Triangular 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Free Triangular**.

Size 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 8, 11, 17–19, and 22 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type `0.15`.

Size 2

1 Right-click **Free Triangular 1** and choose **Size**.

 ©2013 COMSOL

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 6 and 7 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type `0.4`.

Size 3

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

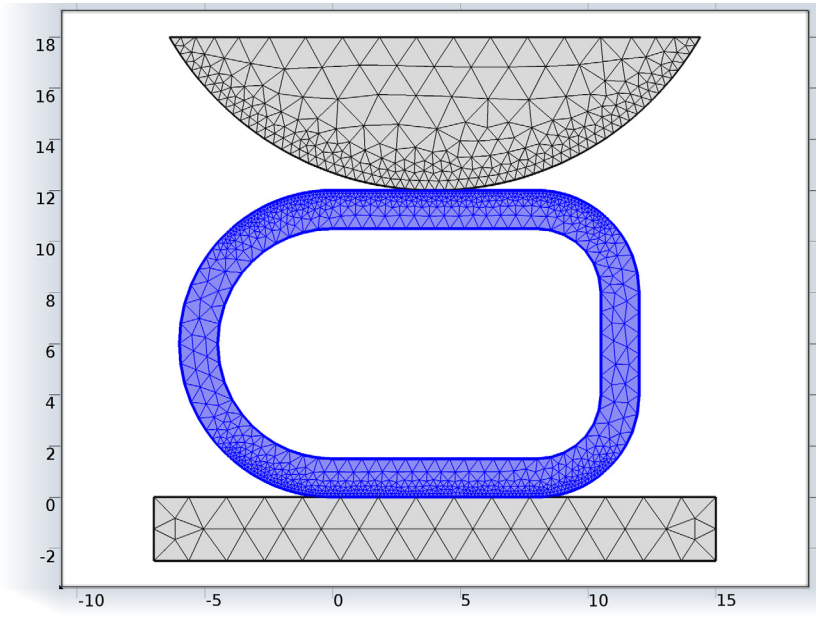
4 Select Domain 3 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type `0.8`.

8 Click the **Build All** button.



STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Stationary** settings window, click to expand the **Study Extensions** section.

3 Select the **Continuation** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Continuation parameter	Parameter value list
para	`range(0,0.25,4)`

6 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

The default scale for the displacement variables is calculated from the entire geometry size. For models with prescribed displacements as domain or boundary constraints, the maximum prescribed displacement usually gives a better estimate of the scale.

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** node, then click **Displacement field (Material) (mod1.u)**.

2 In the **Field** settings window, locate the **Scaling** section.

3 In the **Scale** edit field, type `1e-3`.

Change the scales for the pressure and contact variables to account for the material properties of the seal made of soft rubber.

4 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **Pressure (mod1.solid.pw)**.

5 In the **Field** settings window, locate the **Scaling** section.

6 In the **Scale** edit field, type `1e5`.

7 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **Contact pressure, contact pair p1 (mod1.solid.Tn_p1)**.

8 In the **Field** settings window, locate the **Scaling** section.

9 In the **Scale** edit field, type `1e5`.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **Contact pressure, contact pair p2 (mod1.solid.Tn_p2)**.

11 In the **Field** settings window, locate the **Scaling** section.

 ©2013 COMSOL

12 In the **Scale** edit field, type `1e5`.

13 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **Friction force, contact pair p1 (Spatial) (mod1.solid.Tt_p1)**.

14 In the **Field** settings window, locate the **Scaling** section.

15 In the **Scale** edit field, type `1e4`.

16 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **Friction force, contact pair p2 (Spatial) (mod1.solid.Tt_p2)**.

17 In the **Field** settings window, locate the **Scaling** section.

18 In the **Scale** edit field, type `1e4`.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Stationary** settings window, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

The default plot shows the von Mises stress distribution in the seal; see Figure 2.

The following steps show how to display the contact pressure at the top of the seal.

1D Plot Group 2

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Parameter selection (para)** list, choose **Manual**.

4 In the **Parameter indices (1-17)** edit field, type `range(2,3,17)`.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Contact pressure profile`.

7 Right-click **Results>1D Plot Group 2** and choose **Rename**.

8 Go to the **Rename 1D Plot Group** dialog box and type `Contact Force Without Pressure` in the **New name** edit field.

9 Click **OK**.

Contact Force Without Pressure

1 Right-click **Results>1D Plot Group 2** and choose **Line Graph**.

2 Select Boundaries 11, 18, and 22 only.

3 In the **Line Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Solid Mechanics>Contact>Contact pressure, contact pair p1 (solid.Tn_p1)**.

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. Set the **Width** value to **1**.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 Click the **Plot** button.

The plot in the **Graphics** window should now look like that in Figure 3.

Now you can compute the solution including the internal pressure.

SOLID MECHANICS

Boundary Load 1

1 In the **Model Builder** window, under **Model 1** right-click **Solid Mechanics** and choose **Boundary Load**.

2 In the **Boundary Load** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inner Boundary**.

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the *p* edit field, type `int_p`.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

3 Click **Finish**.

STUDY 2

Step 1: Stationary

1 In the **Stationary** settings window, click to expand the **Study Extensions** section.

2 Select the **Continuation** check box.

 ©2013 COMSOL

3 Click **Add**.

4 In the table, enter the following settings:

Continuation parameter	Parameter value list
para	`range(0,0.25,4)`

Scale the fields in the same way as you did in Study 1.

5 In the **Model Builder** window, expand the **Study 2** node.

Solver 2

1 Right-click **Study 2** and choose **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solver 2>Dependent Variables 1** node, then click **Contact pressure, contact pair p1 (mod1.solid.Tn_p1)**.

3 In the **Field** settings window, locate the **Scaling** section.

4 In the **Scale** edit field, type `1e5`.

5 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2>Dependent Variables 1** click **Contact pressure, contact pair p2 (mod1.solid.Tn_p2)**.

6 In the **Field** settings window, locate the **Scaling** section.

7 In the **Scale** edit field, type `1e5`.

8 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2>Dependent Variables 1** click **Displacement field (Material) (mod1.u)**.

9 In the **Field** settings window, locate the **Scaling** section.

10 In the **Scale** edit field, type `1e-3`.

11 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2>Dependent Variables 1** click **Pressure (mod1.solid.pw)**.

12 In the **Field** settings window, locate the **Scaling** section.

13 In the **Scale** edit field, type `1e5`.

14 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2>Dependent Variables 1** click **Friction force, contact pair p1 (Spatial) (mod1.solid.Tt_p1)**.

15 In the **Field** settings window, locate the **Scaling** section.

16 In the **Scale** edit field, type `1e4`.

17 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2>Dependent Variables 1** click **Friction force, contact pair p2 (Spatial) (mod1.solid.Tt_p2)**.

18 In the **Field** settings window, locate the **Scaling** section.

19 In the **Scale** edit field, type `1e4`.

20 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Stress (solid) 1

You can see that the detachment region has disappeared as a result of the seal pressurization; compare with Figure 4.

Contact Force Without Pressure 1

1 In the **Model Builder** window, under **Results** right-click **Contact Force Without Pressure** and choose **Duplicate**.

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

4 Right-click **Results>Contact Force Without Pressure 1** and choose **Rename**.

5 Go to the **Rename 1D Plot Group** dialog box and type `Contact Force with Pressure` in the **New name** edit field.

6 Click **OK**.

Derived Values

Finally, compute the force needed for the compression as the sum of all vertical reaction forces on the indentor.

1 In the **Model Builder** window, right-click **Derived Values** and choose **Integration>Surface Integration**.

2 Select Domain 2 only.

3 In the **Surface Integration** settings window, locate the **Expression** section.

4 In the **Expression** edit field, type `-solid.RFy/solid.d`.

5 From the **Unit** list, choose **N/mm**.

6 Locate the **Integration Settings** section. From the **Method** list, choose **Summation**.

7 Click the **Evaluate** button.

8 Locate the **Data** section. From the **Data set** list, choose **Solution 2**.

 ©2013 COMSOL

9 Right-click **Results>Derived Values>Surface Integration 1** and choose **Evaluate>Table 1 - Surface Integration 1 (-solid.RFy/solid.d)**.

10 In the **Table** window, click **Table Graph**.

1D Plot Group 5

1 In the **Model Builder** window, under **Results>1D Plot Group 5** click **Table Graph 1**.

2 In the **Table Graph** settings window, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Without pressure
With pressure

6 In the **Model Builder** window, click **1D Plot Group 5**.

7 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

8 Select the **x-axis label** check box.

9 In the associated edit field, type `Indentation (mm)`.

10 Select the **y-axis label** check box.

11 In the associated edit field, type `Force (N/mm)`.

12 Locate the **Title** section. From the **Title type** list, choose **Manual**.

13 In the **Title** text area, type `Compressive force vs compression`.

14 Click the **Plot** button.

Compare with the plot shown in Figure 6.

The analysis is now finished. If you want to store this model and re-use it later, you can introduce a load group which will able you to re-compute both studies (with and without inner pressure) directly.

GLOBAL DEFINITIONS

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Load Group**.

2 In the **Load Group** settings window, locate the **Group Identifier** section.

3 In the **Identifier** edit field, type `pressure`.

SOLID MECHANICS

Boundary Load 1

In the **Model Builder** window, under **Model 1>Solid Mechanics** right-click **Boundary Load 1** and choose **Load Group>Load Group 1**.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Stationary** settings window, locate the **Study Extensions** section.

3 Select the **Define load cases** check box.

4 Click **Add**.

5 In the **Model Builder** window, right-click **Study 1** and choose **Rename**.

6 Go to the **Rename Study** dialog box and type `Without Pressure` in the **New name** edit field.

7 Click **OK**.

STUDY 2

Step 1: Stationary

1 In the **Stationary** settings window, locate the **Study Extensions** section.

2 Select the **Define load cases** check box.

3 Click **Add**.

4 In the table, enter the following settings:

pressure
√

5 In the **Model Builder** window, right-click **Study 2** and choose **Rename**.

6 Go to the **Rename Study** dialog box and type `With Pressure` in the **New name** edit field.

7 Click **OK**.

©2013 COMSOL