Created in COMSOL Multiphysics 5.6

Hyperelastic Seal

Introduction

In this example 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.

It is of special interest to investigate the effect of air confined within the seal.

See the *Nonlinear Structural Materials Module User's Guide* for theory about hyperelastic material.

Model Definition

The seal is compressed between a stationary plane surface and an indenting cylinder. There is also a vertical rigid wall at a distance of 1 mm from the initial position of the seal. Figure 1 shows the undeformed geometry of the seal and the contacting surfaces.

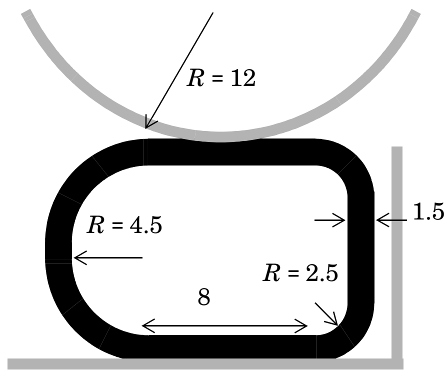


Figure 1: Model geometry.

The model describes a cross section of the seal assuming plane strain conditions. The (arbitrary) thickness in the out-of-plane direction is chosen as 50 mm. 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 the divergence theorem, and converting the original surface integral to a contour integral:

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

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

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

Here the cross-section area is denoted by A, with the undeformed value $A_0 = 123.63\ \mathrm{mm}^2$. 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

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

CONSTRAINTS AND LOADS

- One contact pair is used between the cylinder and the seal.
- One contact pair is used between the stationary plates and the seal.
- The lower straight part of the seal is glued to the car body. This is modeled with an adhesion condition.
- The rigid cylinder is lowered using the parameter of the parametric continuation solver as the negative y displacement. It starts with a gap of 0 mm and is lowered 4 mm.

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.

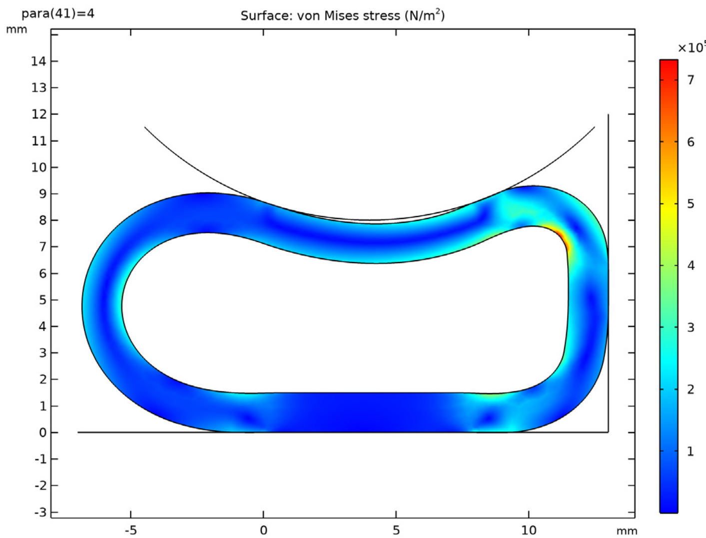


Figure 2: Seal deformation at 4 mm indentation without internal pressure.

Figure 3 shows the corresponding contact pressure plot. The detachment region appears first at an 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.

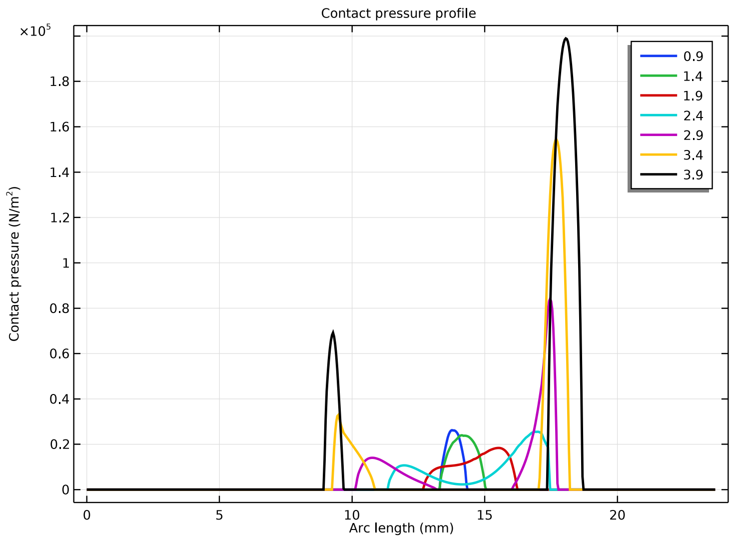


Figure 3: The contact pressure distribution over the area between the seal and cylinder for different indentations without internal pressure.

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

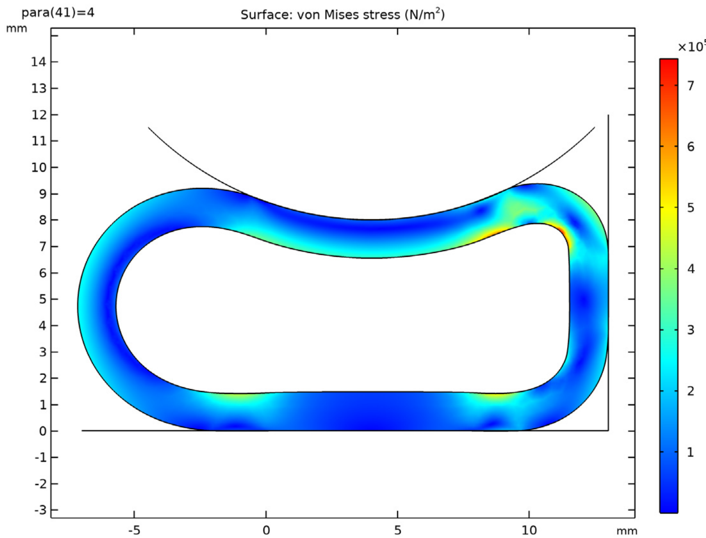


Figure 4: Seal deformation at 4 mm indentation with the internal pressure.

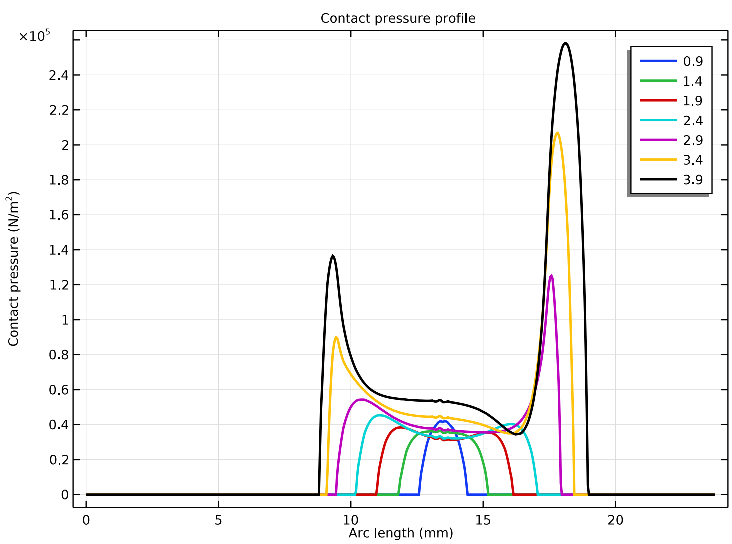


Figure 5: The contact pressure distribution for different indentations with the internal pressure taken into account.

Figure 6 contains a plot of the force per unit length versus the indentation of the rigid cylinder, with and without the internal pressure taken into account. The distinct change in slope of the curves is when the rightmost part of the seal comes into contact with the vertical wall, and no longer can deform in that direction.

Notice that the forces needed to compress the seal can be almost one order of magnitude larger when the effect of the confined 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.

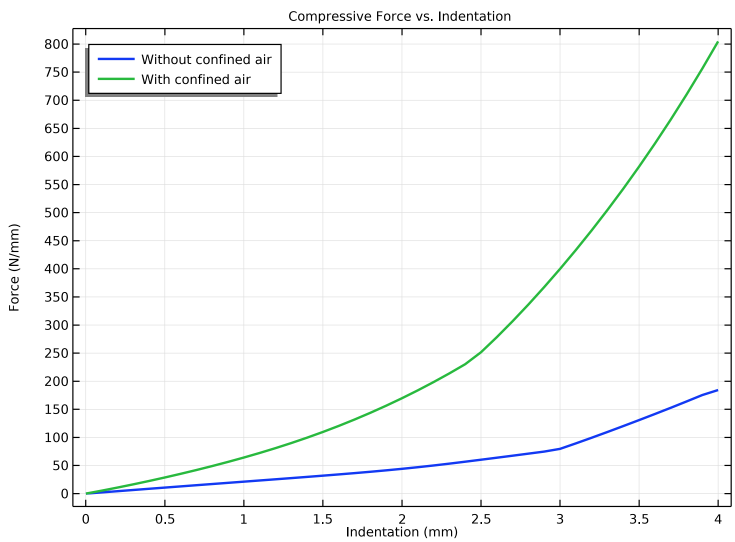


Figure 6: Compressive force per unit length versus indentation with and without internal pressure.

Notes About the COMSOL Implementation

As an optional feature of the model, an **Elastic Predeformation** node is added to the physics. This feature updates the geometry with the deformation from the previous increment, meaning that, for example, the 'incremental' deformation gradient becomes smaller, which can improve convergence when deformations are large. Note that the total deformation gradient still considers the deformation with respect to the reference configuration of the geometry. Adding this feature speeds up the convergence of the model, but it would solve and converge to the same results also without the **Elastic Predeformation** node.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/hyperelastic_seal`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GEOMETRY 1

If you do not want to build all the geometry, you can load the geometry sequence from the stored model. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `hyperelastic_seal.mph`. You can then continue to the **Global Definitions** section below.

To build the geometry from scratch, continue from here.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

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

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

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **y** text field, type `6`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	1.5

Circle 2 (c2)

1 In the **Geometry** toolbar, click **Circle**.

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

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

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **x** text field, type `8`.

6 In the **y** text field, type `8`.

7 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	1.5

Circle 3 (c3)

1 In the **Geometry** toolbar, click **Circle**.

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

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

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **x** text field, type `8`.

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

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

8 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	1.5

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type 8.
4 In the **Height** text field, type 12.
5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	1.5
Layer 2	9

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type 1.5.
4 In the **Height** text field, type 4.
5 Locate the **Position** section. In the **x** text field, type 10.5.
6 In the **y** text field, type 4.
7 Click **Build Selected**.
8 Click the **Zoom Extents** button in the **Graphics** toolbar.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 On the object **c1**, select Domain 2 only.
5 On the object **c2**, select Domain 1 only.
6 On the object **c3**, select Domain 1 only.
7 On the object **r1**, select Domain 2 only.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Click in the **Graphics** window and then press Ctrl+A to select all objects.
3 In the **Settings** window for **Union**, locate the **Union** section.
4 Clear the **Keep interior boundaries** check box.

Create the indentor.

Indenter

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, type `Indenter` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `12`.

4 In the **Sector angle** text field, type `90`.

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

6 In the **y** text field, type `24`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-135`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 From the **Show in physics** list, choose **Boundary selection**.

Create the support.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `20`.

4 Locate the **Position** section. In the **x** text field, type `-7`.

5 In the **y** text field, type `-1`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **Boundary selection**.

8 Find the **Cumulative selection** subsection. Click **New**.

9 In the **New Cumulative Selection** dialog box, type `Rigid base` in the **Name** text field.

10 Click **OK**.

Rectangle 4 (r4)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type `12`.

4 Locate the **Position** section. In the **x** text field, type `13`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Rigid base**.

Convert to Curve 1 (ccur1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Curve**.

2 Select the objects **c4**, **r3**, and **r4** only.

3 In the **Settings** window for **Convert to Curve**, click **Build Selected**.

Delete Entities 2 (del2)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **ccur1**, select Boundaries 1, 2, 4–6, and 8–10 only.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

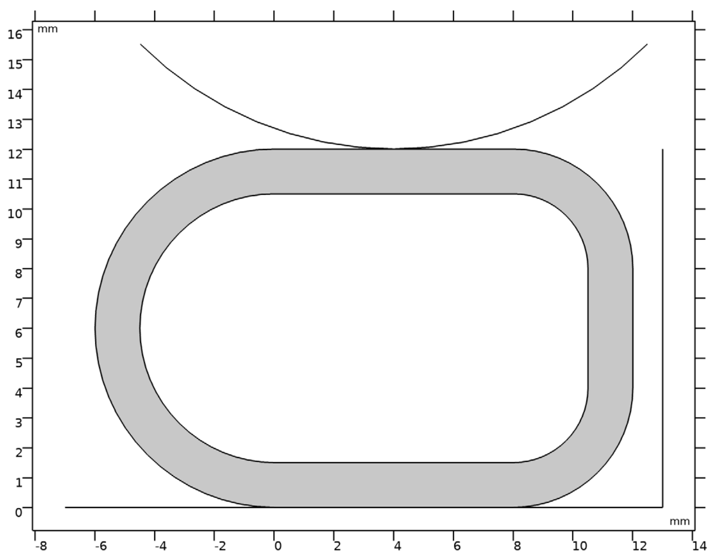
3 From the **Action** list, choose **Form an assembly**.

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

5 Click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

The model geometry is now complete.



Inner seal boundary

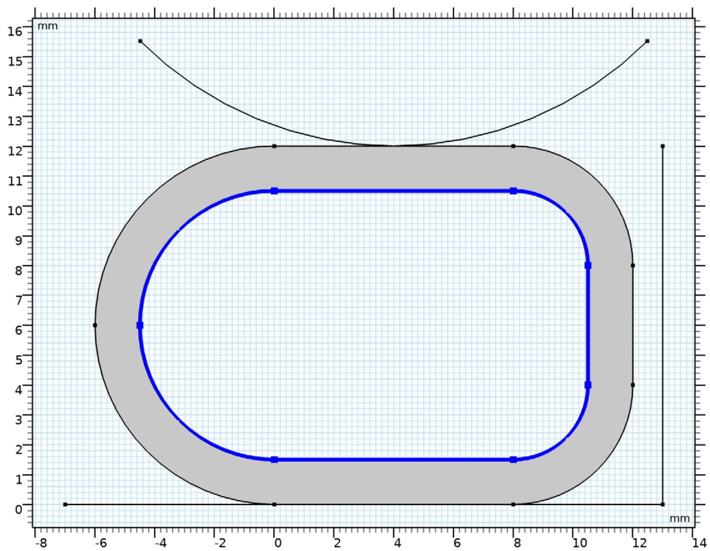
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Inner seal boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** check box.

5 Select one of the boundaries on the inside of the seal.



Outer seal boundary

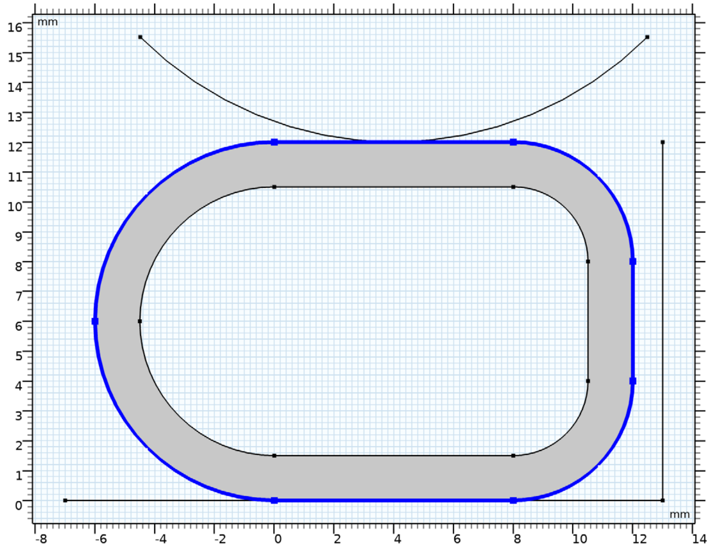
1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Outer seal boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** check box.

5 Select one of the boundaries on the outside of the seal.



Glued seal boundary

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Glued seal boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 4 only.

GLOBAL DEFINITIONS

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

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`para`	`0`	0	`Vertical displacement parameter`
`d`	`50[mm]`	0.05 m	`Out-of-plane thickness`

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, type `upper` in the **Pair name** text field.

3 Locate the **Source Boundaries** section. From the **Selection** list, choose **Indenter**.

4 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Outer seal boundary**.

Contact Pair 2 (p2)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, type `lower` in the **Pair name** text field.

3 Locate the **Source Boundaries** section. From the **Selection** list, choose **Rigid base**.

4 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Outer seal boundary**.

The boundaries in the contact pairs are unnecessarily large because it was convenient to reuse existing selections. In large 3D models, you should however keep down the size of the contact boundaries for performance reasons.

Prescribe the deformation of the indentor by using a moving mesh with prescribed deformation. An alternative would have been to include the indentor in the Solid Mechanics interface, and prescribe its deformation.

Prescribed Deformation 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Prescribed Deformation**.

2 In the **Settings** window for **Prescribed Deformation**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Indenter**.

5 Locate the **Prescribed Deformation** section. Specify the *dx* vector as

`0`	X
`-para*1[mm]`	Y

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

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

3 In the *d* text field, type `d`.

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

Hyperelastic Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.

2 In the **Settings** window for **Hyperelastic Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney-Rivlin, two parameters**.

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

Contact 1

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

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

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (upper)** in the **Pairs** list.

5 Click **OK**.

The source boundaries are not within the Solid Mechanics interface itself. This must be indicated.

6 In the **Settings** window for **Contact**, locate the **Contact Surface** section.

7 Select the **Source external to current physics** check box.

Friction 1

1 In the **Physics** toolbar, click **Attributes** and choose **Friction**.

2 In the **Settings** window for **Friction**, locate the **Friction Parameters** section.

3 In the μ text field, type `0.3`.

Contact 2

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

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

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 2 (lower)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Surface** section.

7 Select the **Source external to current physics** check box.

Friction 1

1 In the **Physics** toolbar, click **Attributes** and choose **Friction**.

2 In the **Settings** window for **Friction**, locate the **Friction Parameters** section.

3 In the μ text field, type `0.3`.

Add an adhesion condition to model the glue layer at the bottom of the seal.

Contact 2

In the **Model Builder** window, click **Contact 2**.

Adhesion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Adhesion**.

2 In the **Settings** window for **Adhesion**, locate the **Adhesive Activation** section.

3 From the **Activation criterion** list, choose **User defined**.

4 In the text field, type `dom==4`.

5 Locate the **Adhesive Stiffness** section. From the **Adhesive stiffness** list, choose **User defined**.

6 Specify the **k** vector as

1e10[N/m^3]	tl
2e10[N/m^3]	n

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `AreaInt` in the **Operator name** text field.

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

4 From the **Selection** list, choose **Inner seal boundary**.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
p0	0.1[MPa]	Pa	Initial air pressure
A0	123.63[mm^2]	m^2	Undeformed enclosed area
gamma	1.4		Adiabatic constant
EnclosedArea	AreaInt(-x*solid.nx)	m^2	Enclosed area
int_p	p0*((A0/EnclosedArea)^gamma-1)	Pa	Air pressure

SOLID MECHANICS (SOLID)

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

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

3 From the **Selection** list, choose **Inner seal boundary**.

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

5 In the p text field, type `int_p`.

Since large relative displacements and large deformations are expected in this model, the use of deformed geometry can give a substantial speed up of the computations.

Elastic Predeformation 1

In the **Physics** toolbar, click **Domains** and choose **Elastic Predeformation**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Model parameters	C10	0.37[MPa]	Pa	Mooney-Rivlin

Property	Variable	Value	Unit	Property group
Model parameters	C01	0.11[MPa]	Pa	Mooney-Rivlin
Density	rho	1100[kg/m^3]	kg/m³	Basic

MESH 1

Edge 1

1 In the **Mesh** toolbar, click **Edge**.

2 Select Boundaries 1–3 only.

Distribution 1

1 Right-click **Edge 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type 1.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Rigid base**.

Distribution 2

1 In the **Model Builder** window, right-click **Edge 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type 50.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Indenter**.

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size 1

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

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

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

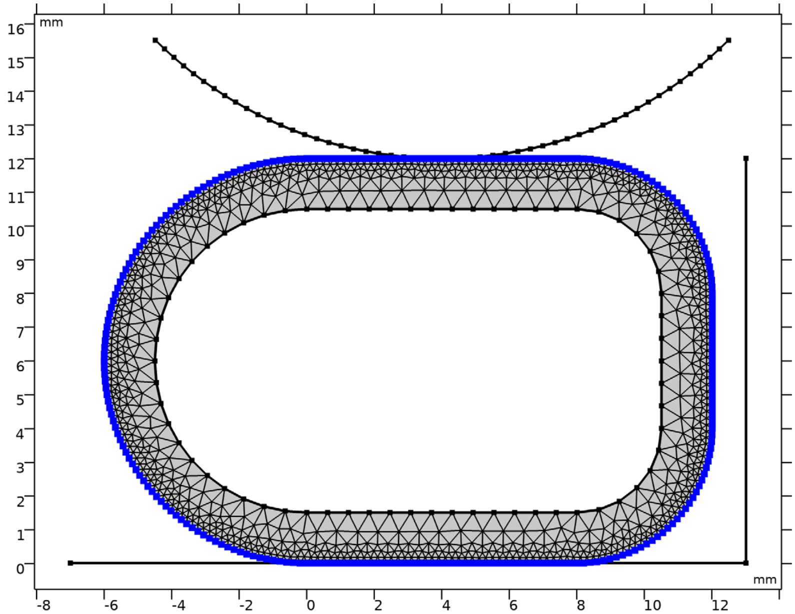
4 From the **Selection** list, choose **Outer seal boundary**.

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 text field, type 0.2.

8 Click **Build All**.



STUDY 1

Step 1: Stationary

In the first study, disable the effect of the internal pressure.

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Boundary Load 1**.

5 Click **Disable**.

Set up an auxiliary continuation sweep for the `para` parameter. Start at a nonzero value to avoid ill-conditioning during initiation of the contact.

1 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

2 Click **Add**.

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Vertical displacement parameter)	`1e-3 range(0.1,0.1,4)`	

4 In the **Model Builder** window, click **Study 1**.

5 In the **Settings** window for **Study**, type `Study: Without Pressure` in the **Label** text field.

6 In the **Study** toolbar, click **Get Initial Value**.

Solver Configurations

In the **Model Builder** window, expand the **Study: Without Pressure>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study: Without Pressure>Solver Configurations>Solution 1 (sol1)** node.

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.

2 In the **Model Builder** window, expand the **Study: Without Pressure>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

3 In the **Settings** window for **Field**, locate the **Scaling** section.

4 In the **Scale** text field, type `1e-4`.

Change the scale for the auxiliary pressure to account for the material properties of the seal made of soft rubber.

5 In the **Model Builder** window, click **Auxiliary pressure (comp1.solid.pw)**.

6 In the **Settings** window for **Field**, locate the **Scaling** section.

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

Step 1: Stationary

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

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

3 Select the **Plot** check box.

RESULTS

Stress, without Pressure

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Stress, without Pressure` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

STUDY: WITHOUT PRESSURE

In the **Study** toolbar, click **Compute**.

RESULTS

Stress, without Pressure

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

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 bottom of the seal.

Contact Pressure without Confined Air

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Contact Pressure without Confined Air` in the **Label** text field.

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

4 In the **Parameter indices (1-41)** text field, type `range(10,5,40)`.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

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

Line Graph 1

1 Right-click **Contact Pressure without Confined Air** and choose **Line Graph**.

2 Select Boundaries 7, 11, and 17 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Contact>Contact 1>solid.cnt1.Tn - Contact pressure - N/m²**.

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

5 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

6 In the **Contact Pressure without Confined Air** toolbar, click **Plot**.

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

Now you can compute the solution including the internal pressure.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

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

2 Select the **Auxiliary sweep** check box.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Vertical displacement parameter)	`1e-3 range(0.1,0.1,4)`	

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for **Study**, type `Study: With Pressure` in the **Label** text field.

Use **Get Inital Value** to get the default plots generated, so that you can select the correct plot for **Results While Solving**.

7 In the **Study** toolbar, click **Get Initial Value**.

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

9 In the **Settings** window for **Stationary**, locate the **Results While Solving** section.

10 Select the **Plot** check box.

11 From the **Plot group** list, choose **Stress (solid)**.

Solver Configurations

In the **Model Builder** window, expand the **Study: With Pressure>Solver Configurations** node.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study: With Pressure>Solver Configurations> Solution 2 (sol2)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

2 In the **Settings** window for **Field**, locate the **Scaling** section.

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

4 In the **Model Builder** window, click **Auxiliary pressure (comp1.solid.pw)**.

5 In the **Settings** window for **Field**, locate the **Scaling** section.

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

RESULTS

Stress, with Pressure

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Stress, with Pressure` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

4 In the **Study** toolbar, click **Compute**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

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

Contact Pressure with Confined Air

1 In the **Model Builder** window, right-click **Contact Pressure without Confined Air** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Contact Pressure with Confined Air` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: With Pressure/ Solution 2 (sol2)**.

4 In the **Contact Pressure with Confined Air** toolbar, click **Plot**.

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

Compressive Force vs. Indentation

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Compressive Force vs. Indentation` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

Global 1

1 In the **Compressive Force vs. Indentation** toolbar, click **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`-solid.cnt1.T_toty/d`	`N/m`	`Without confined air`

4 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: With Pressure/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`-solid.cnt1.T_toty/d`	`N/m`	`With confined air`

Compressive Force vs. Indentation

1 In the **Model Builder** window, click **Compressive Force vs. Indentation**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

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

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

5 In the **x-axis label** text field, type `Indentation (mm)`.

6 In the **y-axis label** text field, type `Force (N/mm)`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Compare with the plot shown in Figure 6.