

# Nonlinear Structural Materials Module

Application Library Manual

# Nonlinear Structural Materials Application Library

© 1998–2017 COMSOL

Protected by patents listed on [www.comsol.com/patents](http://www.comsol.com/patents), and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement ([www.comsol.com/comsol-license-agreement](http://www.comsol.com/comsol-license-agreement)) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see [www.comsol.com/trademarks](http://www.comsol.com/trademarks).

Version: COMSOL 5.3a

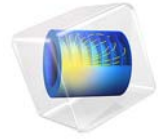
## Contact Information

Visit the Contact COMSOL page at [www.comsol.com/contact](http://www.comsol.com/contact) to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at [www.comsol.com/contact/offices](http://www.comsol.com/contact/offices) for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at [www.comsol.com/support/case](http://www.comsol.com/support/case). Other useful links include:

- Support Center: [www.comsol.com/support](http://www.comsol.com/support)
- Product Download: [www.comsol.com/product-download](http://www.comsol.com/product-download)
- Product Updates: [www.comsol.com/support/updates](http://www.comsol.com/support/updates)
- COMSOL Blog: [www.comsol.com/blogs](http://www.comsol.com/blogs)
- Discussion Forum: [www.comsol.com/community](http://www.comsol.com/community)
- Events: [www.comsol.com/events](http://www.comsol.com/events)
- COMSOL Video Gallery: [www.comsol.com/video](http://www.comsol.com/video)
- Support Knowledge Base: [www.comsol.com/support/knowledgebase](http://www.comsol.com/support/knowledgebase)

Part number: CM022902



# Arterial Wall Mechanics

## *Introduction*

---

Arteries are blood vessels that carry freshly oxygenated blood from the heart throughout the rest of the body. They are layered structures with the intima inside, followed by the media and the adventitia. The two outer layers are predominantly responsible for the mechanical behavior of healthy arteries. Both layers are made of collagenous soft tissues that show prominent strain stiffening. Families of collagen fibers give each layer anisotropic properties. These fiber reinforced structures enable blood vessels to sustain large elastic deformation.

The Holzapfel-Gasser-Ogden (HGO) constitutive model described in [Ref. 1](#) captures the anisotropic nonlinear mechanical response observed in excised artery experiments.

This model demonstrates how this hyperelastic material is implemented in COMSOL Multiphysics, and the results are compared to those reported in [Ref. 1](#).

## *Model Definition*

---

The model geometry represents a sector of a carotid artery from a rabbit. Following [Ref. 1](#), the media and adventitia are modeled as a layered cylindrical tube. Model symmetry allows the reduction of the tube to the  $10^\circ$  sector shown in [Figure 1](#).

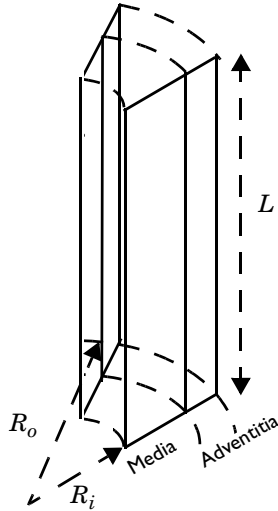


Figure 1: Carotid artery section of  $L = 2.5$  mm in length. The inner radius  $R_i$  is 0.71 mm, the outer radius  $R_o$  is 1.1 mm, the media thickness is 0.26 mm, and the adventitia thickness is 0.13 mm.

Typical mechanical experiments measure the response of arterial sections subject to combined axial stretch and internal blood pressure. The set of boundary conditions replicate these experiments.

Roller boundary conditions allow the bottom end of the artery to freely expand in the radial direction. On the top surface, prescribed displacements in the axial direction account for the axial stretching. Internal pressure is applied with a pressure boundary load on the inner surface.

This model considers axial stretches between 1.5 and 1.9 and internal pressures between 0 and 160 mmHg. The mechanical response in this range is highly nonlinear resulting in large elastic deformations, and it is mathematically described within the theory of hyperelasticity.

The HGO model is a nearly incompressible anisotropic hyperelastic material model defined by a strain energy density function of the form

$$W_s = W_1 + W_4 + W_6 + W_{vol} \quad (1)$$

The four terms on the right-hand side of [Equation 1](#) depend on invariants of the right Cauchy-Green tensor.

The first term describes the mechanical behavior of the elastic ground substance. The isotropic function  $W_1$  depends on one material parameter and the first isochoric invariant  $I_1(\overline{C}_{el})$ , defined in the same fashion as for the neo-Hookean material (see [Ref. 3](#) for more details on this invariant)

$$W_1 = \frac{c}{2}(I_1(\overline{C}_{el}) - 3)$$

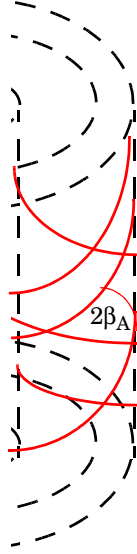
The second and third terms on the right-hand side of [Equation 1](#) describe the mechanical contribution of the collagen fiber network. Following [Ref. 1](#), these expressions are written as

$$W_4 = \frac{k_1}{2k_2}(e^{k_2(I_4 - 1)^2} - 1)$$

$$W_6 = \frac{k_1}{2k_2}(e^{k_2(I_6 - 1)^2} - 1)$$

here, the fiber network is reduced to two families of fibers with material properties  $k_1$  and  $k_2$ .

The deformation of each fiber family is measured by the invariants  $I_4$  and  $I_6$ . You can find a detailed background for this formulation in [Ref. 1](#) and [Ref. 2](#).



*Figure 2: Angle between the two fiber families in the Adventitia.*

Briefly, a family  $i$  of fibers is defined by a vector field  $\mathbf{a}_{0i}$  in the undeformed direction. The fibers deform under the action of the isochoric deformation gradient, so that  $\overline{\mathbf{F}}_{el} \cdot \mathbf{a}_{0i}$  is the deformed fiber configuration. The length of  $\overline{\mathbf{F}}_{el} \cdot \mathbf{a}_{0i}$  is the fiber stretch, to be used for the constitutive equations.

The HGO model uses the square of the fiber stretches computed according to the invariants  $I_4$  and  $I_6$

$$I_4 = I_4(\overline{\mathbf{C}}_{el}, \mathbf{a}_{01}) = (\overline{\mathbf{F}}_{el} \cdot \mathbf{a}_{01}) \cdot (\overline{\mathbf{F}}_{el} \cdot \mathbf{a}_{01}) = \mathbf{a}_{01} \cdot \overline{\mathbf{C}}_{el} \cdot \mathbf{a}_{01}$$

$$I_6 = I_6(\overline{\mathbf{C}}_{el}, \mathbf{a}_{02}) = (\overline{\mathbf{F}}_{el} \cdot \mathbf{a}_{02}) \cdot (\overline{\mathbf{F}}_{el} \cdot \mathbf{a}_{02}) = \mathbf{a}_{02} \cdot \overline{\mathbf{C}}_{el} \cdot \mathbf{a}_{02}$$

Also, the angle  $\beta$  is the relative angle between  $\mathbf{a}_{01}$  and  $\mathbf{a}_{02}$ .

The mechanical properties of both the media and adventitia are governed by these expressions. Each layer has a distinct set of material parameters  $c$ ,  $k_1$ ,  $k_2$ , and the initial fiber directions  $\mathbf{a}_{01}$  and  $\mathbf{a}_{02}$  are aligned at different angles, as shown in [Figure 3](#).

The last term  $W_{vol}$  on the right-hand side of [Equation 1](#) defines the volumetric strain energy density. This is part of the mixed formulation employed to resolve nearly incompressibility, and it is defined with the initial bulk modulus  $\kappa$  and the elastic volume ratio  $J_{el}$  as

$$W_{vol} = \frac{\kappa}{2}(J_{el} - 1)^2$$

## MATERIAL

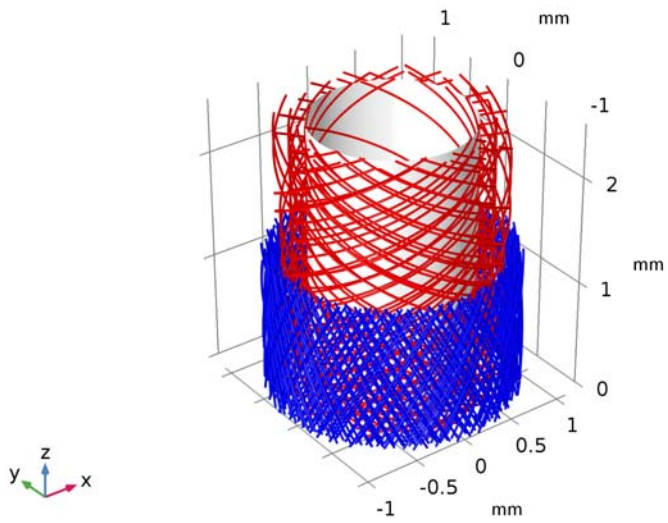
The material parameters are given in the following tables.

MATERIAL PROPERTIES IN MEDIA ( <a href="#">Ref. 1</a> )	VALUE
$c$	3 [kPa]
$k_1$	2.3632 [kPa]
$k_2$	0.8393
$\beta$	29 [deg]
$\kappa$	100 [MPa]

MATERIAL PROPERTIES IN ADVENTITIA ( <a href="#">Ref. 1</a> )	VALUE
$c$	0.3 [kPa]
$k_1$	0.5620 [kPa]
$k_2$	0.7112
$\beta$	62 [deg]
$\kappa$	1000 [MPa]

## *Results and Discussion*

The model computes the static response to the applied boundary conditions. [Figure 3](#) displays the fiber layout of both the media and the adventitia fiber families.



*Figure 3: Fiber layout in the undeformed configuration of the media (inner, red) and the adventitia (outer, blue). Note the different angles between fiber families.*

Figure 4 shows the radial stress distribution through the thickness of the wall at an axial stretch of 1.9 and internal pressure of 160 mmHg.

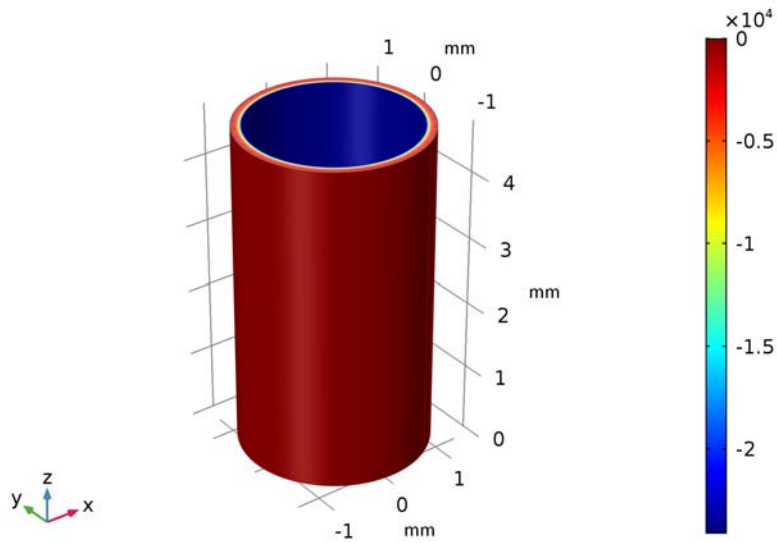


Figure 4: Radial stress distribution in the artery wall at an axial stretch of 1.9 and 160 mmHg internal pressure.

Figure 5 plots the internal pressure against the expansion of the inner radius for the entire load range. The results are in good agreements with the data reproduced from Ref. 1.

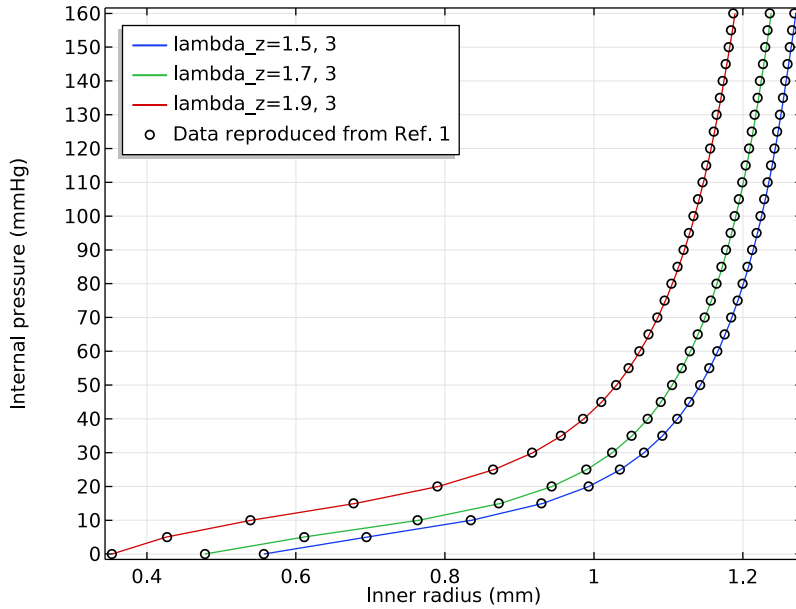


Figure 5: Plot of internal pressure vs. inner radius for three different axial stretches. Data reproduced from Ref. 1 (circles) coincides well with the model results.

## References

1. G. Holzapfel, T. Gasser, and R. Ogden, “A New Constitutive Framework for Arterial Wall Mechanics and a Comparative Study of Material Models,” *J. Elasticity*, vol. 61, pp. 1–48, 2000.
2. G. Holzapfel, *Nonlinear Solid Mechanics: A Continuum Approach for Engineering*, John Wiley & Sons, 2000.
3. *Nonlinear Structural Materials Module User’s Guide*, COMSOL Multiphysics.

## Notes About the COMSOL Implementation

The most important aspect in this model is the implementation of the HGO material model with a user-defined strain energy density function. The initial fiber directions are user-defined variables. Since the model geometry is a cylinder, use a cylindrical coordinate system as the base for the vectors’ components.

There are two fiber families in each arterial layer. The mathematical expressions in the HGO model are the same for the media as for the adventitia, except for the different material parameters. Only the expressions for the media are discussed below, denoted by the index  $M$ . Replace this index with the index  $A$  to obtain the expressions for the adventitia.

The vector field  $\mathbf{a}_{01M}$  of the first fiber family in the media is represented by the components  $a_{01M1}$  (radial),  $a_{01M2}$  (azimuthal), and  $a_{01M3}$  (axial). The second fiber family  $\mathbf{a}_{02M}$  in the media has the components  $a_{02M1}$ ,  $a_{02M2}$ , and  $a_{02M3}$ .

Each fiber family has an associated invariant computed according to [Equation 5](#). The internal variables for the isochoric right Cauchy-Green deformation tensor  $\overline{C}_{el}$  in the local coordinate system are `solid.CIelIJ` ([Ref. 3](#)), where  $I$  and  $J$  are indexes running from 1 to 3. In the **Cylindrical System**, 1 represents the radial direction, 2 the azimuthal and 3 the axial direction.

In the settings for the hyperelastic material, select the cylindrical system as a reference coordinate system. The invariant associated with the first fiber family is named `I4CIe1`, and according to [Equation 5](#) it is computed as

$$a_{01M1}^2 \text{solid.CIel11} + 2 a_{01M1} a_{01M2} \text{solid.CIel12} + a_{01M2}^2 \text{solid.CIel13} + a_{01M3}^2 \text{solid.CIel22} + 2 a_{01M2} a_{01M3} \text{solid.CIel23} + a_{01M3}^2 \text{solid.CIel33}$$

The invariant `I6CIe1` is defined similarly, but it uses the components of the second fiber family,  $a_{02M1}$ ,  $a_{02M2}$ , and  $a_{02M3}$ .

Once the fiber directions and related invariants are defined, implement the strain energy density functions. [Equation 2](#) defines an isotropic isochoric function. The corresponding invariant  $I_1(\overline{C}_{el})$  is defined by the variable `solid.I1CIe1`, thus the expression in the media reads  $c_M/2 * (\text{solid.I1CIe1} - 3)$ .

The second and third terms on the right-hand side of [Equation 1](#) are the anisotropic strain energy functions for the fiber families  $\mathbf{a}_{01M}$  and  $\mathbf{a}_{02M}$ . With the definition of the fiber invariants in the media for the fiber family  $\mathbf{a}_{01M}$ , [Equation 3](#) becomes

$$k_{1M} / (2 * k_{2M}) * (\exp(k_{2M} * (I4CIe1 - 1)^2) - 1) * (I4CIe1 > 1)$$

The last factor,  $(I4CIe1 > 1)$ , evaluates to zero if the fiber stretch is smaller than one. This means that the fibers only contribute to tensile stress. To implement [Equation 4](#), simply replace `I4CIe1` with `I6CIe1`, which corresponds to the second fiber family  $\mathbf{a}_{02M}$ .

The last term on the right-hand side of [Equation 1](#) is the volumetric strain energy density. This term is needed for the mixed formulation used in nearly incompressible hyperelasticity ([Ref. 3](#)). The initial bulk modulus  $\kappa$  is several orders of magnitude

higher than the other material constants and chosen to enforce incompressibility. With the variable for the elastic volume ratio `solid.Je1`, the expression for Equation 7 reads  $\kappa * (\text{solid.Je1} - 1)^2$ .

This model also shows how to use the Curvilinear Coordinates interface to plot the configuration of the fiber families. The interface takes care of the mapping from the cylindrical coordinate system to the global coordinates system used in the result plots.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Hyperelasticity/arterial\_wall\_mechanics

---

### *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 **3D**.
- 2** In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 3** Click **Add**.
- 4** In the **Select Physics** tree, select **Mathematics>Curvilinear Coordinates (cc)**.

Use Curvilinear Coordinates to visualize the fiber layout. Add it four times, once for each fiber family.

- 5** Click **Add**.
- 6** In the **Select Physics** tree, select **Mathematics>Curvilinear Coordinates (cc)**.
- 7** Click **Add**.
- 8** In the **Select Physics** tree, select **Mathematics>Curvilinear Coordinates (cc)**.
- 9** Click **Add**.
- 10** In the **Select Physics** tree, select **Mathematics>Curvilinear Coordinates (cc)**.
- 11** Click **Add**.
- 12** Click **Study**.
- 13** In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.

14 Click **Done**.

## GLOBAL DEFINITIONS

Load all model parameters from a file containing parameters for the geometry, material properties and boundary conditions.

### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `arterial_wall_mechanics_parameters.txt`.

Now add an interpolation function for importing pressure vs. radius data reproduced from [Ref. 1](#). Use it for comparison.

### *Interpolation 1 (int1)*

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 From the **Data source** list, choose **File**.
- 4 Click **Browse**.
- 5 Browse to the model's Application Libraries folder and double-click the file `arterial_wall_mechanics_pressure_radius.txt`.  
This file contains the data adapted from [Ref. 1](#).
- 6 In the **Number of arguments** text field, type 1.
- 7 Click **Import**.
- 8 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
hgo_pr_1_5	1
hgo_pr_1_7	2
hgo_pr_1_9	3

- 9 Locate the **Units** section. In the **Arguments** text field, type kPa.
- 10 In the **Function** text field, type mm.

## GEOMETRY 1

Construct the model geometry by first drawing two circular sections on a work plane. Then, form a difference between them and extrude it.

- 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**.

### *Work Plane 1 (wp1)*

- 1 On the **Geometry** toolbar, click **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

### *Circle 1 (c1)*

- 1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type  $R_i$ .
- 4 In the **Sector angle** text field, type `sector_angle`.
- 5 Right-click **Circle 1 (c1)** and choose **Build Selected**.

### *Circle 2 (c2)*

- 1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type  $R_o$ .
- 4 In the **Sector angle** text field, type `sector_angle`.
- 5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	HA

- 6 Right-click **Circle 2 (c2)** and choose **Build Selected**.

### *Difference 1 (dif1)*

- 1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **c2** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 5 Select the object **c1** only.
- 6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

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

*Work Plane 1 (wp1)*

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

*Extrude 1 (ext1)*

- 1 On the **Geometry** toolbar, click **Extrude**.
- 2 In the **Settings** window for **Extrude**, locate the **Distances** section.
- 3 In the table, enter the following settings:

<b>Distances (mm)</b>
L

- 4 Right-click **Extrude 1 (ext1)** and choose **Build Selected**.
- 5 Click the **Zoom Extents** button on the **Graphics** toolbar.

## DEFINITIONS

Add a cylindrical coordinate system aligned with the artery geometry. It is easier to define the components of the fiber directions in this local coordinate system. Also, use this coordinate system as a local system for the hyperelastic material.

*Cylindrical System 2 (sys2)*

- 1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.
- 2 In the **Settings** window for **Cylindrical System**, locate the **Settings** section.
- 3 From the **Frame** list, choose **Material (X, Y, Z)**.
- 4 Find the **Coordinate names** subsection. In the table, enter the following settings:

<b>First</b>	<b>Second</b>	<b>Third</b>
R	PHI	A

Load all model variables from files. These define the initial directions of all fiber families. The files also contain the expressions for the user defined strain energy density functions.

*Variables 1*

- 1 On the **Definitions** toolbar, click **Local Variables**.
- 2 In the **Settings** window for **Variables**, type Fiber Directions Media in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

- 4 Select Domain 1 only.
- 5 Locate the **Variables** section. Click **Load from File**.
- 6 Browse to the model's Application Libraries folder and double-click the file `arterial_wall_mechanics_fiber_directions_media.txt`.

#### *Variables 2*

- 1 On the **Definitions** toolbar, click **Local Variables**.
- 2 In the **Settings** window for **Variables**, type Fiber Directions Adventitia in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.
- 5 Locate the **Variables** section. Click **Load from File**.
- 6 Browse to the model's Application Libraries folder and double-click the file `arterial_wall_mechanics_fiber_directions_adventitia.txt`.

#### *Variables 3*

- 1 On the **Definitions** toolbar, click **Local Variables**.
- 2 In the **Settings** window for **Variables**, type Strain Energy Density Media in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 1 only.
- 5 Locate the **Variables** section. Click **Load from File**.
- 6 Browse to the model's Application Libraries folder and double-click the file `arterial_wall_mechanics_strain_energy_density_media.txt`.

#### *Variables 4*

- 1 On the **Definitions** toolbar, click **Local Variables**.
- 2 In the **Settings** window for **Variables**, type Strain Energy Density Adventitia in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.
- 5 Locate the **Variables** section. Click **Load from File**.

- 6 Browse to the model's Application Libraries folder and double-click the file `arterial_wall_mechanics_strain_energy_density_adventitia.txt`.

## **SOLID MECHANICS (SOLID)**

### *Hyperelastic Material 1*

- 1 On 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 **Coordinate System Selection** section. From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

The cylindrical coordinate system is now the local coordinate system for the hyperelastic material.

- 5 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **User defined**.
- 6 Select the **Nearly incompressible material** check box.  
The HGO model is incompressible, select the check box to apply a mixed formulation.
- 7 In the  $W_{\text{iso}}$  text field, type  $W1+W4+W6$ .  
This is the isochoric part of the strain energy density function. The functions  $W1$ ,  $W4$  and  $W6$  are defined with different properties in the media and adventitia.
- 8 In the  $W_{\text{vol}}$  text field, type  $Wv01$ .  
This is the volumetric strain energy density that enforces incompressibility.
- 9 From the  $\rho$  list, choose **User defined**. In the associated text field, type 1100.
- 10 In the **Model Builder** window, click **Solid Mechanics (solid)**.
- 11 In the **Settings** window for **Solid Mechanics**, locate the **Structural Transient Behavior** section.
- 12 From the list, choose **Quasi-static**.

### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 2, 3, 5, 7, 8, and 10 only.

### *Prescribed Displacement 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
- 2 Select Boundaries 4 and 9 only.

- 3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
- 4 Select the **Prescribed in z direction** check box.
- 5 In the  $u_{0z}$  text field, type  $(\lambda_z - 1) * L$ .  
Use  $\lambda_z$  as a continuation parameter in the solver to vary axial stretch.

#### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 From the **Load type** list, choose **Pressure**.
- 5 In the  $p$  text field, type  $p_i$ .  
Use  $p_i$  as a continuation parameter in the solver settings to vary the internal pressure.

#### **CURVILINEAR COORDINATES (CC)**

Now set up the curvilinear coordinates for all fiber families by adding a user defined vector field in the cylindrical system for the components of the fiber directions.

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Curvilinear Coordinates (cc)**.
- 2 Select Domain 1 only.

#### *User Defined 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **User Defined**.
- 2 In the **Settings** window for **User Defined**, locate the **User Defined** section.
- 3 From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.  
Select the cylindrical coordinate system as the reference system for the fiber directions.  
Now set the components of the vector field to the cylindrical components of the first fiber family.

- 4 Specify the  $\mathbf{u}$  vector as

a01M1	R
a01M2	PHI
a01M3	A

5 Set up the other curvilinear coordinates similarly according to the table:

	<b>Curvilinear Coordinates (cc)</b>	<b>Curvilinear Coordinates 2 (cc2)</b>	<b>Curvilinear Coordinates 3 (cc3)</b>	<b>Curvilinear Coordinates 4 (cc4)</b>
Domain selection	1	1	2	2
Coordinate system	Cylindrical System 2 (sys2)	Cylindrical System 2 (sys2)	Cylindrical System 2 (sys2)	Cylindrical System 2 (sys2)
<b>u</b> vector, R component	a01M1	a02M1	a01A1	a02A1
<b>u</b> vector, PHI component	a01M2	a02M2	a01A2	a02A2
<b>u</b> vector, A component	a01M3	a02M3	a01A3	a02A3

## MESH I

### *Mapped I*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh I** and choose **More Operations>Mapped**.

2 Select Boundaries 3 and 8 only.

### *Size*

1 In the **Model Builder** window, right-click **Mesh I** and choose **Swept**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.

## STUDY I

Now set up a study to compute the static response of the artery segment subject to combined axial stretch and internal pressure.

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

2 Clear the **Generate default plots** check box.

You will not need the default plots in this model.

### *Step 1: Stationary*

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

- 2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.
- 3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 4 Click **Add**.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
lambda_z (Axial stretch)	1.5 1.7 1.9	

The parameter lambda\_z controls the axial stretch.

- 6 Click **Add**.
- 7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_i (Internal pressure)	range(0, 5, 160)	mmHg

Use p\_i to vary the internal pressure from 0 to 160 mmHg with steps of 5 mmHg.

- 8 From the **Sweep type** list, choose **All combinations**.
- 9 From the **Reuse solution from previous step** list, choose **Auto**.  
Using the **Auto** option for **Reuse solution from previous step** is suitable for this kind of multiparameter sweep with continuation.

#### *Solution 1 (sol1)*

- 1 On the **Study** toolbar, click **Show Default Solver**.  
Using constant prediction for the continuation sweep improves convergence when the solution is very nonlinear in the swept parameter.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
- 3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.
- 4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
- 5 From the **Predictor** list, choose **Constant**.
- 6 On the **Study** toolbar, click **Compute**.

## **RESULTS**

In the **Model Builder** window, expand the **Results** node.

### *Data Sets*

Before you examine the results, create a Sector 3D data set. This data set creates the full cylindrical geometry from the section that was used for the computation.

#### *Study 1/Solution 1 (sol1)*

- 1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/Solution 1 (sol1)**.
- 2 In the **Settings** window for **Solution**, locate the **Solution** section.
- 3 From the **Frame** list, choose **Material (X, Y, Z)**.

#### *Sector 3D 1*

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Sector 3D**.
- 2 In the **Settings** window for **Sector 3D**, locate the **Symmetry** section.
- 3 In the **Number of sectors** text field, type 36.

#### *Study 1/Solution 1 (sol1)*

Now duplicate the data sets and add a selection. Use these in one of the plots below.

#### *Selection*

- 1 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.
- 2 On the **Results** toolbar, click **Selection**.
- 3 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
- 4 From the **Geometric entity level** list, choose **Boundary**.
- 5 Select Boundary 1 only.

#### *Sector 3D 2*

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Sector 3D**.
- 2 In the **Settings** window for **Sector 3D**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.
- 4 Locate the **Symmetry** section. In the **Number of sectors** text field, type 36.

### *Data Sets*

In the **Model Builder** window, collapse the **Results>Data Sets** node.

Create a 3D plot group for the radial stress distribution.

#### *3D Plot Group 1*

- 1 On the **Results** toolbar, click **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Radial Stress in the **Label** text field.

- 3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 4 Locate the **Data** section. From the **Data set** list, choose **Sector 3D 1**.

#### *Surface 1*

- 1 Right-click **Radial Stress** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Stress (Gauss points)>Stress tensor, local coordinate system, Gauss-point evaluation>solid.sIgp11 - Stress tensor, local coordinate system, Gauss-point evaluation, 11 component**.

Remember that you have set the local coordinate system of the hyperelastic material to the cylindrical system. The local 11 stress component is therefore the radial component.

#### *Deformation 1*

- 1 Right-click **Results>Radial Stress>Surface 1** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 Select the **Scale factor** check box.
- 4 In the associated text field, type 1.

#### *Radial Stress*

- 1 In the **Model Builder** window, under **Results** click **Radial Stress**.
- 2 On the **Radial Stress** toolbar, click **Plot**.
- 3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Create a 1D plot group to compare the pressure vs radius relationship to the data reproduced from [Ref. 1](#).

#### *1D Plot Group 2*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
- 2 In the **Settings** window for **1D Plot Group**, type Pressure vs. Radius in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 5 In the associated text field, type Inner radius (mm).
- 6 Select the **y-axis label** check box.
- 7 In the associated text field, type Internal pressure (mmHg).
- 8 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

### Point Graph 1

- 1 Right-click **Pressure vs. Radius** and choose **Point Graph**.
- 2 Select Point 3 only.
- 3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type  $p_i$ .
- 5 In the **Unit** field, type mmHg.
- 6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 7 In the **Expression** text field, type  $R_i+u$ .

This expression computes the radius as a function of internal pressure.  $R_i$  is the initial radius, and the dependent variable  $u$  represents the displacement in the x direction, which for point 3 is also the radial direction.

- 8 On the **Pressure vs. Radius** toolbar, click **Plot**.
- 9 Click to expand the **Legends** section. Select the **Show legends** check box.

### Global 1

- 1 In the **Model Builder** window, under **Results** right-click **Pressure vs. Radius** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study 1/Solution 1 (1) (sol1)**.
- 4 From the **Parameter selection (lambda\_z)** list, choose **Last**.
- 5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
$p_i$	mmHg	Internal pressure

- 6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 7 In the **Expression** text field, type  $hgo\_pr\_1\_5(p_i)$ .

This is the interpolation function with data reproduced from [Ref. 1](#). It returns the inner radius as a function of internal pressure at an axial stretch of 1.5.
- 8 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 9 From the **Color** list, choose **Black**.
- 10 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
- 11 From the **Positioning** list, choose **In data points**.
- 12 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

**B3** In the table, enter the following settings:

---

### Legends

---

Data reproduced from Ref. 1

---

#### Global 2

- 1 Right-click **Results>Pressure vs. Radius>Global 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.
- 3 In the **Expression** text field, type  $\text{hgo\_pr\_1\_7}(p\_i)$ .  
This is the inner radius as a function of internal pressure at an axial stretch of 1.7.
- 4 Click to expand the **Legends** section. Clear the **Show legends** check box.

#### Global 3

- 1 Right-click **Results>Pressure vs. Radius>Global 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.
- 3 In the **Expression** text field, type  $\text{hgo\_pr\_1\_9}(p\_i)$ .  
This is the inner radius as a function of internal pressure at an axial stretch of 1.9.
- 4 On the **Pressure vs. Radius** toolbar, click **Plot**.

Create a 3D plot group to display the fiber layout.

#### 3D Plot Group 3

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type **Fiber Directions** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Sector 3D 1**.
- 4 Locate the **Title** section. From the **Title type** list, choose **None**.
- 5 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

#### Streamline 1

- 1 Right-click **Fiber Directions** and choose **Streamline**.
- 2 In the **Settings** window for **Streamline**, locate the **Expression** section.
- 3 In the **X component** text field, type  $cc.vX$ .
- 4 In the **Y component** text field, type  $cc.vY$ .
- 5 In the **Z component** text field, type  $cc.vZ$ .  
This is the vector field of the first Curvilinear Coordinate physics and corresponds to the first fiber family in the media.

- 6 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.
- 7 In the **Separating distance** text field, type 0.05.
- 8 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.
- 9 In the **Tube radius expression** text field, type 0.1.

#### *Streamline 2*

- 1 Right-click **Results>Fiber Directions>Streamline 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Streamline**, locate the **Expression** section.
- 3 In the **X component** text field, type  $cc2.vX$ .
- 4 In the **Y component** text field, type  $cc2.vY$ .
- 5 In the **Z component** text field, type  $cc2.vZ$ .

This field is the second fiber family in the media.

#### *Streamline 3*

- 1 Right-click **Results>Fiber Directions>Streamline 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Streamline**, locate the **Expression** section.
- 3 In the **X component** text field, type  $cc3.vX*(Z<L/2)$ .
- 4 In the **Y component** text field, type  $cc3.vY*(Z<L/2)$ .
- 5 In the **Z component** text field, type  $cc3.vZ*(Z<L/2)$ .
- 6 Locate the **Streamline Positioning** section. In the **Separating distance** text field, type 0.02.
- 7 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

This field is the first fiber family in the adventitia. The boolean expression restricts the plotting of fibers to the half of the geometry.

#### *Streamline 4*

- 1 Right-click **Results>Fiber Directions>Streamline 3** and choose **Duplicate**.
- 2 In the **Settings** window for **Streamline**, locate the **Expression** section.
- 3 In the **X component** text field, type  $cc4.vX*(Z<L/2)$ .
- 4 In the **Y component** text field, type  $cc4.vY*(Z<L/2)$ .
- 5 In the **Z component** text field, type  $cc4.vZ*(Z<L/2)$ .

This is the second fiber family in the adventitia.

#### *Surface 1*

- 1 In the **Model Builder** window, under **Results** right-click **Fiber Directions** and choose **Surface**.

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

**3** From the **Data set** list, choose **Sector 3D 2**.

The surface plot uses the data set with the selection of the inner surface. Use it to display this boundary in the plot.

**4** Locate the **Expression** section. In the **Expression** text field, type 1.

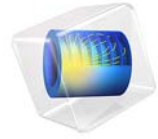
**5** Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

**6** From the **Color** list, choose **White**.

**7** On the **Fiber Directions** toolbar, click **Plot**.

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





# Arterial Wall Viscoelasticity

## Introduction

---

Arteries are blood vessels that carry freshly oxygenated blood from the heart throughout the rest of the body. The Holzapfel-Gasser-Ogden (HGO) model proposes a mechanical description of the young healthy arteries in Ref. 1 based on anisotropic hyperelastic properties, which is implemented in the [Arterial Wall Mechanics](#) example. Here we study the dynamic behavior of the artery, based on Ref. 2 and calculate the time-dependent response given a sudden axial stretching.

## Model Definition

---

The geometry, physics interface, and material models are the same as in the example [Arterial Wall Mechanics](#) (Figure 1).

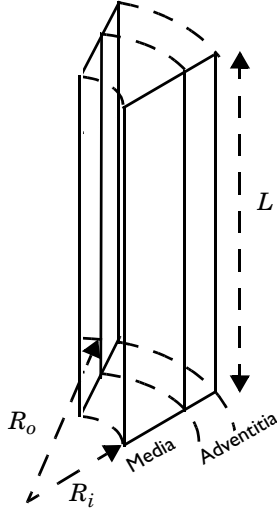


Figure 1: Carotid artery section, made of media and adventitia layers

The two modeled layers (media and adventitia) are described by the anisotropic HGO hyperelastic material model:

$$W_s = W_1 + W_4 + W_6 + W_{\text{vol}} = W_{\text{iso}} + W_{\text{vol}}$$

$$S = S_{\text{iso}} + S_{\text{vol}} = \frac{\partial W_{\text{iso}}}{\partial \epsilon} + \frac{\partial W_{\text{vol}}}{\partial \epsilon}$$

Only the media layer includes viscoelastic behavior. The generalized Maxwell viscoelastic model is used to represent relaxations time at different time-scales (Ref. 2):

$$S = S_{\text{iso}} + S_{\text{vol}} + \sum_m Q_m \quad (1)$$

For each branch of the generalized Maxwell model the viscoelastic stress follows the equation:

$$\dot{Q}_m + \frac{Q_m}{\tau_m} = \beta_m \dot{S}_{\text{iso}}$$

here,  $\tau_m$  is the relaxation time and  $\beta_m$  is the energy factor per branch.

Applying the variable change  $q_m = \beta_m S_{\text{iso}} - Q_m$  :we obtain

$$\tau_m \dot{q}_m + q_m = \beta_m S_{\text{iso}} \quad (2)$$

In this example a five-branches generalized Maxwell viscoelastic model is used with the following values taken from Ref. 2:

BRANCH	ENERGY FACTOR	RELAXATION TIME
1	0.3353	0.001 [s]
2	0.286	0.01 [s]
3	0.298	0.1 [s]
4	0.285	1 [s]
5	0.348	10 [s]

The artery is first loaded with an internal pressure of 100 mmHg and an initial axial stretch of 1.5. After initialization the stretch is increased to 1.7 and the viscoelastic relaxation is calculated.

## Results and Discussion

The total force is computed by integrating the axial stress on the top section surface. The plot shown in [Figure 2](#) is similar to the force relaxation in [Ref. 2](#). The force relaxes almost constantly from  $10^{-3}$  s to 10 s due to the wide range of relaxation times.

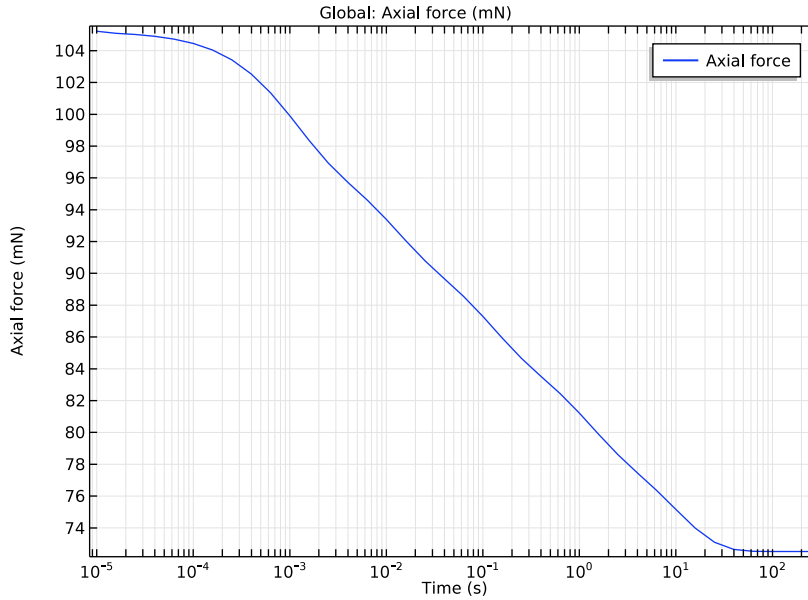


Figure 2: Axial force relaxation after stretching

[Figure 3](#) shows the evolution of total viscoelastic stress and viscoelastic stress for each branch over time. The viscoelastic stress for the branches is calculated with the expression  $Q_m = \beta_m S_{iso} - q_m$ . The viscoelastic stress relaxes from its initial value to near zero value.

It is not exactly zero because of the interpolation function. Refining the mesh in axial direction gives better results.

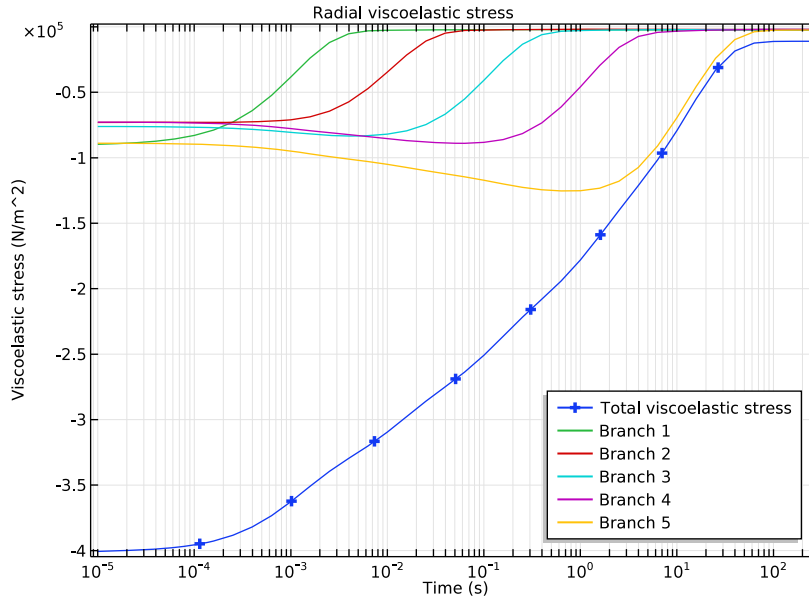


Figure 3: Viscoelastic stress, total and in each branch

### Notes About the COMSOL Implementation

A stationary study step is needed to prestress the artery with initial pressure and stretch. As this initial state is assumed to be a steady-state, the static stiffness option must be set to Long-term. This ensures that the viscoelasticity node has no effects in the stationary step.

The stress gradient is high in the media layer. The viscoelastic auxiliary stress  $q_m$  calculated in Equation 2 is discretized with linear discontinuous Lagrange shape functions, whereas the stress  $S_{iso}$  is discretized with quadratic functions. To provide accurate results, the mesh must be refined in radial direction. At the same time, the mesh size in axial direction is decreased to limit the increase in computing time and memory.

## References

---

1. G. Holzapfel, T. Gasser, and R. Ogden, “A New Constitutive Framework for Arterial Wall Mechanics and a Comparative Study of Material Models,” *J. Elasticity*, vol. 61, pp. 1–48, 2000.
2. G.A. Holzapfel, T.C. Gasser, M. Stadler, “A Structural Model for the Viscoelastic Behavior of Arterial Walls: Continuum Formulation and Finite Element Analysis”, *European Journal of Mechanics A/Solid*, vol.21, pp. 441–463, 2002

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Viscoelasticity/arterial\_wall\_viscoelasticity

---

## Modeling Instructions

---

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

From the Application Libraries root, browse to the folder Nonlinear\_Structural\_Materials\_Module/Hyperelasticity and double-click the file arterial\_wall\_mechanics.mph.

### GLOBAL DEFINITIONS

#### Parameters

Set the parameters used for loads and constraints for the time dependent study.  $t$  is the parameter for time, it is needed to define the prescribed displacement in the stationary study step.

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
p_i	100 [mmHg]	13332 Pa	Internal pressure
lambda_z	1.7	1.7	Axial stretch
lambda_z0	1.5	1.5	Initial axial stretch
t	0 [s]	0 s	Time

Create a step function to apply the stretch for time-dependent study.

*Step 1 (step 1)*

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Step**.
- 2 In the **Settings** window for **Step**, type  $\lambda$  in the **Function name** text field.
- 3 Locate the **Parameters** section. In the **Location** text field, type  $5e-6$ .
- 4 In the **From** text field, type  $\lambda_{z0}$ .
- 5 In the **To** text field, type  $\lambda_z$ .
- 6 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type  $1e-5$ .
- 7 Click **Plot**.

**COMPONENT 1 (COMP1)**

In the **Model Builder** window, expand the **Component 1 (comp1)** node.

**SOLID MECHANICS (SOLID)**

*Hyperelastic Material 1*

Add a generalized Maxwell viscoelasticity model according to [Ref. 1](#).

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then click **Hyperelastic Material 1**.

*Viscoelasticity 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Viscoelasticity**.
- 2 Select Domain 1 only.
- 3 In the **Settings** window for **Viscoelasticity**, locate the **Viscoelasticity Model** section.
- 4 Click **Add** four times.
- 5 In the table, enter the following settings:

Branch	Energy factor (I)	Relaxation time (s)
1	0.353	0.001
2	0.286	0.01
3	0.298	0.1
4	0.285	1
5	0.348	10

### *Prescribed Displacement 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Prescribed Displacement 1**.
- 2 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
- 3 In the  $u_{0z}$  text field, type  $(\lambda(\tau[1/s]) - 1) * L$ .

## **DEFINITIONS**

### *Integration 1 (intop1)*

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 4 and 9 only.
- 5 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

The stress gradient is high near the inner wall. Refine the mesh in radial direction in order to interpolate the degrees of freedom more accurately.

## **MESH 1**

### *Size*

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1** node, then click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Normal**.

### *Distribution 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 1** and choose **Distribution**.
- 2 Select Edge 7 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 20.

### *Distribution 2*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Edge 15 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 10.

#### *Distribution 3*

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

2 Select Edge 2 only.

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

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

5 Click **Build All**.

Add a new study. The first stationary step is used to prestress the artery with stretch and internal pressure. Long-term stiffness is selected in the **Viscoelasticity** node, so the viscoelastic effect is disabled. The time-dependent step is used to compute the dynamic reaction to an additional stretch.

#### **ADD STUDY**

1 On 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 **Preset Studies>Stationary**.

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

#### **STUDY 2**

##### *Step 1: Stationary*

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

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

3 In the table, enter the following settings:

<b>Physics interface</b>	<b>Solve for</b>	<b>Discretization</b>
Curvilinear Coordinates (cc)		physics
Curvilinear Coordinates 2 (cc2)		physics
Curvilinear Coordinates 3 (cc3)		physics
Curvilinear Coordinates 4 (cc4)		physics

4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_i (Internal pressure)	range (0, 10, 100)	mmHg

*Step 2: Time Dependent*

- 1 On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.
- 3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Curvilinear Coordinates (cc)		physics
Curvilinear Coordinates 2 (cc2)		physics
Curvilinear Coordinates 3 (cc3)		physics
Curvilinear Coordinates 4 (cc4)		physics

- 4 Locate the **Study Settings** section. In the **Times** text field, type range (0, 1e-6, 9e-6)  $10^{\{\text{range}(-5, 0.2, 2.4)\}}$ .
- 5 From the **Tolerance** list, choose **User controlled**.
- 6 In the **Relative tolerance** text field, type 0.001.
- 7 In the **Model Builder** window, click **Step 2: Time Dependent**.
- 8 In the **Settings** window for **Time Dependent**, click to expand the **Values of dependent variables** section.
- 9 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 10 From the **Method** list, choose **Solution**.
- 11 From the **Study** list, choose **Study 2, Stationary**.
- 12 From the **Selection** list, choose **Last**.

*Solution 2 (sol2)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver I**.
- 3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

- 4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.
- 5 On the **Study** toolbar, click **Compute**.

## RESULTS

### *Study 2/Solution 2 (sol2)*

- 1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 2/Solution 2 (sol2)**.
- 2 In the **Settings** window for **Solution**, locate the **Solution** section.
- 3 From the **Frame** list, choose **Material (X, Y, Z)**.

### *Stress (solid)*

In the **Model Builder** window, expand the **Stress (solid)** node.

### *Surface 1*

Integrate the reaction force on the top surfaces to calculate the reaction force and reproduce [Figure 2](#).

### *ID Plot Group 5*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Force Relaxation in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
- 4 From the **Time selection** list, choose **Interpolated**.
- 5 In the **Times (s)** text field, type  $10^{\{\text{range}(-5, 0.2, 2.4)\}}$ .
- 6 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.
- 7 On the **Force Relaxation** toolbar, click **Plot**.

### *Global 1*

- 1 Right-click **Force Relaxation** and choose **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
$36 \cdot \text{intop1}(\text{solid.RFz})$	mN	Axial force

The operator is multiplied by 36 because the geometry is only a  $10^\circ$  sector.

- 4 On the **Force Relaxation** toolbar, click **Plot**.

Add a new plot group to plot viscoelastic stress and reproduce [Figure 3](#).

#### *ID Plot Group 6*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Viscoelastic Stress** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
- 4 From the **Time selection** list, choose **Interpolated**.
- 5 In the **Times (s)** text field, type  $10^{\{\text{range}(-5, 0.2, 2.4)\}}$ .
- 6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 7 In the **Title** text area, type **Radial viscoelastic stress**.
- 8 Locate the **Plot Settings** section. Select the **y-axis label** check box.
- 9 In the associated text field, type **Viscoelastic stress (N/m<sup>2</sup>)**.
- 10 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

#### *Point Graph 1*

- 1 Right-click **Viscoelastic Stress** and choose **Point Graph**.
- 2 Select Point 4 only.
- 3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Viscoelastic stress tensor, local coordinate system>solid.hmm1.vis1.Sql11 - Viscoelastic stress tensor, local coordinate system, 11 component**.
- 4 Click to expand the **Legends** section. Select the **Show legends** check box.
- 5 From the **Legends** list, choose **Manual**.
- 6 In the table, enter the following settings:

---

<b>Legends</b>
Total viscoelastic stress

---

#### *Point Graph 2*

- 1 Right-click **Results>Viscoelastic Stress>Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `solid.hmm1.vis1.betavm1*solid.S1iso11-solid.hmm1.vis1.qm1_11`.

4 Locate the **Legends** section. In the table, enter the following settings:

<b>Legends</b>
Branch 1

5 Duplicate the point graph four times and change properties according to the table:

<b>Point graph</b>	<b>Expression of y-Axis†Data</b>	<b>Legends</b>
Point Graph 3	solid.hmm1.vis1.betavm2*solid.Sliso11-solid.hmm1.vis1.qm2_11	Branch 2
Point Graph 4	solid.hmm1.vis1.betavm3*solid.Sliso11-solid.hmm1.vis1.qm3_11	Branch 3
Point Graph 5	solid.hmm1.vis1.betavm4*solid.Sliso11-solid.hmm1.vis1.qm4_11	Branch 4
Point Graph 6	solid.hmm1.vis1.betavm5*solid.Sliso11-solid.hmm1.vis1.qm5_11	Branch 5

#### *Point Graph 1*

- 1 In the **Model Builder** window, under **Results>Viscoelastic Stress** click **Point Graph 1**.
- 2 In the **Settings** window for **Point Graph**, click to expand the **Coloring and style** section.
- 3 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.

#### *Viscoelastic Stress*

- 1 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.
- 2 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Results>Viscoelastic Stress**.
- 3 On the **Viscoelastic Stress** toolbar, click **Plot**.

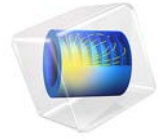
Disable viscoelasticity in study 1 in order to decrease the computation cost if you want to run it again.

## **STUDY 1**

### *Step 1: Stationary*

- 1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Hyperelastic Material 1>Viscoelasticity 1**.

**5** Click **Disable**.



# Inflation of a Spherical Rubber Balloon

## Introduction

---

this example aims to investigate the inflation of a rubber balloon using different hyperelastic material models, and to compare the results to analytical expressions.

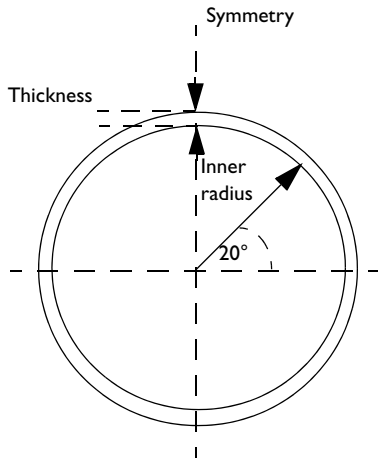
A controlled inflation could be of importance in clinical applications, cardiovascular research, and medical device industry (Ref. 2), among others.

The example is taken from the book *Nonlinear Solid Mechanics* by G. Holzapfel (Ref. 1).

## Model Definition

---

this example compares the hoop stress and inflation pressure as a function of the stretch for a spherical rubber balloon.



*Figure 1: Model geometry. The initial inner radius is set to 10 cm, and the initial thickness to 1 mm.*

In this example, the following four hyperelastic material models are compared: neo-Hookean, Money-Rivlin, Ogden, and Varga.

Due to the spherical symmetry, an arbitrary sector in the azimuthal direction can be used. Here, a 20 degrees sector is modeled in a 2D axial symmetry plane.

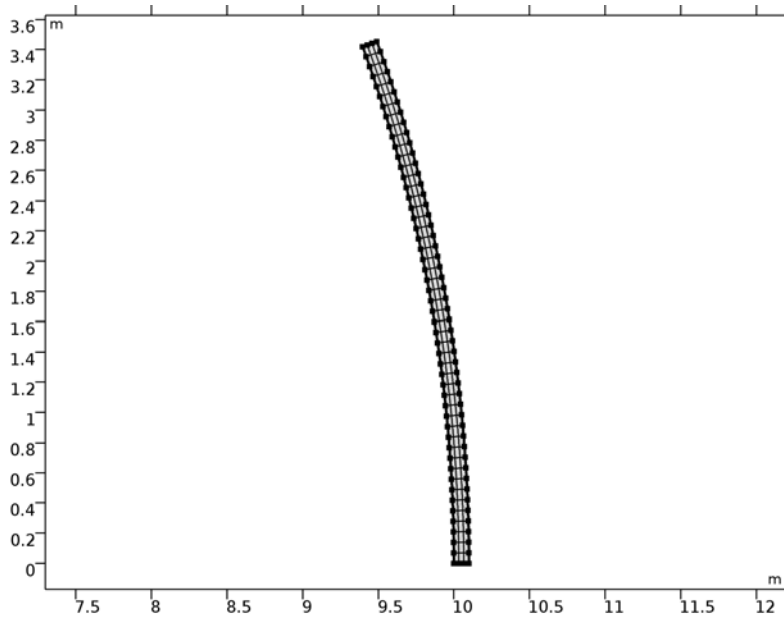


Figure 2: 2D axisymmetric geometry and mesh.

### Results and Discussion

The results are compared to the analytical expression for a thin-walled vessel. The inflation pressure is a function of the hoop stress  $\sigma_\theta$ , current inner radius  $r$  and current thickness  $h$

$$p_i = 2\frac{h}{r}\sigma_\theta$$

For spherical balloons, the hoop stress  $\sigma_\theta$  is equal to the largest principal stresses  $\sigma_1$  and  $\sigma_2$ . Two of the principal stretches lay on the plane tangential to the sphere and are equal,  $\lambda = \lambda_1 = \lambda_2 = r/R$ , which is typical for equibiaxial deformation. Here,  $r$  and  $R$  are the current and initial inner radii, respectively.

Due to the nearly incompressibility assumption, the third principal stretch (this is the stretch in the radial direction) is equal to  $\lambda_3 = 1/\lambda^2 = h/H$ , where  $h$  and  $H$  are the current and initial balloon thicknesses, respectively.

The analytical expression for the hoop stress for the Ogden material model becomes (Ref. 1)

$$\sigma_{\theta} = \sum_{p=1}^N \mu_p (\lambda^{\alpha_p} - \lambda^{-2\alpha_p})$$

where  $\alpha_p$  and  $\mu_p$  are Ogden parameters, and  $\lambda$  is the largest principal stretch.

Since  $r = R\lambda$  and  $h = H/\lambda^2$ , the analytical expression for the inflation pressure is calculated as a function of Ogden parameters, stretch, initial thickness and initial inner radius

$$p_i = 2\frac{h}{r}\sigma_{\theta} = 2\frac{H}{R}\sum_{p=1}^N \mu_p (\lambda^{\alpha_p-3} - \lambda^{-2\alpha_p-3})$$

The results are in excellent agreement with experimental results and the figures portrayed in [Ref. 1](#).

The experiments show a rapid rise in the internal pressure until reaching a maximum value, followed by a pressure decrease until reaching a minimum, and then increasing again. This phenomenon is similar to snap-through buckling, and can only be recovered by the Ogden material model. The Neo-Hookean and the Varga material models can only reproduce balloon inflations at small strain levels.

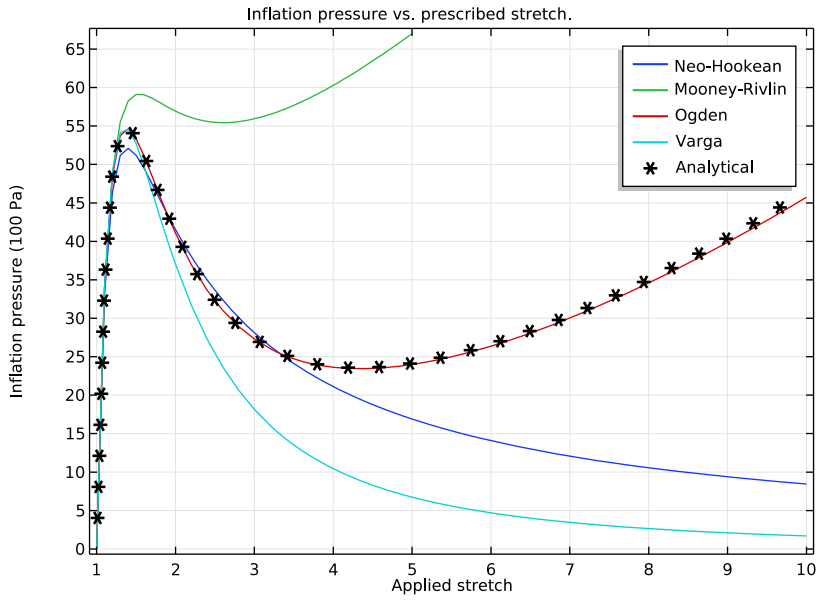


Figure 3: Computed inflation pressure as a function of circumferential stretch for different material models, compared to the analytical expression for Ogden material.

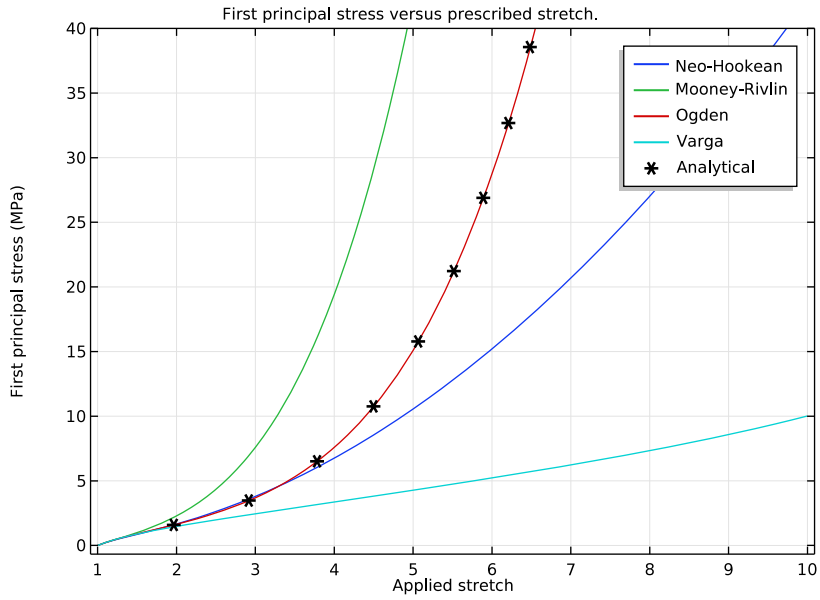


Figure 4: Computed hoop stress as a function of circumferential stretch for different material models, compared to the analytical expression for Ogden material.

Both plots are in an excellent agreement with the results described in Ref. 1, page 241.

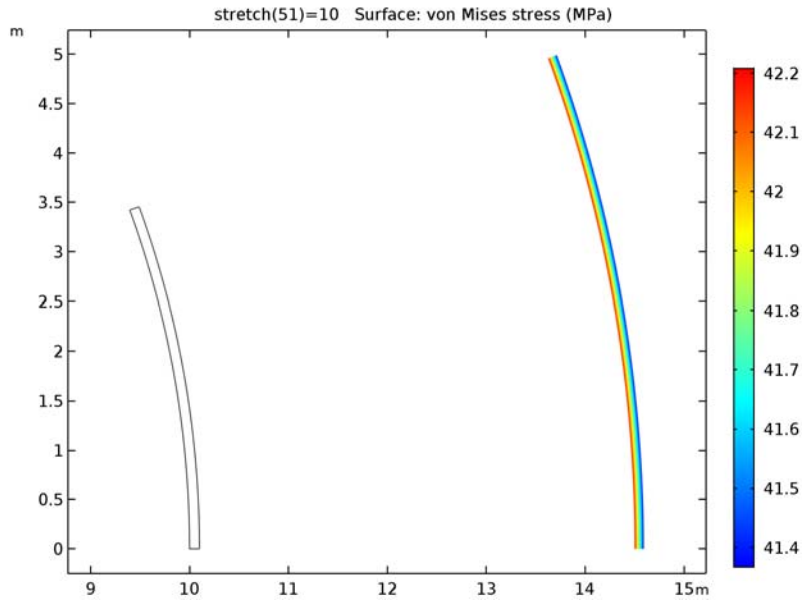


Figure 5: The first default plot shows the von Mises stress on the modeled 2D cross section for the Neo-Hookean material at maximum inflation.

### Notes About the COMSOL Implementation

Different hyperelastic material models are constructed by specifying different elastic strain energy expressions. The Nonlinear Structural Materials Module provides several predefined material models together with an option to enter user defined expressions for the strain energy density.

The predefined nearly incompressible version of the neo-Hookean material uses the isochoric invariant  $I_1(\bar{C}_{el})$  and the initial bulk modulus  $\kappa$

$$W_s = \frac{1}{2}\mu(\bar{I}_1 - 3) + \frac{1}{2}\kappa(J_{el} - 1)^2$$

In this example,  $\mu = 422.5$  kPa and  $\kappa = 10^5\mu$ . The Lamé parameter  $\mu$  can be seen as representing the shear modulus at small strains.

The predefined nearly incompressible Mooney-Rivlin material has an elastic strain energy density written in terms of the two isochoric invariant of the elastic right Cauchy-Green deformation tensors  $I_1(\bar{C}_{el})$  and  $I_2(\bar{C}_{el})$ , and the elastic volume ratio  $J_{el}$

$$W_s = C_{10}(\bar{I}_1 - 3) + C_{01}(\bar{I}_2 - 3) + \frac{1}{2}\kappa(J_{el} - 1)^2$$

The material parameters  $C_{10}$  and  $C_{01}$  are related to the shear modulus  $\mu = 2(C_{10} + C_{01})$ . In this example, they are set as  $C_{10} = 7/16\mu$  and  $C_{01} = \mu/16$ , so that the relation  $C_{10} = 7C_{01}$  is fulfilled.

The predefined nearly incompressible Ogden material is implemented with the isochoric elastic stretches and the initial bulk modulus  $\kappa$

$$W_s = \sum_{p=1}^N \frac{\mu_p}{\alpha_p} (\bar{\lambda}_{cl1}^{-\alpha_p} + \bar{\lambda}_{cl2}^{-\alpha_p} + \bar{\lambda}_{cl3}^{-\alpha_p} - 3) + \frac{1}{2}\kappa(J_{el} - 1)^2$$

with  $N = 3$ , and the Ogden parameters as written in [Table 1](#)

TABLE 1: OGDEN PARAMETERS

p	$\alpha_p$	$\mu_p$ (kPa)
1	1.3	630
2	5.0	1.2
3	-2.0	-10

The Varga material model is implemented with a user defined strain energy density

$$W_s = 2\mu(\lambda_{cl1} + \lambda_{cl2} + \lambda_{cl3} - 3) + \frac{1}{2}\kappa(J_{el} - 1)^2$$

When the relation between the applied load and the displacement is not unique, a suitable modeling technique is to use an algebraic equation that controls the applied pressure, so that the model reaches the desired displacement increments.

In this example, a Global Equation uses the radial displacement at point 3 to add an extra degree of freedom for the inflation pressure.

Global equations are a way of adding an additional equation to a model. A global equation can be used to describe a load, constraint, material property, or anything else in the model that has a uniquely definable solution. In this example, the model is augmented by a global equation which solves for the inflation pressure to achieve a desired applied stretch.

## References

---

1. G.A. Holzapfel, *Nonlinear Solid Mechanics: A Continuum Approach for Engineering*, John Wiley & Sons, 2000.
2. H. Azarnoush, S. Vergnole, B. Boulet, R. DiRaddo, and G. Lamouche, “Real-time control of angioplasty balloon inflation based on feedback from intravascular optical coherence tomography: preliminary study on an artery phantom,” *IEEE Trans Biomed Eng.* vol. 59, pp. 697–705, 2012.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Hyperelasticity/balloon\_inflation

---

## 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 Axisymmetric**.
- 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 **Preset Studies>Stationary**.
- 6 Click **Done**.

### **GLOBAL DEFINITIONS**

Begin by defining model parameters.

#### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
Ri	10[m]	10 m	Inner radius
H	0.1[m]	0.1 m	Thickness
mu	4.225e5[Pa]	4.225E5 Pa	Shear modulus
kappa	1e5*mu	4.225E10 Pa	Bulk modulus
stretch	1	1	Applied stretch

Setting the bulk modulus to  $10^5$  times the shear modulus is based on the assumption that the material is incompressible.

#### Variables 1

- 1 On the **Home** toolbar, click **Variables** and choose **Global Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
u_appl	(stretch-1)*Ri	m	Applied displacement

Use the stretch and geometry parameters to calculate the applied displacement.

#### GEOMETRY 1

Due to symmetry, it suffices to model a 20-degree sector of the balloon.

#### Circle 1 (c1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type Ri+H.
- 4 In the **Sector angle** text field, type 20.
- 5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	H

- 6 Click **Build All Objects**.
- 7 Click the **Zoom Extents** button on 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 1 only.
- 5 Click **Build All Objects**.
- 6 Click the **Zoom Extents** button on the **Graphics** toolbar.

## **SOLID MECHANICS (SOLID)**

Add the four hyperelastic material models.

### *Hyperelastic Material 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Neo-Hookean in the **Label** text field.
- 3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.
- 4 Locate the **Hyperelastic Material** section. Select the **Nearly incompressible material** check box.
- 5 In the  $\kappa$  text field, type kappa.
- 6 From the  $\mu$  list, choose **User defined**. In the associated text field, type  $\mu$ .

### *Hyperelastic Material 2*

- 1 On the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Mooney-Rivlin in the **Label** text field.
- 3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.
- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney-Rivlin, two parameters**.
- 5 From the  $C_{10}$  list, choose **User defined**. In the associated text field, type  $0.4375*\mu$ .
- 6 From the  $C_{01}$  list, choose **User defined**. In the associated text field, type  $0.0625*\mu$ .
- 7 In the  $\kappa$  text field, type kappa.

### *Hyperelastic Material 3*

- 1 On the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Ogden in the **Label** text field.
- 3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Ogden**.
- 5 Select the **Nearly incompressible material** check box.
- 6 Click **Add** twice.
- 7 In the **Ogden parameters** table, enter the following settings:

p	Shear modulus (Pa)	Alpha parameter
1	6.3e5	1.3
2	0.012e5	5
3	-0.1e5	-2

- 8 In the  $\kappa$  text field, type kappa.

#### *Hyperelastic Material 4*

- 1 On the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Varga in the **Label** text field.
- 3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.
- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **User defined**.
- 5 Select the **Nearly incompressible material** check box.
- 6 In the  $W_{\text{siso}}$  text field, type  $2*\mu*(\text{solid.stchelp1}+\text{solid.stchelp2}+\text{solid.stchelp3}-3)$ .
- 7 In the  $W_{\text{svol}}$  text field, type  $0.5*\text{kappa}*(\text{solid.Je1}-1)^2$ .

Apply symmetry conditions.

#### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 1 and 2 only.

Control the inflation of the balloon by the pressure.

#### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 From the **Load type** list, choose **Pressure**.
- 5 In the  $p$  text field, type  $p\_f$ .

You will define the pressure  $p_f$  using a Global Equation feature shortly. First, define an integration coupling operator to evaluate the displacement at Point 3.

## DEFINITIONS

### *Integration 1 (intop1)*

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Point**.
- 4 Select Point 3 only.
- 5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

### *Variables 2*

- 1 On 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
ub	intop1(u)	m	Radial displacement, inner boundary

## SOLID MECHANICS (SOLID)

In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu to allow to add a global equation and other advanced modeling features to the Solid Mechanics interface.

### *Global Equations 1*

- 1 On the **Physics** toolbar, click **Global** and choose **Global Equations**.
- 2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.
- 3 In the table, enter the following settings:

Name	$f(u,ut,utt,t)$ (l)	Initial value (u_0) (l)	Initial value (u_t0) (l/s)	Description
p_f	ub-u_app1	0	0	

- 4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.
- 5 In the **Physical Quantity** dialog box, In the associated text field, type id:pressure.
- 6 Click **Filter**.
- 7 In the tree, select **Transport>Pressure (Pa)**.

- 8 Click **OK**.
- 9 In the **Settings** window for **Global Equations**, locate the **Units** section.
- 10 Click **Select Source Term Quantity**.
- 11 In the **Physical Quantity** dialog box, In the associated text field, type `id:displacement`.
- 12 Click **Filter**.
- 13 In the tree, select **Solid>Displacement field (m)**.
- 14 Click **OK**.

## **MESH 1**

### *Distribution 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
- 2 Right-click **Mapped 1** and choose **Distribution**.
- 3 Select Boundary 2 only.
- 4 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 5 In the **Number of elements** text field, type 3.

### *Distribution 2*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 50.
- 5 In the **Model Builder** window, click **Mesh 1**.
- 6 In the **Settings** window for **Mesh**, click **Build All**.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

## **STUDY 1**

The first study solves the problem with a Neo-Hookean material model.

### *Step 1: Stationary*

- 1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 2 Select the **Modify model configuration for study step** check box.
- 3 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Mooney-Rivlin, Component 1 (comp1)>**

**Solid Mechanics (solid), Controls spatial frame>Ogden, and Component 1 (comp1)>  
Solid Mechanics (solid), Controls spatial frame>Varga.**

4 Click **Disable**.

Define a parametric analysis where the applied stretch varies from 1 to 10 using the continuation solver.

5 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
stretch (Applied stretch)	range(1, 0.1, 2) range(2.2, 0.2, 10)	

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

9 In the **Settings** window for **Study**, type Neo-Hookean in the **Label** text field.

Modify the default solver to improve convergence.

*Solution 1 (sol1)*

On the **Study** toolbar, click **Show Default Solver**.

## NEO-HOOKEAN

*Solution 1 (sol1)*

Use manual scaling to help the nonlinear solver at the first steps. A constant predictor is also suitable for nonlinear materials.

1 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Dependent Variables 1**.

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

3 From the **Method** list, choose **Manual**.

4 In the **Model Builder** window, expand the **Neo-Hookean>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Direct**.

5 In the **Settings** window for **Direct**, locate the **General** section.

6 From the **Solver** list, choose **PARDISO**.

7 In the **Model Builder** window, under **Neo-Hookean>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Parametric 1**.

- 8 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
- 9 From the **Predictor** list, choose **Constant**.
- 10 In the **Model Builder** window, under **Neo-Hookean>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Fully Coupled 1**.
- 11 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.
- 12 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Constant (Newton)**.
- 13 On the **Study** toolbar, click **Compute**.

Add a second study to solve for the Mooney-Rivlin material model, then repeat the steps described above.

#### **ADD STUDY**

- 1 On the **Study** 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 **Preset Studies**.
- 4 In the **Select Study** tree, select **Preset Studies>Stationary**.
- 5 Click **Add Study** in the window toolbar.

#### **STUDY 2**

*Step 1: Stationary*

- 1 In the **Model Builder** window, click **Study 2**.
- 2 In the **Settings** window for **Study**, type Mooney-Rivlin in the **Label** text field.
- 3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

#### **MOONEY-RIVLIN**

*Step 1: Stationary*

- 1 In the **Model Builder** window, under **Mooney-Rivlin** 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>Neo-Hookean, Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Ogden, and Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Varga**.

5 Click **Disable**.

For the Mooney-Rivlin material, use a parametric continuation analysis that changes from 1 to 5 in 50 steps.

6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
stretch (Applied stretch)	range(1, 0.1, 5)	

#### *Solution 2 (sol2)*

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Dependent Variables I**.

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

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

5 In the **Model Builder** window, expand the **Mooney-Rivlin>Solver Configurations>Solution 2 (sol2)>Stationary Solver I** node, then click **Direct**.

6 In the **Settings** window for **Direct**, locate the **General** section.

7 From the **Solver** list, choose **PARDISO**.

8 In the **Model Builder** window, under **Mooney-Rivlin>Solver Configurations>Solution 2 (sol2)>Stationary Solver I** click **Parametric I**.

9 In the **Settings** window for **Parametric**, locate the **Continuation** section.

10 From the **Predictor** list, choose **Constant**.

11 In the **Model Builder** window, under **Mooney-Rivlin>Solver Configurations>Solution 2 (sol2)>Stationary Solver I** click **Fully Coupled I**.

12 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

13 From the **Nonlinear method** list, choose **Constant (Newton)**.

14 On the **Study** toolbar, click **Compute**.

Continue with a third study for the Ogden material model.

#### **ADD STUDY**

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

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

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

### STUDY 3

#### *Step 1: Stationary*

- 1 In the **Model Builder** window, click **Study 3**.
- 2 In the **Settings** window for **Study**, type Odgen in the **Label** text field.
- 3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

### ODGEN

#### *Step 1: Stationary*

- 1 In the **Model Builder** window, under **Odgen** 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>Neo-Hookean, Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Mooney-Rivlin**, and **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Varga**.
- 5 Click **Disable**.
- 6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 7 Click **Add**.
- 8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
stretch (Applied stretch)	range(1, 0.1, 2) range(2.2, 0.2, 10)	

#### *Solution 3 (sol3)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Dependent Variables 1**.
- 3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.
- 4 From the **Method** list, choose **Manual**.
- 5 In the **Model Builder** window, expand the **Odgen>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node, then click **Direct**.
- 6 In the **Settings** window for **Direct**, locate the **General** section.

- 7 From the **Solver** list, choose **PARDISO**.
- 8 In the **Model Builder** window, under **Odgen>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** click **Parametric 1**.
- 9 In the **Settings** window for **Parametric**, locate the **Continuation** section.
- 10 From the **Predictor** list, choose **Constant**.
- 11 In the **Model Builder** window, under **Odgen>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** click **Fully Coupled 1**.
- 12 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
- 13 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 14 On the **Study** toolbar, click **Compute**.

Finally, add a fourth study for Varga material model.

#### **ADD STUDY**

- 1 Go to the **Add Study** window.
- 2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.
- 3 Click **Add Study** in the window toolbar.
- 4 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

#### **STUDY 4**

- 1 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 2 Clear the **Generate default plots** check box.

#### *Step 1: Stationary*

- 1 In the **Model Builder** window, under **Study 4** 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>Neo-Hookean, Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Mooney-Rivlin, and Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Ogden**.
- 5 Click **Disable**.
- 6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
stretch (Applied stretch)	range(1, 0.1, 2) range(2.2, 0.2, 10)	

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

10 In the **Settings** window for **Study**, type Varga in the **Label** text field.

*Solution 4 (sol4)*

On the **Study** toolbar, click **Show Default Solver**.

## VARGA

*Solution 4 (sol4)*

1 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Dependent Variables I**.

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

3 From the **Method** list, choose **Manual**.

4 In the **Model Builder** window, expand the **Varga>Solver Configurations>Solution 4 (sol4)>Stationary Solver I** node, then click **Direct**.

5 In the **Settings** window for **Direct**, locate the **General** section.

6 From the **Solver** list, choose **PARDISO**.

7 In the **Model Builder** window, under **Varga>Solver Configurations>Solution 4 (sol4)>Stationary Solver I** click **Parametric I**.

8 In the **Settings** window for **Parametric**, locate the **Continuation** section.

9 From the **Predictor** list, choose **Constant**.

10 In the **Model Builder** window, under **Varga>Solver Configurations>Solution 4 (sol4)>Stationary Solver I** click **Fully Coupled I**.

11 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

12 From the **Nonlinear method** list, choose **Constant (Newton)**.

13 On the **Study** toolbar, click **Compute**.

The first default plot shows the von Mises stress on the modeled 2D cross section for the Neo Hookean material at maximum inflation. When you adjust the scaling, the plot should become similar to [Figure 5](#).

## RESULTS

### *Surface 1*

- 1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.

### *Deformation*

- 1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 In the **Scale factor** text field, type 0.05.
- 4 On the **Stress (solid)** toolbar, click **Plot**.
- 5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The second default plot shows the von Mises stress in a 3D revolved plot. To reproduce [Figure 3](#), proceed as follows.

### *ID Plot Group 3*

- 1 In the **Model Builder** window, under **Results** click **ID Plot Group 3**.
- 2 In the **Settings** window for **ID Plot Group**, type Inflation Pressure in the **Label** text field.
- 3 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 4 In the associated text field, type Applied stretch.
- 5 Select the **y-axis label** check box.
- 6 In the associated text field, type Inflation pressure (100 Pa).
- 7 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 8 In the **Title** text area, type Inflation pressure vs. prescribed stretch..

### *Point Graph 1*

- 1 Right-click **Results>Inflation Pressure** and choose **Point Graph**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Data set** list, choose **Neo-Hookean/Solution 1 (sol1)**.
- 4 Select Point 3 only.
- 5 Locate the **y-Axis Data** section. In the **Expression** text field, type  $p_f/100$ .
- 6 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Global Definitions>Parameters>stretch - Applied stretch**.

- 7 Click to expand the **Legends** section. Select the **Show legends** check box.
- 8 From the **Legends** list, choose **Manual**.
- 9 In the table, enter the following settings:

<b>Legends</b>
Neo-Hookean

- 10 On the **Inflation Pressure** toolbar, click **Plot**.

#### *Point Graph 2*

- 1 Right-click **Results>Inflation Pressure>Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Data set** list, choose **Mooney-Rivlin/Solution 2 (sol2)**.
- 4 Locate the **Legends** section. In the table, enter the following settings:

<b>Legends</b>
Mooney-Rivlin

- 5 On the **Inflation Pressure** toolbar, click **Plot**.

#### *Point Graph 3*

- 1 Right-click **Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Data set** list, choose **Odgen/Solution 3 (sol3)**.
- 4 Locate the **Legends** section. In the table, enter the following settings:

<b>Legends</b>
Ogden

- 5 On the **Inflation Pressure** toolbar, click **Plot**.

#### *Point Graph 4*

- 1 Right-click **Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Data set** list, choose **Varga/Solution 4 (sol4)**.
- 4 Locate the **Legends** section. In the table, enter the following settings:

<b>Legends</b>
Varga

5 On the **Inflation Pressure** toolbar, click **Plot**.

#### *Point Graph 5*

- 1 Right-click **Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type  $2 * (H/Ri) * ((6.3e5[Pa] * (stretch^{(1.3-3)} - stretch^{(-2*1.3-3)})) + (0.012e5[Pa] * (stretch^{(5-3)} - stretch^{(-2*5-3)})) - (0.1e5[Pa] * (stretch^{(-2-3)} - stretch^{(2*2-3)}))) / 100$ .
- 4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 5 From the **Color** list, choose **Black**.
- 6 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.
- 7 In the **Number** text field, type 40.
- 8 Locate the **Legends** section. In the table, enter the following settings:

---

#### **Legends**

---

Analytical

---

9 On the **Inflation Pressure** toolbar, click **Plot**.

To reproduce [Figure 4](#), proceed as follows.

#### *Inflation Pressure 1*

- 1 In the **Model Builder** window, under **Results** right-click **Inflation Pressure** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type First Principal Stress in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type First principal stress versus prescribed stretch..
- 4 Locate the **Plot Settings** section. In the **y-axis label** text field, type First principal stress (MPa).
- 5 Locate the **Axis** section. Select the **Manual axis limits** check box.
- 6 In the **y maximum** text field, type 40.

#### *Point Graph 1*

- 1 In the **Model Builder** window, expand the **Results>First Principal Stress** node, then click **Point Graph 1**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

**3** In the **Expression** text field, type `solid.sp1`.

**4** From the **Unit** list, choose **MPa**.

#### *Point Graph 2*

**1** In the **Model Builder** window, under **Results>First Principal Stress** click **Point Graph 2**.

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

**3** In the **Expression** text field, type `solid.sp1`.

**4** From the **Unit** list, choose **MPa**.

#### *Point Graph 3*

**1** In the **Model Builder** window, under **Results>First Principal Stress** click **Point Graph 3**.

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

**3** In the **Expression** text field, type `solid.sp1`.

**4** From the **Unit** list, choose **MPa**.

#### *Point Graph 4*

**1** In the **Model Builder** window, under **Results>First Principal Stress** click **Point Graph 4**.

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

**3** In the **Expression** text field, type `solid.sp1`.

**4** From the **Unit** list, choose **MPa**.

#### *Point Graph 5*

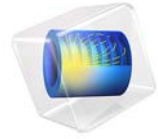
**1** In the **Model Builder** window, under **Results>First Principal Stress** click **Point Graph 5**.

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

**3** In the **Expression** text field, type  $((6.3e5[\text{Pa}] * (\text{stretch}^{1.3} - \text{stretch}^{(-2*1.3)})) + (0.012e5[\text{Pa}] * (\text{stretch}^5 - \text{stretch}^{(-2*5)})) - (0.1e5[\text{Pa}] * (\text{stretch}^{(-2)} - \text{stretch}^{(2*2)})))$ .

**4** From the **Unit** list, choose **MPa**.

**5** On the **First Principal Stress** toolbar, click **Plot**.



# Inflation of a Spherical Rubber Balloon— Membrane Version

## *Introduction*

---

The purpose of this example is to illustrate how membrane interface can be used to model thin hyperelastic structures. The model is identical to the Application Library example, [Inflation of a Spherical Rubber Balloon](#), however in this example the Membrane interface has been used for modeling instead of the Solid Mechanics interface.

The difference between using the Membrane and Solid Mechanics interfaces is that the variation across the thickness is not computed in membrane version as the membrane interface is based on plane stress assumption for thin structures. However, since the membrane interface is on one geometric dimension lower than solid mechanics this example has an advantage of being more computationally efficient.

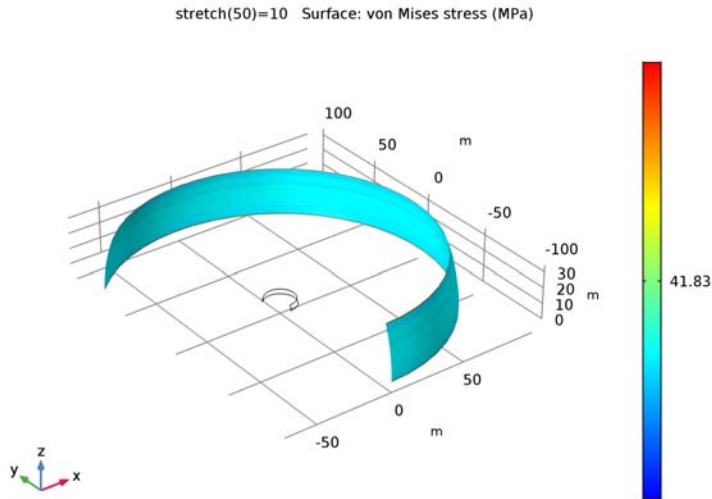
All relevant details about the model can be found in the example, [Inflation of a Spherical Rubber Balloon](#).

## *Results and Discussions*

---

The results from the membrane version, shown in subsequent figures are almost equivalent to the results from solid mechanics version.

[Figure 1](#) shows the von Mises stress on the revolution 3D data set for neo-Hookean material model at maximum inflation. A uniform stress of 41.8 MPa is obtained from membrane interface which agrees quite well with the solid mechanics version in which the stress varies from 41.4 MPa to 42.2 MPa across the thickness.



*Figure 1: von Mises stress for the Neo-Hookean material at maximum inflation*

The variation of inflation pressure for different hyperelastic material models with applied stretch is shown in [Figure 2](#). The plot is found to be identical to solid mechanics version for all material models.

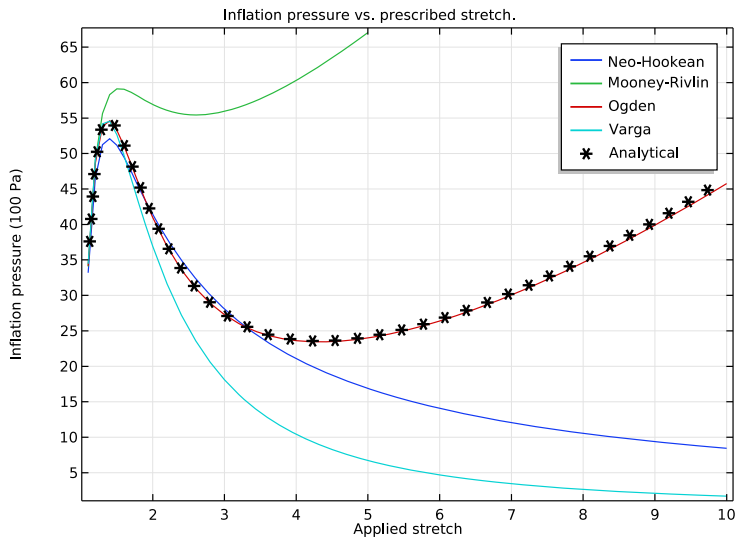


Figure 2: Computed inflation pressure as a function of circumferential stretch for different material models, compared to the analytical expression for Ogden material.

The variation of hoop stress for different hyperelastic material models with applied stretch is shown in Figure 3. The plot is found to be identical to solid mechanics version for all material models.

In Figure 4, a comparison between membrane and solid mechanics versions is made by plotting the deformed thickness of the balloon for both the versions. Since, the results are overlapping for the two versions, it can be said that the thinning of the balloon can be accurately captured using the membrane interface.

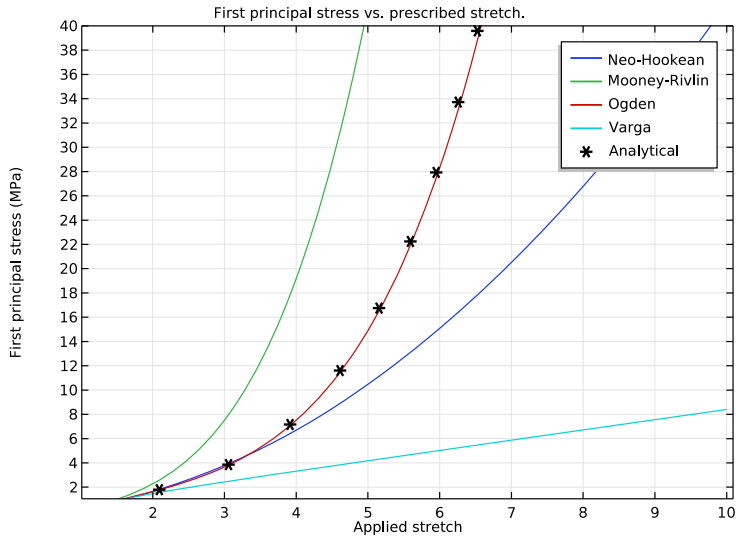


Figure 3: Computed hoop stress as a function of circumferential stretch for different material models, compared to the analytical expression for Ogden material.

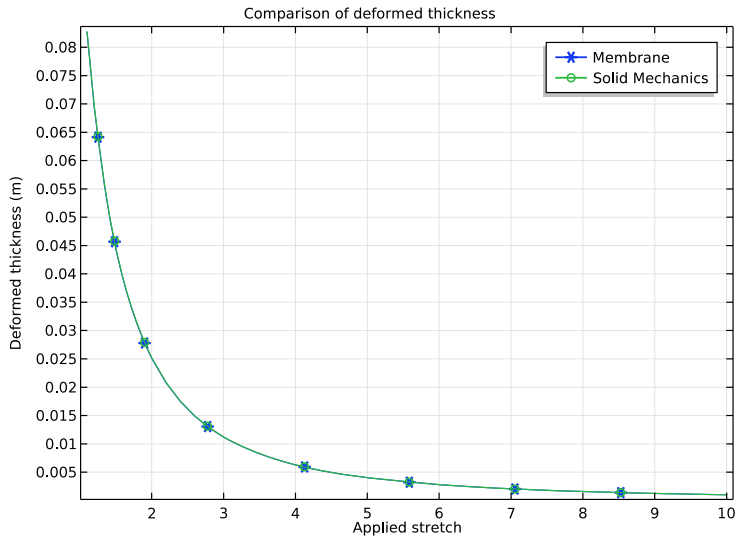


Figure 4: Comparison of deformed thickness of balloon for membrane and solid mechanics versions.

## *Notes About the COMSOL Implementation*

---

Due to the absence of bending stiffness in membrane it requires a prestress to solve the model. For this a separate study has been created before the inflation of the balloon is carried out in further studies. Results from this study are used as initial values for the inflation studies.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Hyperelasticity/balloon\_inflation\_membrane

---

## *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 Axisymmetric**.
- 2 In the **Select Physics** tree, select **Structural Mechanics>Membrane (mbrn)**.
- 3 Click **Add**.
- 4 Click **Study**.
- 5 In the **Select Study** tree, select **Preset Studies>Stationary**.
- 6 Click **Done**.

### **GLOBAL DEFINITIONS**

Begin by defining model parameters.

#### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
Ri	10[m]	10 m	Inner radius
H	0.1[m]	0.1 m	Thickness
mu	4.225e5[Pa]	4.225E5 Pa	Shear modulus
kappa	1e5*mu	4.225E10 Pa	Bulk modulus
stretch	1	1	Applied stretch

Setting the bulk modulus to  $10^5$  times the shear modulus is based on the assumption that the material is incompressible.

Use the stretch and geometry parameters to calculate the applied displacement.

#### Variables 1

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
u_appl	(stretch-1)*Ri	m	Applied displacement

Using an interpolation function, import the deformed thickness as a function stretch from solid mechanics version.

#### Interpolation 1 (int1)

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 Click **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `balloon_inflation_membrane_interpolation.txt`.
- 5 Locate the **Units** section. In the **Arguments** text field, type 1.
- 6 In the **Function** text field, type m.

#### GEOMETRY 1

Due to symmetry, it suffices to model a 20-degree sector of the balloon.

#### Circle 1 (c1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

- 2 In the **Settings** window for **Circle**, locate the **Object Type** section.
- 3 From the **Type** list, choose **Curve**.
- 4 Locate the **Size and Shape** section. In the **Radius** text field, type  $R_i + H/2$ .
- 5 In the **Sector angle** text field, type 20.

*Delete Entities 1 (del1)*

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.
- 2 On the object **c1**, select Boundaries 2 and 3 only.
- 3 Right-click **Component 1 (comp1)>Geometry 1>Delete Entities 1 (del1)** and choose **Build Selected**.

**MEMBRANE (MBRN)**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Membrane (mbrn)**.
- 2 In the **Settings** window for **Membrane**, locate the **Thickness** section.
- 3 In the  $d$  text field, type  $H$ .

Add the four hyperelastic material models.

*Hyperelastic Material 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Non-Hookean in the **Label** text field.
- 3 Locate the **Boundary Selection** section. From the **Selection** list, choose **All boundaries**.
- 4 Locate the **Hyperelastic Material** section. Select the **Nearly incompressible material** check box.
- 5 From the  $\mu$  list, choose **User defined**. In the associated text field, type  $\mu$ .
- 6 In the  $\kappa$  text field, type  $\kappa$ .

*Hyperelastic Material 2*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Mooney-Rivlin in the **Label** text field.
- 3 Locate the **Boundary Selection** section. From the **Selection** list, choose **All boundaries**.
- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney-Rivlin, two parameters**.
- 5 From the  $C_{10}$  list, choose **User defined**. In the associated text field, type  $0.4375 * \mu$ .

- 6 From the  $C_{01}$  list, choose **User defined**. In the associated text field, type  $0.0625*\mu$ .
- 7 In the  $\kappa$  text field, type kappa.

#### *Hyperelastic Material 3*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Ogden in the **Label** text field.
- 3 Locate the **Boundary Selection** section. From the **Selection** list, choose **All boundaries**.
- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Ogden**.
- 5 Select the **Nearly incompressible material** check box.
- 6 Click **Add** twice.
- 7 In the **Ogden parameters** table, enter the following settings:

p	Shear modulus (Pa)	Alpha parameter
1	6.3e5	1.3
2	0.012e5	5
3	-0.1e5	-2

- 8 In the  $\kappa$  text field, type kappa.

#### *Hyperelastic Material 4*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Varga in the **Label** text field.
- 3 Locate the **Boundary Selection** section. From the **Selection** list, choose **All boundaries**.
- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Varga**.
- 5 From the  $c_1$  list, choose **User defined**. In the associated text field, type  $2*\mu$ .
- 6 From the  $c_2$  list, choose **User defined**. In the  $\kappa$  text field, type kappa.

Apply symmetry conditions.

#### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Points** and choose **Symmetry**.
- 2 In the **Settings** window for **Symmetry**, locate the **Point Selection** section.
- 3 From the **Selection** list, choose **All points**.

Prescribe the displacement in normal direction for pre-stretch analysis.

#### *Prescribed Displacement 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

- 2 In the **Settings** window for **Prescribed Displacement**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.
- 4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System I (sys1)**.
- 5 Locate the **Prescribed Displacement** section. Select the **Prescribed in n direction** check box.
- 6 In the  $u_{0n}$  text field, type -0.1.

Control the inflation of the balloon by the pressure.

#### *Face Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Face Load**.
- 2 In the **Settings** window for **Face Load**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.
- 4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.
- 5 In the  $p$  text field, type  $p\_f$ .

You will define the pressure  $p\_f$  using a Global Equation feature shortly. First, define an integration coupling operator to evaluate the displacement at Point 2.

### **DEFINITIONS**

#### *Integration 1 (intop1)*

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Point**.
- 4 Select Point 2 only.
- 5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

#### *Variables 2*

- 1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

<b>Name</b>	<b>Expression</b>	<b>Unit</b>	<b>Description</b>
ub	intop1(u)	m	Radial displacement

## MEMBRANE (MBRN)

In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu to allow to add a global equation and other advanced modeling features to the Membrane interface.

### *Global Equations 1*

- 1 On the **Physics** toolbar, click **Global** and choose **Global Equations**.
- 2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.
- 3 In the table, enter the following settings:

Name	$f(u,ut,utt,t)$ (l)	Initial value (u_0) (l)	Initial value (u_t0) (l/s)	Description
p_f	ub-u_app1	0	0	

- 4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.
- 5 In the **Physical Quantity** dialog box, In the associated text field, type id:pressure.
- 6 Click **Filter**.
- 7 In the tree, select **Transport>Pressure (Pa)**.
- 8 Click **OK**.
- 9 In the **Settings** window for **Global Equations**, locate the **Units** section.
- 10 Click **Select Source Term Quantity**.
- 11 In the **Physical Quantity** dialog box, In the associated text field, type id:length.
- 12 Click **Filter**.
- 13 In the tree, select **General>Length (m)**.
- 14 Click **OK**.

## MESH 1

### *Edge 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.
- 2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.

### *Distribution 1*

- 1 Right-click **Component 1 (comp1)>Mesh 1>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** Click **Build All**.

## **STUDY 1**

The first study solves for the pre-stretch analysis.

**1** In the **Settings** window for **Study**, type Study: Pre-stretch in the **Label** text field.

**2** Locate the **Study Settings** section. Clear the **Generate default plots** check box.

## **STUDY: PRE-STRETCH**

### *Step 1: Stationary*

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

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

**3** Select the **Modify physics tree and variables for study step** check box.

**4** In the **Physics and variables selection** tree, select **Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Face Load 1** and **Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Global Equations 1**.

**5** Click **Disable**.

**6** On the **Home** toolbar, click **Compute**.

Add study for Neo-Hookean material model.

## **ADD STUDY**

**1** On 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 **Preset Studies**.

**4** In the **Select Study** tree, select **Preset Studies>Stationary**.

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

**6** On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

## **STUDY 2**

### *Step 1: Stationary*

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

**2** In the **Settings** window for **Study**, type Study: Neo-Hookean in the **Label** text field.

## STUDY: NEO-HOOKEAN

### Step 1: Stationary

- 1 In the **Model Builder** window, under **Study: Neo-Hookean** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Mooney-Rivlin, Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Ogden, Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Varga, and Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Prescribed Displacement 1**.
- 5 Click **Disable**.
- 6 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 7 From the **Method** list, choose **Solution**.
- 8 From the **Study** list, choose **Study: Pre-stretch, Stationary**.  
Define a parametric analysis where the applied stretch varies from 1 to 10 using the continuation solver.
- 9 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 10 Click **Add**.
- 11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
stretch (Applied stretch)	range(1.1,0.1,2) range(2.2,0.2,10)	

Modify the default solver to improve convergence. Use manual scaling to help the nonlinear solver at the first steps, a constant predictor is also suitable for nonlinear materials.

### Solution 2 (sol2)

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Dependent Variables 1**.
- 3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

- 4 From the **Method** list, choose **Manual**.
- 5 In the **Model Builder** window, expand the **Study: Neo-Hookean>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Direct**.
- 6 In the **Settings** window for **Direct**, locate the **General** section.
- 7 From the **Solver** list, choose **PARDISO**.
- 8 In the **Model Builder** window, under **Study: Neo-Hookean>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** click **Parametric 1**.
- 9 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
- 10 From the **Predictor** list, choose **Constant**.
- 11 In the **Model Builder** window, under **Study: Neo-Hookean>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** click **Fully Coupled 1**.
- 12 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.
- 13 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Constant (Newton)**.
- 14 On the **Study** toolbar, click **Compute**.

Add study to solve for the Mooney-Rivlin material model, then repeat the steps described above.

#### **ADD STUDY**

- 1 On the **Study** 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 **Preset Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.
- 5 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

#### **STUDY 3**

##### *Step 1: Stationary*

- 1 In the **Model Builder** window, click **Study 3**.
- 2 In the **Settings** window for **Study**, type **Study: Mooney-Rivlin** in the **Label** text field.
- 3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

## STUDY: MOONEY-RIVLIN

### Step 1: Stationary

- 1 In the **Model Builder** window, under **Study: Mooney-Rivlin** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Non-Hookean, Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Ogden, Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Varga, and Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Prescribed Displacement 1**.
- 5 Click **Disable**.
- 6 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 7 From the **Method** list, choose **Solution**.
- 8 From the **Study** list, choose **Study: Pre-stretch, Stationary**.  
For the Mooney-Rivlin material, use a parametric continuation analysis that changes from 1 to 5 in 50 steps.
- 9 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 10 Click **Add**.
- 11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
stretch (Applied stretch)	range(1.1, 0.1, 5)	

### Solution 3 (sol3)

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Dependent Variables 1**.
- 3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.
- 4 From the **Method** list, choose **Manual**.
- 5 In the **Model Builder** window, expand the **Study: Mooney-Rivlin>Solver Configurations> Solution 3 (sol3)>Stationary Solver 1** node, then click **Direct**.
- 6 In the **Settings** window for **Direct**, locate the **General** section.

- 7 From the **Solver** list, choose **PARDISO**.
- 8 In the **Model Builder** window, under **Study: Mooney-Rivlin>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** click **Parametric 1**.
- 9 In the **Settings** window for **Parametric**, locate the **Continuation** section.
- 10 From the **Predictor** list, choose **Constant**.
- 11 In the **Model Builder** window, under **Study: Mooney-Rivlin>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** click **Fully Coupled 1**.
- 12 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
- 13 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 14 On the **Study** toolbar, click **Compute**.

Add study for the Ogden material model.

#### **ADD STUDY**

- 1 On the **Study** 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 **Preset Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.
- 5 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

#### **STUDY 4**

*Step 1: Stationary*

- 1 In the **Model Builder** window, click **Study 4**.
- 2 In the **Settings** window for **Study**, type Study: Ogden in the **Label** text field.
- 3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

#### **STUDY: OGDEN**

*Step 1: Stationary*

- 1 In the **Model Builder** window, under **Study: Ogden** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Non-Hookean, Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Mooney-Rivlin, Component 1 (comp1)>**

Membrane (mbrn), Controls spatial frame>Varga, and Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Prescribed Displacement 1.

- 5 Click **Disable**.
- 6 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 7 From the **Method** list, choose **Solution**.
- 8 From the **Study** list, choose **Study: Pre-stretch, Stationary**.
- 9 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 10 Click **Add**.
- 11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
stretch (Applied stretch)	range(1.1,0.1,2) range(2.2,0.2,10)	

*Solution 4 (sol4)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Dependent Variables 1**.
- 3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.
- 4 From the **Method** list, choose **Manual**.
- 5 In the **Model Builder** window, expand the **Study: Ogden>Solver Configurations>Solution 4 (sol4)>Stationary Solver 1** node, then click **Direct**.
- 6 In the **Settings** window for **Direct**, locate the **General** section.
- 7 From the **Solver** list, choose **PARDISO**.
- 8 In the **Model Builder** window, under **Study: Ogden>Solver Configurations>Solution 4 (sol4)>Stationary Solver 1** click **Parametric 1**.
- 9 In the **Settings** window for **Parametric**, locate the **Continuation** section.
- 10 From the **Predictor** list, choose **Constant**.
- 11 In the **Model Builder** window, under **Study: Ogden>Solver Configurations>Solution 4 (sol4)>Stationary Solver 1** click **Fully Coupled 1**.
- 12 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
- 13 From the **Nonlinear method** list, choose **Constant (Newton)**.

14 On the **Study** toolbar, click **Compute**.

Add study for the Varga material model.

#### **ADD STUDY**

1 On the **Study** 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 **Preset Studies>Stationary**.

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

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

#### **STUDY 5**

*Step 1: Stationary*

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

2 In the **Settings** window for **Study**, type Study: Varga in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

#### **STUDY: VARGA**

*Step 1: Stationary*

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

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

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Non-Hookean, Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Mooney-Rivlin, Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Ogden, and Component 1 (comp1)> Membrane (mbrn), Controls spatial frame>Prescribed Displacement 1**.

5 Click **Disable**.

6 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study: Pre-stretch, Stationary**.

9 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

10 Click **Add**.

11 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
stretch (Applied stretch)	range (1.1, 0.1, 2) range (2.2, 0.2, 10)	

#### *Solution 5 (sol5)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 5 (sol5)** node, then click **Dependent Variables I**.
- 3 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.
- 4 From the **Method** list, choose **Manual**.
- 5 In the **Model Builder** window, expand the **Study: Varga>Solver Configurations>Solution 5 (sol5)>Stationary Solver I** node, then click **Direct**.
- 6 In the **Settings** window for **Direct**, locate the **General** section.
- 7 From the **Solver** list, choose **PARDISO**.
- 8 In the **Model Builder** window, under **Study: Varga>Solver Configurations>Solution 5 (sol5)>Stationary Solver I** click **Parametric I**.
- 9 In the **Settings** window for **Parametric**, locate the **Continuation** section.
- 10 From the **Predictor** list, choose **Constant**.
- 11 In the **Model Builder** window, under **Study: Varga>Solver Configurations>Solution 5 (sol5)>Stationary Solver I** click **Fully Coupled I**.
- 12 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
- 13 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 14 On the **Study** toolbar, click **Compute**.

## RESULTS

#### *Stress (mbrn)*

The first default plot shows the von Mises stress on the modeled 1D cross section for the Neo Hookean material at maximum inflation. You can adjust the settings for a better view as described below.

- 1 In the **Model Builder** window, expand the **Stress (mbrn)** node.

#### *Line*

- 1 In the **Model Builder** window, expand the **Results>Stress (mbrn)>Line** node, then click **Line**.

- 2 In the **Settings** window for **Line**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.
- 4 Locate the **Coloring and Style** section. Select the **Radius scale factor** check box.
- 5 In the **Tube radius expression** text field, type 3.
- 6 On the **Stress (mbrn)** toolbar, click **Plot**.

#### *Deformation*

- 1 In the **Model Builder** window, under **Results>Stress (mbrn)>Line** click **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 Select the **Scale factor** check box.
- 4 In the associated text field, type 0.05.
- 5 On the **Stress (mbrn)** toolbar, click **Plot**.
- 6 Click the **Zoom Extents** button on the **Graphics** toolbar.

#### *Stress, 3D (mbrn)*

The second default plot shows the von Mises stress in a 3D revolved plot. To reproduce [Figure 1](#), proceed as follows.

#### *Surface*

- 1 In the **Model Builder** window, expand the **Stress, 3D (mbrn)** node, then click **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.

#### *Deformation*

- 1 In the **Model Builder** window, expand the **Surface** node, then click **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 Select the **Scale factor** check box.
- 4 In the associated text field, type 1.
- 5 On the **Stress, 3D (mbrn)** toolbar, click **Plot**.
- 6 Click the **Zoom Extents** button on the **Graphics** toolbar.

To reproduce [Figure 2](#), proceed as follows.

#### *ID Plot Group 3*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Inflation Pressure in the **Label** text field.

- 3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Inflation pressure vs. prescribed stretch..
- 5 Locate the **Plot Settings** section. Select the **y-axis label** check box.
- 6 In the associated text field, type Inflation pressure (100 Pa).

*Point Graph 1*

- 1 Right-click **Inflation Pressure** and choose **Point Graph**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study: Neo-Hookean/Solution 2 (sol2)**.
- 4 Select Point 2 only.
- 5 Locate the **y-Axis Data** section. In the **Expression** text field, type  $p_f/100$ .
- 6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 7 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Global Definitions>Parameters>stretch - Applied stretch**.
- 8 Click to expand the **Legends** section. Click to expand the **Coloring and style** section. Locate the **Legends** section. Select the **Show legends** check box.
- 9 From the **Legends** list, choose **Manual**.
- 10 In the table, enter the following settings:

---

**Legends**

Neo-Hookean

---

*Point Graph 2*

- 1 Right-click **Results>Inflation Pressure>Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study: Mooney-Rivlin/Solution 3 (sol3)**.
- 4 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

Mooney-Rivlin

---

*Point Graph 3*

- 1 Right-click **Results>Inflation Pressure>Point Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study: Ogden/Solution 4 (sol4)**.

4 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

Ogden

---

*Point Graph 4*

1 Right-click **Results>Inflation Pressure>Point Graph 3** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study: Varga/Solution 5 (sol5)**.

4 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

Varga

---

*Point Graph 5*

1 Right-click **Results>Inflation Pressure>Point Graph 4** and choose **Duplicate**.

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

3 In the **Expression** text field, type  $2 * (H/Ri) * ((6.3e5[Pa] * (\text{stretch}^{(1.3-3)} - \text{stretch}^{(-2*1.3-3)})) + (0.012e5[Pa] * (\text{stretch}^{(5-3)} - \text{stretch}^{(-2*5-3)})) - (0.1e5[Pa] * (\text{stretch}^{(-2-3)} - \text{stretch}^{(2*2-3)}))) / 100$ .

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **Black**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

7 In the **Number** text field, type 40.

8 Click to expand the **Legends** section. In the table, enter the following settings:

---

**Legends**

Analytical

---

9 On the **Inflation Pressure** toolbar, click **Plot**.

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

*Inflation Pressure*

To reproduce [Figure 3](#), proceed as follows.

### *Inflation Pressure 1*

- 1 In the **Model Builder** window, under **Results** right-click **Inflation Pressure** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type First principal stress in the **Label** text field.
- 3 Click to expand the **Title** section. In the **Title** text area, type First principal stress vs. prescribed stretch..
- 4 Locate the **Plot Settings** section. In the **y-axis label** text field, type First principal stress (MPa).

### *First principal stress*

- 1 Locate the **Axis** section. Select the **Manual axis limits** check box.
- 2 In the **y maximum** text field, type 40.

### *Point Graph 1*

- 1 In the **Model Builder** window, expand the **Results>First principal stress** node, then click **Point Graph 1**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `mbrn.sp1`.
- 4 From the **Unit** list, choose **MPa**.

### *Point Graph 2*

- 1 In the **Model Builder** window, under **Results>First principal stress** click **Point Graph 2**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `mbrn.sp1`.
- 4 From the **Unit** list, choose **MPa**.

### *Point Graph 3*

- 1 In the **Model Builder** window, under **Results>First principal stress** click **Point Graph 3**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `mbrn.sp1`.
- 4 From the **Unit** list, choose **MPa**.

### *Point Graph 4*

- 1 In the **Model Builder** window, under **Results>First principal stress** click **Point Graph 4**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `mbrn.sp1`.

4 From the **Unit** list, choose **MPa**.

#### *Point Graph 5*

1 In the **Model Builder** window, under **Results>First principal stress** click **Point Graph 5**.

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

3 In the **Expression** text field, type  $((6.3e5[\text{Pa}] * (\text{stretch}^{(1.3)} - \text{stretch}^{(-2*1.3)})) + (0.012e5[\text{Pa}] * (\text{stretch}^{(5)} - \text{stretch}^{(-2*5)})) - (0.1e5[\text{Pa}] * (\text{stretch}^{(-2)} - \text{stretch}^{(2*2)})))$ .

4 From the **Unit** list, choose **MPa**.

5 On the **First principal stress** toolbar, click **Plot**.

Finally, to reproduce [Figure 4](#), proceed as follows.

#### *ID Plot Group 5*

1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.

2 In the **Settings** window for **ID Plot Group**, type `Deformed thickness` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study: Neo-Hookean/ Solution 2 (sol2)**.

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

5 In the **Title** text area, type `Comparison of deformed thickness`.

#### *Point Graph 1*

1 Right-click **Deformed thickness** and choose **Point Graph**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Membrane> Physical properties>mbrn.ddef - Deformed thickness**.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Global Definitions>Parameters>stretch - Applied stretch**.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

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

9 In the table, enter the following settings:

---

**Legends**

---

Membrane

---

*Global 1*

- 1 In the **Model Builder** window, under **Results** right-click **Deformed thickness** and choose **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
int1(stretch)	m	Deformed thickness

- 4 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Global Definitions>Parameters>stretch - Applied stretch**.
- 5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 6 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
- 7 In the table, enter the following settings:

---

**Legends**

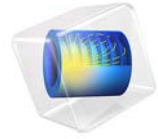
---

Solid Mechanics

---

- 8 On the **Deformed thickness** toolbar, click **Plot**.
- 9 Click the **Zoom Extents** button on the **Graphics** toolbar.





# Plastic Deformation During the Expansion of a Biomedical Stent

## Introduction

---

Percutaneous transluminal angioplasty with stenting is a widely spread method for the treatment of atherosclerosis. During the procedure, a stent is deployed into the artery by using a balloon as an expander. Once the balloon-stent package is in place, the balloon is inflated to expand the stent. The balloon is then deflated and removed, but the stent remains expanded to act as a scaffold, keeping the blood vessel open.

Stent design is of significance for this procedure, since serious damage can be inflicted to the artery during the expansion procedure. One of the most common defect is the non-uniform deformation of the stent, where the ends expand more than the middle section, phenomenon which is also called *dogboning*. Foreshortening of the stent can also damage the artery, and it could make the positioning difficult.

The dogboning is defined according to

$$\text{dogboning} = \frac{r_{\text{distal}} - r_{\text{central}}}{r_{\text{distal}}}$$

where  $r_{\text{distal}}$  and  $r_{\text{central}}$  are the radii at the end and middle of the stent, respectively.

The foreshortening is defined as

$$\text{foreshortening} = \frac{L_0 - L_{\text{load}}}{L_0}$$

here,  $L_0$  is the original length of the stent and  $L_{\text{load}}$  is the deformed length of the stent.

Other common parameters in stent design are the longitudinal and radial recoil. These parameters give information on the stent behavior when removing the inflated balloon.

The longitudinal recoil is defined as

$$L_{\text{recoil}} = \frac{L_{\text{load}} - L_{\text{unload}}}{L_{\text{load}}}$$

here,  $L_{\text{unload}}$  is the length of the stent once the balloon is removed, and  $L_{\text{load}}$  is the length of the stent when the balloon is fully inflated.

The radial recoil can be defined as follow

$$R_{\text{recoil}} = \frac{R_{\text{load}} - R_{\text{unload}}}{R_{\text{load}}}$$

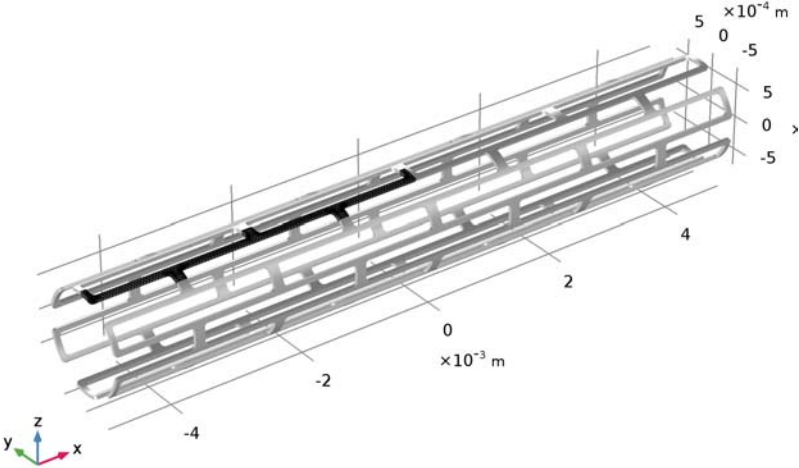
here,  $R_{\text{unload}}$  is the radius of the stent once the balloon is removed, and  $R_{\text{load}}$  is the radius of the stent when the balloon is fully inflated.

To check the viability of a stent design, you can study the deformation process under the influence of the radial pressure that expands the stent. With this example you can both monitor the dogboning and foreshortening effects, and draw conclusions on how to change the geometry design parameters for optimum performance.

*Model Definition*

The model studies the Palmaz-Schatz stent model. Due to the stent’s circumferential and longitudinal symmetry, it is possible to model only one twenty-fourth of the geometry. [Figure 1](#) shows the geometry used in the study, represented with the meshed domain.

t(103)=1.02 s Surface: Total displacement (m) Surface: Total displacement (m)



*Figure 1: reduced geometry used in the study (meshed) and full stent geometry.*

The main focus of the study consists in the stress evaluation in the stent. The angioplasty balloon is assumed to stretch with a maximum expansion radius of 2 mm.

## **MATERIAL**

The stent is made of stainless steel. The material parameters are given in the following table.

<b>MATERIAL PROPERTY</b>	<b>VALUE</b>
Young's modulus	193 [GPa]
Poisson's ratio	0.27
Initial yield stress	207 [MPa]
Isotropic tangent modulus	692 [MPa]

## **LOADS**

Apply a radial outward pressure on the inner surface of the stent to represent the balloon expansion.

## *Results and Discussion*

---

The stent is expanded from an original diameter of 0.74 mm to a diameter of 2 mm in the middle section.

[Figure 2](#) shows the stress distribution at maximum balloon inflation. [Figure 3](#) shows the residual stress after the balloon deflation.

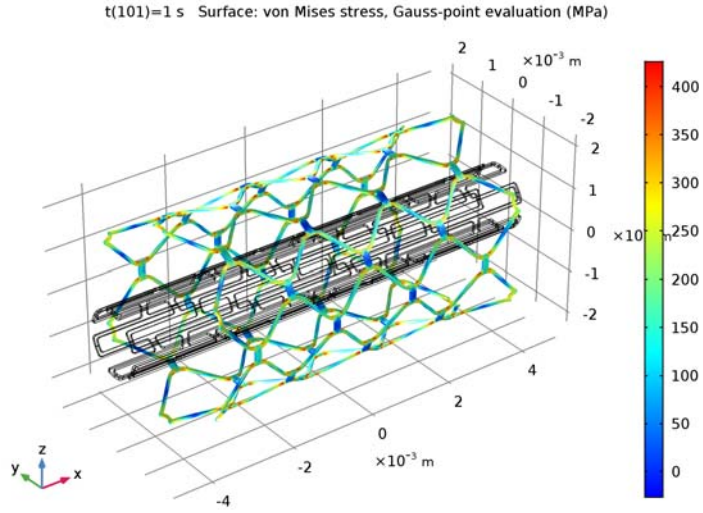


Figure 2: Maximum stress in the stent during the balloon inflation.

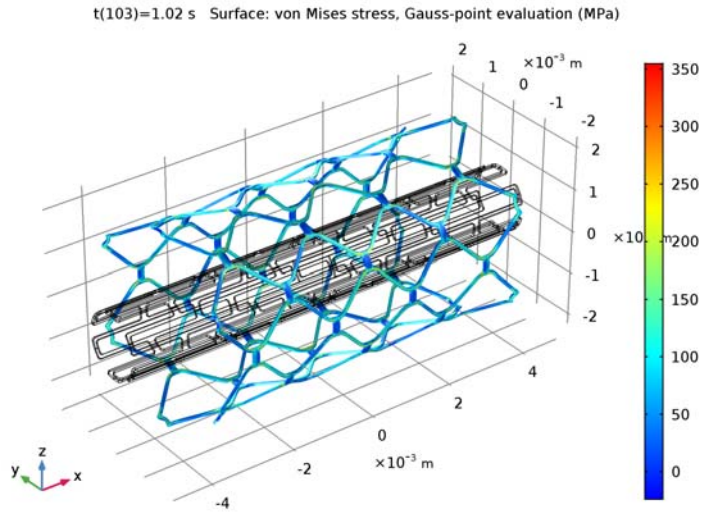


Figure 3: Remanent stress in the stent after balloon deflation.

Figure 4 shows the effective plastic strains at the maximum balloon inflation.

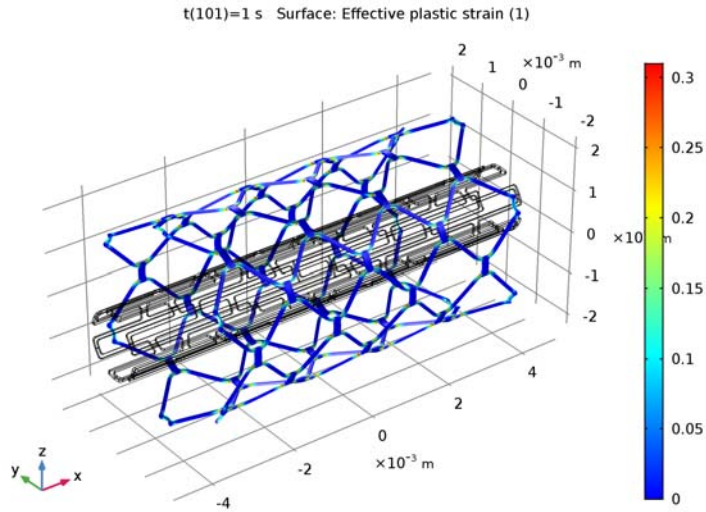


Figure 4: Effective plastic strain after stent deformation.

In Figure 5, you can see the evolution of the dogboning and foreshortening effects with respect to the pressure during the balloon inflation.

The longitudinal recoil is about  $-0.9\%$ , the distal radial recoil is about  $0.4\%$ , and the central radial recoil is about  $0.7\%$ .

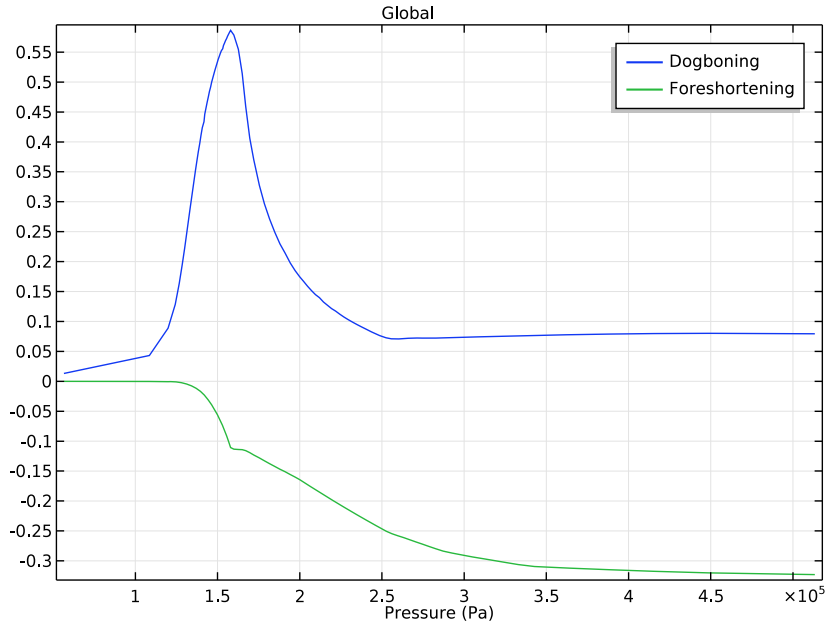


Figure 5: Stent dogboning (blue) and foreshortening (green) versus pressure inside the angioplasty balloon.

### Notes About the COMSOL Implementation

The maximum radius of the angioplasty balloon is represented with a step function: the pressure is applied as long as the stent’s inner radius is lower than the maximum balloon radius. Above this limit the pressure is set to zero.

For a highly nonlinear problem like this, the choice of the continuation parameter can improve the convergence during the computation of the solution. A displacement control parameter is usually better than a load parameter. In this example, the average displacement of the stent’s inner radius is prescribed, and a global equation is used computes the corresponding applied pressure load.

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Plasticity/biomedical\_stent

## *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 **3D**.
- 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 **Preset Studies>Stationary**.
- 6 Click **Done**.

### **GEOMETRY I**

#### *Import 1 (imp1)*

- 1 On the **Home** toolbar, click **Import**.
- 2 In the **Settings** window for **Import**, locate the **Import** section.
- 3 From the **Source** list, choose **COMSOL Multiphysics file**.
- 4 Click **Browse**.
- 5 Browse to the model's Application Libraries folder and double-click the file `biomedical_stent.mphbin`.
- 6 Click **Import**.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

### **SOLID MECHANICS (SOLID)**

#### *Linear Elastic Material 1*

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

#### *Plasticity 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Plasticity**.
- 2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.
- 3 From the **Plasticity model** list, choose **Large plastic strains**.

## 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
Young's modulus	E	193 [GPa]	Pa	Basic
Poisson's ratio	nu	0.27		Basic
Density	rho	7050	kg/m <sup>3</sup>	Basic
Initial yield stress	sigmags	207 [MPa]	Pa	Elastoplastic material model
Isotropic tangent modulus	Et	692 [MPa]	Pa	Elastoplastic material model

## DEFINITIONS

### Step 1 (step1)

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Step**.
- 2 In the **Settings** window for **Step**, locate the **Parameters** section.
- 3 In the **Location** text field, type  $2e-3$ .
- 4 In the **From** text field, type 1.
- 5 In the **To** text field, type 0.
- 6 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type  $1e-5$ .

### Variables 1

- 1 On the **Home** toolbar, click **Variables** and choose **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
r	$\text{sqrt}(y^2+z^2)$	m	Radial distance from x-axis

### *Average 1 (aveop1)*

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.
- 2 In the **Settings** window for **Average**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Edge**.
- 4 Select Edge 28 only.
- 5 Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

### *Piecewise 1 (pw1)*

- 1 On the **Definitions** toolbar, click **Piecewise**.
- 2 In the **Settings** window for **Piecewise**, type r0 in the **Function name** text field.
- 3 Locate the **Definition** section. In the **Argument** text field, type t.
- 4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	1	$(2e-3-7.1e-4)*t+7.1e-4$
1	2	$(2e-3-7.1e-4)*(1-t)+2e-3$

- 5 Locate the **Units** section. In the **Arguments** text field, type s.
- 6 In the **Function** text field, type m.
- 7 Click **Plot**.

## **SOLID MECHANICS (SOLID)**

### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 5, 12, 18, 24, 30, and 31 only.

### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 4 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 From the **Load type** list, choose **Pressure**.
- 5 In the  $p$  text field, type  $p*\text{step1}(r/1[\text{m}])$ .
- 6 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

### Global Equations 1

- 1 On the **Physics** toolbar, click **Global** and choose **Global Equations**.
- 2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.
- 3 In the table, enter the following settings:

Name	$f(u,ut,utt,t)$ (l)	Initial value (u_0) (l)	Initial value (u_t0) (l/s)	Description
p	aveop1(r) - r0(t)	0	0	Pressure

- 4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.
- 5 In the **Physical Quantity** dialog box, In the associated text field, type id:pressure.
- 6 Click **Filter**.
- 7 In the tree, select **Transport>Pressure (Pa)**.
- 8 Click **OK**.
- 9 In the **Settings** window for **Global Equations**, locate the **Units** section.
- 10 Click **Select Source Term Quantity**.
- 11 In the **Physical Quantity** dialog box, In the associated text field, type id:length.
- 12 Click **Filter**.
- 13 In the tree, select **General>Length (m)**.
- 14 Click **OK**.

### MESH 1

#### Free Triangular 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.
- 2 Select Boundary 3 only.

#### Size 1

- 1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 5 In the associated text field, type  $4.5e-5$ .
- 6 Select the **Minimum element size** check box.
- 7 In the associated text field, type  $4e-6$ .

**8** Select the **Maximum element growth rate** check box.

**9** In the associated text field, type 1.4.

**10** Select the **Curvature factor** check box.

**11** In the associated text field, type 0.3.

#### *Distribution 1*

**1** In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

**2** Right-click **Swept 1** and choose **Distribution**.

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

**4** In the **Number of elements** text field, type 2.

**5** Click **Build All**.

#### **DEFINITIONS**

Create variables for the results processing.

#### *Integration 1 (intop1)*

**1** On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

**2** In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

**4** Select Point 57 only.

**5** In the **Operator name** text field, type central.

**6** Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

#### *Integration 2 (intop2)*

**1** On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

**2** In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

**4** Select Point 3 only.

**5** In the **Operator name** text field, type distal.

**6** Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

#### *Variables 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Variables 1**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
dogboning	$(\text{distal}(r) - \text{central}(r)) / \text{distal}(r)$		Dogboning
length	$2 * \text{abs}(\text{distal}(x) - \text{central}(x))$	m	Length of the deformed stent
L0	$2 * \text{abs}(\text{distal}(X) - \text{central}(X))$	m	Length of the undeformed stent
foreshortening	$(\text{length} - L0) / \text{length}$		Foreshortening

## GLOBAL DEFINITIONS

### Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
t	0	0	Time

## STUDY I

### Step 1: Stationary

- 1 In the **Model Builder** window, under **Study I** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, click to expand the **Results while solving** section.
- 3 Locate the **Results While Solving** section. Select the **Plot** check box.  
Set up an auxiliary continuation sweep for the t parameter.
- 4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 5 Click **Add**.
- 6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t	range(0, 1e-2, 1.5)	s

### Solution 1 (sol1)

- 1 On the **Study** toolbar, click **Show Default Solver**.

- 2 In the **Model Builder** window, expand the **Solution I (sol1)** node.
- 3 In the **Model Builder** window, expand the **Study I>Solver Configurations>Solution I (sol1)>Dependent Variables I** node, then click **Pressure (comp1.ODE1)**.
- 4 In the **Settings** window for **State**, locate the **Scaling** section.
- 5 From the **Method** list, choose **Manual**.
- 6 In the **Scale** text field, type  $1e6$ .  
Add a stop condition to prevent the computed pressure from becoming negative.
- 7 In the **Model Builder** window, expand the **Study I>Solver Configurations>Solution I (sol1)>Stationary Solver I** node.
- 8 Right-click **Parametric I** and choose **Stop Condition**.
- 9 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.
- 10 Click **Add**.
- 11 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
comp1.p<0	true	√	Stop expression 1

Specify that the solution is to be stored just before the stop condition is reached.

- 12 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step before stop**.
- 13 On the **Study** toolbar, click **Compute**.

## RESULTS

### *Data Sets*

Use mirror 3D and sector 3D data sets to display the solution on the entire geometry.

### *Mirror 3D 2*

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.
- 2 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.
- 3 In the **Settings** window for **Mirror 3D**, locate the **Data** section.
- 4 From the **Data set** list, choose **Mirror 3D 1**.
- 5 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

### *Sector 3D 1*

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Sector 3D**.
- 2 In the **Settings** window for **Sector 3D**, locate the **Data** section.

- 3 From the **Data set** list, choose **Mirror 3D 2**.
- 4 Locate the **Axis Data** section. In row **Point 2**, set **x** to 1 and **z** to 0.
- 5 Locate the **Symmetry** section. In the **Number of sectors** text field, type 6.

#### *Stress (solid)*

The default plot group shows von Mises stress and contour levels of effective plastic strain. For better clarity, plot them separately.

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Data set** list, choose **Sector 3D 1**.

#### *Plastic strain*

- 1 In the **Model Builder** window, expand the **Stress (solid)** node.
- 2 Right-click **Plastic strain** and choose **Disable**.

#### *Surface 1*

- 1 In the **Model Builder** window, under **Results>Stress (solid)** click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.
- 4 On the **Stress (solid)** toolbar, click **Plot**.
- 5 Click **Plot Previous** twice to plot the maximum stress.
- 6 On the **Stress (solid)** toolbar, click **Plot**.

#### *Stress (solid) 1*

- 1 In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Duplicate**.
- 2 In the **Settings** window for **3D Plot Group**, type Effective Plastic Strain in the **Label** text field.

#### *Surface 1*

- 1 In the **Model Builder** window, expand the **Results>Effective Plastic Strain** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Effective plastic strain**.

#### *Effective Plastic Strain*

- 1 In the **Model Builder** window, under **Results** click **Effective Plastic Strain**.

2 On the **Effective Plastic Strain** toolbar, click **Plot**.

#### *ID Plot Group 3*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Dogboning and Foreshortening in the **Label** text field.
- 3 Locate the **Data** section. From the **Time selection** list, choose **From list**.
- 4 In the Times list select all the solution steps between 0 and 1.

#### *Global 1*

- 1 Right-click **Dogboning and Foreshortening** and choose **Global**.
- 2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>dogboning - Dogboning**.
- 3 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>foreshortening - Foreshortening**.
- 4 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Solid Mechanics>p - Pressure**.
- 5 On the **Dogboning and Foreshortening** toolbar, click **Plot**.

Evaluate the length recoil, the distal radial recoil, and the central radial recoil.

#### *Global Evaluation 1*

- 1 On the **Results** toolbar, click **Global Evaluation**.
- 2 In the **Settings** window for **Global Evaluation**, type Recoil Evaluation in the **Label** text field.
- 3 Locate the **Data** section. From the **Time selection** list, choose **From list**.
- 4 In the **Parameter values (t (s))** list, select **1**.
- 5 Locate the **Expressions** section. In the table, enter the following settings:

<b>Expression</b>	<b>Unit</b>	<b>Description</b>
$(\text{length-with}(103, \text{length}))/\text{length}$	1	Longitudinal recoil
$(\text{distal}(r)-\text{with}(103, \text{distal}(r)))/\text{distal}(r)$	1	Distal radial recoil
$(\text{central}(r)-\text{with}(103, \text{central}(r)))/\text{central}(r)$	1	Central radial recoil

- 6 Click **Evaluate**.

The steps below illustrate how to display the geometry as in [Figure 1](#).

#### *Surface 1*

- 1** On the **Results** toolbar, click **3D Plot Group**.
- 2** In the **Model Builder** window, right-click **3D Plot Group 4** and choose **Surface**.
- 3** In the **Settings** window for **Surface**, locate the **Data** section.
- 4** From the **Data set** list, choose **Sector 3D 1**.
- 5** Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 6** From the **Color** list, choose **Gray**.

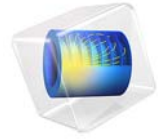
#### *3D Plot Group 4*

- 1** In the **Model Builder** window, under **Results** click **3D Plot Group 4**.
- 2** In the **Settings** window for **3D Plot Group**, type Full Geometry and Mesh in the **Label** text field.

#### *Surface 2*

- 1** Right-click **Results>Full Geometry and Mesh** and choose **Surface**.
- 2** In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
- 3** From the **Coloring** list, choose **Uniform**.
- 4** From the **Color** list, choose **Black**.
- 5** Select the **Wireframe** check box.
- 6** On the **Full Geometry and Mesh** toolbar, click **Plot**.





# Compression of an Elastoplastic Pipe

## Introduction

---

In offshore applications, it is sometimes necessary to quickly seal a pipe as part of the prevention of a blowout. This example shows a simulation, in which a circular pipe is squeezed between two flat stiff indenters.

The tutorial serves as an example of an analysis with very large plastic strains and contact.

## Model Definition

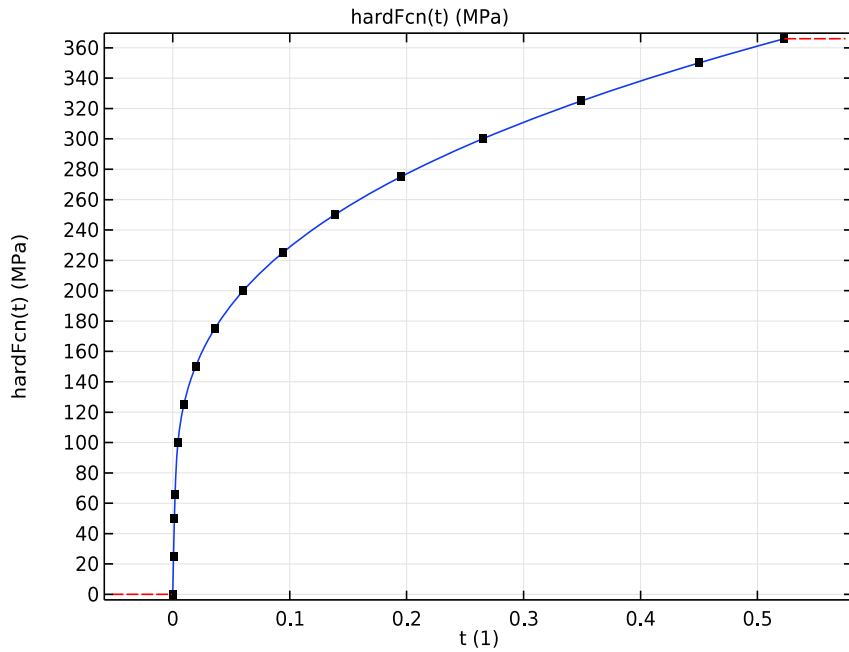
---

The pipe has an external radius,  $R_0$ , of 200 mm and a wall thickness of 25 mm. The material is a stainless steel with the following properties

TABLE 1: MATERIAL DATA

PROPERTY	VALUE
Young's modulus	195 GPa
Poisson's ratio	0.3
Yield stress	250 MPa
Ultimate tensile stress	616 MPa
Ultimate strain	0.52

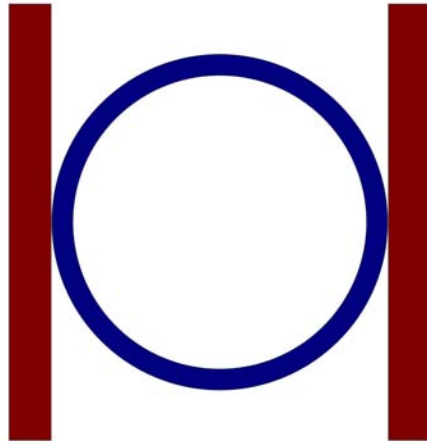
The hardening curve is available as a text file containing pairs of data (plastic strain, stress) which can be imported as a function. The function is shown in [Figure 1](#) below. This nonlinear hardening function  $\sigma_h(\epsilon_{pe})$  is defined in the Materials node by an interpolation function. The stress is measured as true (Cauchy) stress, and the strain is measured as true (logarithmic) strain. The data can thus be used directly as input to COMSOL when large strain plasticity is used. Note that if the strain is above the ultimate strain, then the curve is implicitly flat, so that deformation continues under constant stress.



*Figure 1: The hardening curve.*

The pipe is compressed between two flat indenters that can be considered as rigid. The original position of each indenter is 0.1 mm from the outer pipe wall. During the

compression of the pipe, the distance between the indenters is decreased by 300 mm, and then they are retracted to their original positions.



*Figure 2: The geometry; pipe and indenters.*

Due to the symmetries, only one quarter of the geometry needs to be modeled. The problem is considered as 2D with the plane strain assumption.

### *Results and Discussion*

---

This example exhibits extremely large plastic strains. The deformation and stress state at the maximum compression are shown in [Figure 3](#). The maximum stress displayed is slightly above the ultimate tensile stress (616 MPa). This is caused by the extrapolation of

the results from the integration points inside the elements, where the constitutive law is exactly fulfilled.

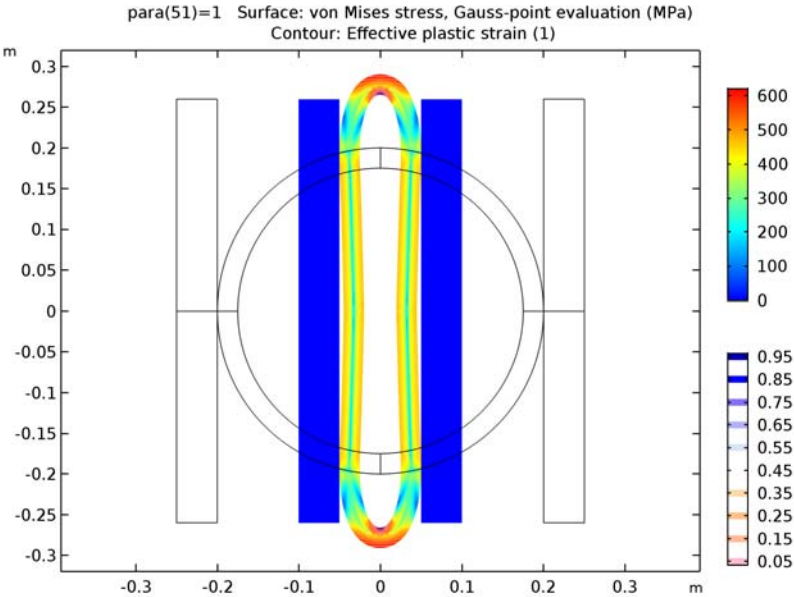


Figure 3: Effective stress at maximum compression.

The distribution of the effective plastic strains is shown in Figure 4 and Figure 5. As can be seen, the peak value (1) is far above the ultimate strain (0.52). All values above the ultimate strain are however on the inside of the pipe, where the strain state is mainly in compression. At the outer edge of the pipe, the plastic strain approximately reaches the ultimate strain. There is thus a certain risk that cracks could start forming. The values of ultimate stress and strain are related to the specimen used for the testing (usually a cylindrical bar) and cannot directly be transferred to general multiaxial conditions.

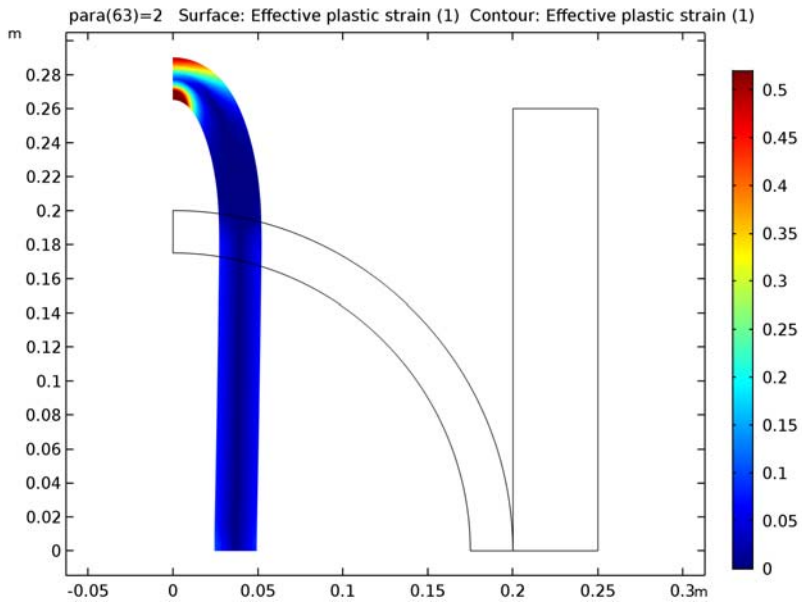


Figure 4: Effective plastic strain at maximum compression.

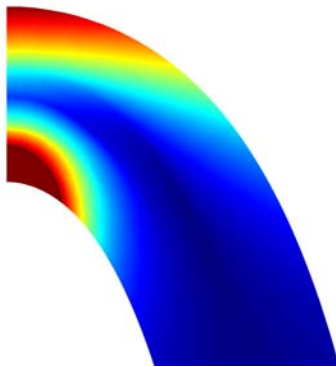
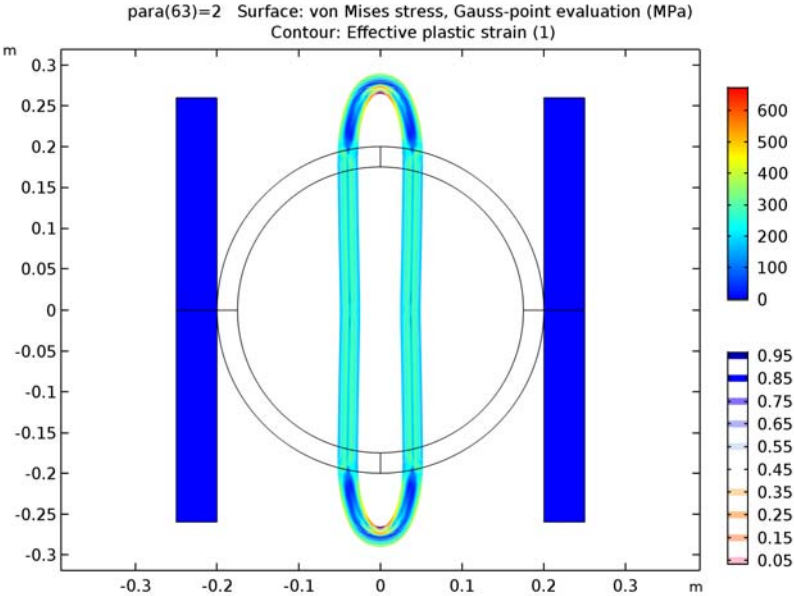


Figure 5: Effective plastic strain at maximum compression, detail. The black contour indicates the ultimate strain (0.52).

The final state after the retraction of the indenters is shown in [Figure 6](#). There is some reversed yielding during the unloading process. The springback effect is very small.



*Figure 6: Deformed shape and residual stresses at the end of the process.*

The load used to compress the pipe is computed from the reaction force in the indenter, and it is shown in [Figure 7](#).

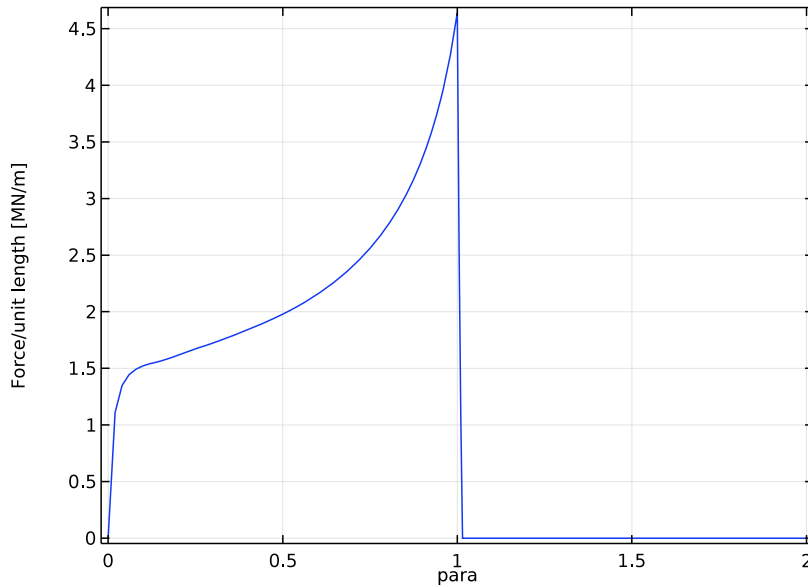


Figure 7: The applied force as function of the loading parameter.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Plasticity/compressed\_elastoplastic\_pipe

---

### *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 **Preset Studies>Stationary**.
- 6 Click **Done**.

## GLOBAL DEFINITIONS

### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
Ro	200[mm]	0.2 m	Outer radius
thic	25[mm]	0.025 m	Pipe wall thickness
Ri	Ro-thic	0.175 m	Inner radius
para	0	0	Solution parameter

Add a function for the displacement of the indenting part.

### *Interpolation 1 (int1)*

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type compr.
- 4 In the table, enter the following settings:

t	f(t)
0	0
1	0.15
2	0

- 5 Locate the **Units** section. In the **Arguments** text field, type 1.
- 6 In the **Function** text field, type m.
- 7 Click **Plot**.

## GEOMETRY I

### *Circle 1 (c1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type  $R_0$ .
- 4 In the **Sector angle** text field, type 90.

#### *Circle 2 (c2)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type  $R_i$ .
- 4 In the **Sector angle** text field, type 90.

#### *Difference 1 (dif1)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **c1** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 5 Select the object **c2** only.
- 6 Click **Build All Objects**.

#### *Rectangle 1 (r1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type  $0.05$ .
- 4 In the **Height** text field, type  $1.3 \cdot R_0$ .
- 5 Locate the **Position** section. In the **x** text field, type  $R_0 + 0.0001$ .

#### *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 On the **Geometry** toolbar, click **Build All**.
- 5 Click the **Zoom Extents** button on the **Graphics** toolbar.

## **DEFINITIONS**

#### *Contact Pair 1 (p1)*

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

- 2 Select Boundary 5 only.
- 3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.
- 4 Select the **Active** toggle button.
- 5 Select Boundary 4 only.

## **SOLID MECHANICS (SOLID)**

### *Linear Elastic Material I*

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material I**.

### *Plasticity I*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Plasticity**.
- 2 Select Domain 1 only.
- 3 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.
- 4 From the **Plasticity model** list, choose **Large plastic strains**.
- 5 Find the **Isotropic hardening model** subsection. From the list, choose **User defined**.

## **MATERIALS**

### *Material I (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:

<b>Property</b>	<b>Variable</b>	<b>Value</b>	<b>Unit</b>	<b>Property group</b>
Young's modulus	E	195 [GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	8000	kg/m <sup>3</sup>	Basic
Initial yield stress	sigmags	250 [MPa]	Pa	Elastoplastic material model

Add the hardening curve for the elastoplastic material.

- 4 In the **Model Builder** window, expand the **Material I (mat1)** node, then click **Elastoplastic material model (ElastoplasticModel)**.

- 5 In the **Settings** window for **Property Group**, locate the **Output Properties and Model Inputs** section.
- 6 Find the **Quantities** subsection. In the tree, select **Model Inputs>Effective Plastic Strain**.
- 7 Click **Add**.

#### *Interpolation 1 (int1)*

- 1 Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)>Elastoplastic material model (ElastoplasticModel)** and choose **Functions>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 Click **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `compressed_elastoplastic_pipe_stress_strain.txt`.
- 5 In the **Function name** text field, type `hardFcn`.
- 6 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.
- 7 Locate the **Units** section. In the **Arguments** text field, type `1`.
- 8 In the **Function** text field, type `MPa`.
- 9 Click **Plot**.

#### *Material 1 (mat1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 1 (mat1)**.
- 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
Hardening function	<code>sigmagh</code>	<code>hardFcn(epe)</code>	Pa	Elastoplastic material model

### **SOLID MECHANICS (SOLID)**

Detailed contact stresses are not of interest in this model. The choice of the penalty contact method then speeds up computation.

#### *Contact 1*

- 1 On the **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Contact**.
- 2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.
- 3 In the **Pairs** list, select **Contact Pair 1 (p1)**.

4 Locate the **Contact Pressure Method** section. From the list, choose **Penalty**.

#### *Symmetry 1*

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1 and 2 only.

#### *Prescribed Displacement 1*

1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Displacement**.

2 Select Domain 2 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

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

6 In the  $u_{0x}$  text field, type  $-\text{compr}(\text{para})$ .

#### **MESH 1**

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

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

3 From the **Sequence type** list, choose **User-controlled mesh**.

#### *Free Triangular 1*

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Delete**.

2 Click **Yes** to confirm.

#### *Distribution 1*

1 Right-click **Mesh 1** and choose **Mapped**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

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

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

5 Select Boundaries 5 and 6 only.

One element is enough on the indenter, since the whole domain is under displacement control.

#### *Distribution 2*

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

- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 Select Boundaries 1 and 2 only.
- 5 In the **Number of elements** text field, type 8.
- 6 In the **Element ratio** text field, type 2.
- 7 Select the **Symmetric distribution** check box.

#### *Distribution 3*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 80.
- 5 Click **Build All**.

### **STUDY 1**

#### *Step 1: Stationary*

Set up an auxiliary continuation sweep for the parameter para.

- 1 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.
- 2 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 3 Click **Add**.
- 4 In the table, enter the following settings:

Parameter name	Parameter value list
para (Solution parameter)	range(0,0.02,1) range(1.005, 0.005, 1.05) 1.1 2

- 5 Click to expand the **Results while solving** section. Locate the **Results While Solving** section. Select the **Plot** check box.
- 6 On the **Home** toolbar, click **Compute**.

### **RESULTS**

#### *Stress (solid)*

Mirror the solution twice to get a full view of the pipe.

#### *Mirror 2D 2*

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 2D**.

- 2 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 2D**.
- 3 In the **Settings** window for **Mirror 2D**, locate the **Data** section.
- 4 From the **Data set** list, choose **Mirror 2D 1**.
- 5 Locate the **Axis Data** section. In row **Point 2**, set **x** to 1 and **y** to 0.

#### *Stress (solid)*

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 3 From the **Data set** list, choose **Mirror 2D 2**.

#### *Contact pressure*

- 1 In the **Model Builder** window, expand the **Stress (solid)** node.
- 2 Right-click **Contact pressure** and choose **Disable**.

#### *Surface 1*

- 1 In the **Model Builder** window, under **Results>Stress (solid)** click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.

#### *Stress (solid)*

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 On the **Stress (solid)** toolbar, click **Plot**.  
Plot the stresses at the maximum compression. This gives [Figure 3](#).
- 3 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 4 From the **Parameter value (para)** list, choose **1**.
- 5 On the **Stress (solid)** toolbar, click **Plot**.  
Create an animation of the compression process.
- 6 Click **Animation** and choose **Player**.

#### *Animation 1*

Create a graph of the applied force as function of the compression.

#### *Surface Integration 1*

- 1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.
- 2 Select Domain 2 only.
- 3 In the **Settings** window for **Surface Integration**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit
-solid.RFx*2/1E6	N

5 Click **Evaluate**.

#### TABLE

- 1 Go to the **Table** window.
- 2 Click **Table Graph** in the window toolbar.

#### RESULTS

##### *ID Plot Group 2*

- 1 In the **Model Builder** window, under **Results** click **ID Plot Group 2**.
- 2 In the **Settings** window for **ID Plot Group**, type **Compression Force** in the **Label** text field.
- 3 Locate the **Plot Settings** section. Select the **y-axis label** check box.
- 4 In the associated text field, type **Force/unit length [MN/m]**.
- 5 On the **Compression Force** toolbar, click **Plot**.

##### *Surface 1*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
- 2 In the **Model Builder** window, right-click **2D Plot Group 3** and choose **Surface**.
- 3 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Effective plastic strain**.  
Set the plot scale limit to the ultimate strain.
- 4 Click to expand the **Range** section. Select the **Manual color range** check box.
- 5 In the **Maximum** text field, type **0.52**.

##### *Deformation 1*

- 1 Right-click **Results>2D Plot Group 3>Surface 1** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 Select the **Scale factor** check box.
- 4 In the associated text field, type **1**.

Add a contour showing where the ultimate strain is exceeded.

### *Contour 1*

- 1** In the **Model Builder** window, under **Results** right-click **2D Plot Group 3** and choose **Contour**.
- 2** In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Effective plastic strain**.
- 3** Locate the **Levels** section. From the **Entry method** list, choose **Levels**.
- 4** In the **Levels** text field, type 0.52.
- 5** Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 6** From the **Color** list, choose **Black**.
- 7** Clear the **Color legend** check box.

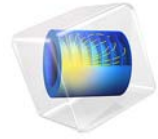
### *Deformation 1*

- 1** Right-click **Results>2D Plot Group 3>Contour 1** and choose **Deformation**.
- 2** In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3** Select the **Scale factor** check box.
- 4** In the associated text field, type 1.
- 5** On the **2D Plot Group 3** toolbar, click **Plot**.
- 6** Click the **Zoom Extents** button on the **Graphics** toolbar.

### *2D Plot Group 3*

- 1** In the **Model Builder** window, under **Results** click **2D Plot Group 3**.
- 2** In the **Settings** window for **2D Plot Group**, type **Plastic Strain** in the **Label** text field.





# Die Forming

## Introduction

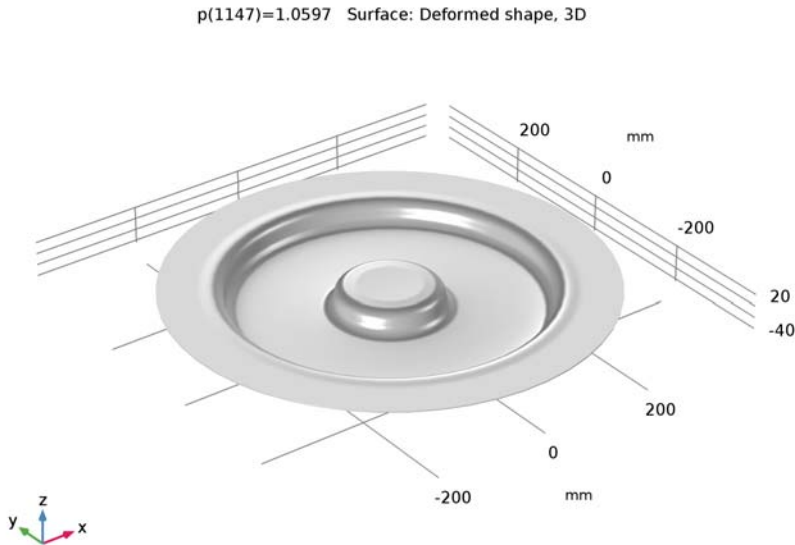
---

Die forming is a widespread sheet-metal forming manufacturing process. The workpiece, usually a metal sheet, is permanently reshaped around a die through plastic deformation by forming and drawing processes.

Simulations can be carried out in order to avoid cracks, tears, wrinkles, too much thinning and stretching. They are also useful to estimate and overcome the springback phenomenon: once the forming process is done and the forming tools are removed, the workpiece attempts to partially recover its initial shape through relaxation of the elastic stresses.

The springback can cause the formed blank to get an unexpected state of warping. To cope with this effect, it is possible to over-bend the sheet. The die, the punch and the blank-holder must be manufactured to incorporate this effect.

In this model, a flat metal sheet made of aluminum is pressed onto a curved die by a similarly shaped punch. Both the forming and the springback phenomena are modeled. From a simulation point of view, the problem is severely nonlinear due to contact, large strain plasticity, and geometric nonlinearity. [Figure 1](#) shows the shape after forming.



*Figure 1: Deformed shape after the forming process.*

## Model Definition

The model geometry is shown in Figure 2. Due to the axial symmetry, a 2D axisymmetric formulation can be used. The die and the blank-holder together clamp the blank to be reshaped, while the punch performs the drawing, stretching and bending.

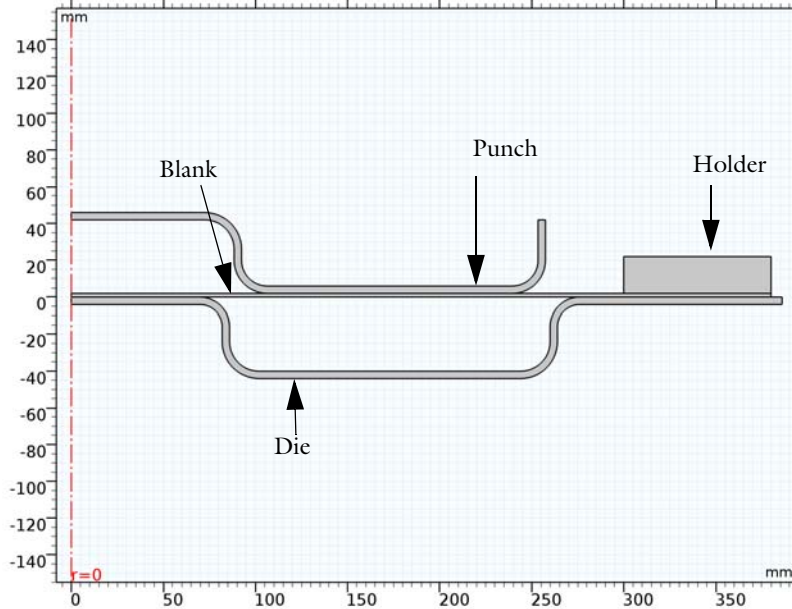


Figure 2: Forming tools setup.

The analysis is carried out in two steps. First, the punch is pushed towards the sheet through a displacement of 40 mm. This step aims to simulate the forming and drawing processes. A second step is used to perform the springback analysis. The punch is released progressively to model the springback phenomenon.

The sheet is made of aluminum. An isotropic elastoplastic material with user-defined isotropic hardening and large plastic strain formulation is used to characterize the plastic deformation.

The die and the punch are made of structural steel, so they are much rigid than the aluminum sheet. The die and holder are fixed, and the punch is deforms the blank with a prescribed vertical displacement which is ramped linearly.

*Results and Discussion*

Figure 3 shows the residual stress after the release.

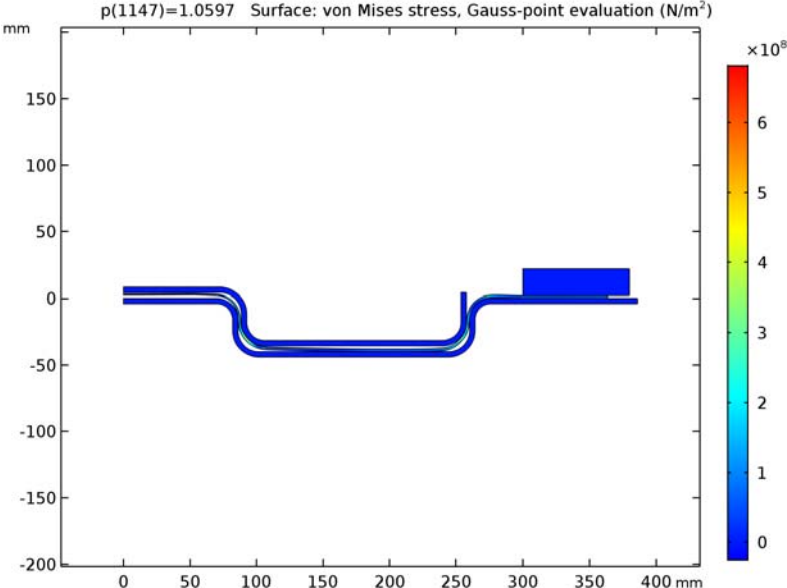


Figure 3: Residual stress after release.

Figure 4 shows the plastic strain. As expected, large plastic strains occur. At the most strained location the plastic strain exceeds 60%.

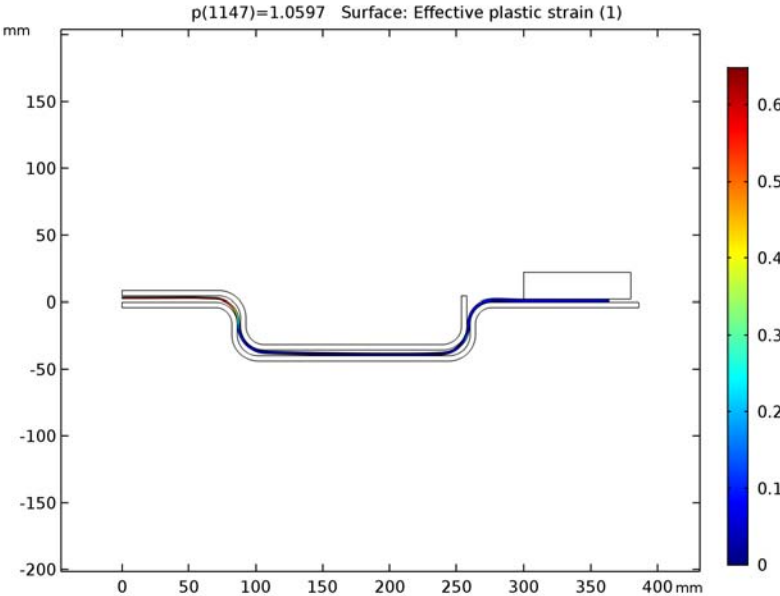


Figure 4: Plastic strains after release.

Figure 5 shows the variation of the thickness of the final deformed part. The initial blank thickness is 0.2 mm. The maximum thinning is observed at the center of the part.

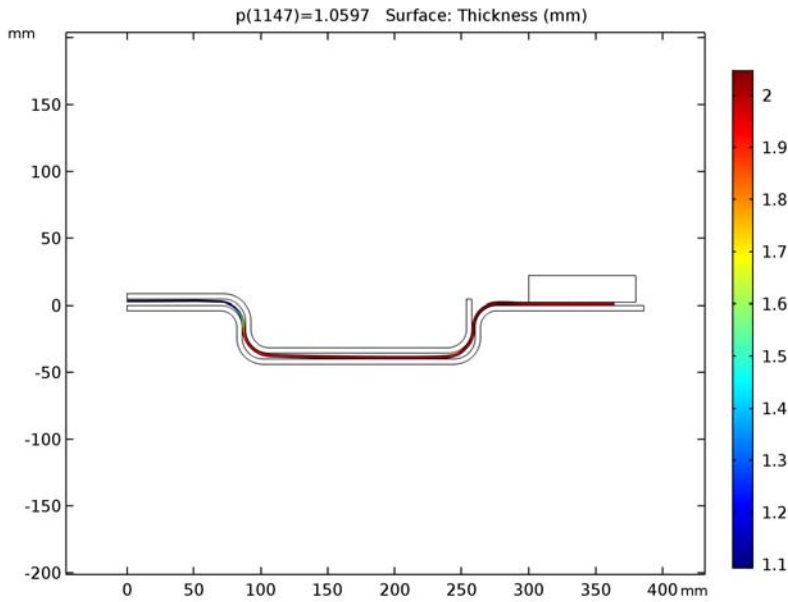


Figure 5: Thickness of the part after forming.

A comparison of the maximum radial position between the forming and the release stage gives an idea of the springback.

### *Notes About the COMSOL Implementation*

The strong nonlinearity of the problem due to contact, plasticity, and geometric nonlinearity may cause some convergence issues. Some useful tips for improving the convergence are given below.

- Choose carefully source and destination in contact pairs. Based on stiffness of the metal parts, the sheet should be the destination. The blank/sheet is meshed finely compared to the source. The curved boundaries of the source are also finely meshed compared to the flat boundaries
- Inclusion of plasticity in a contact problems makes convergence either difficult or considerably slower with Augmented Lagrangian contact pressure method, hence penalty contact pressure method is used.

- The parametric sweep must use steps that are small enough to avoid divergence of the solution. If the parametric step chosen by the solver is too large, the initial guess of a solution based on previous computed solution could be too far from the solution in the current step. Use **Tuning of step size** to manually adjust the parametric steps.
- Because of the small continuation parameter steps, you can set the parametric predictor to constant in order to include the punching and releasing steps in the same study.
- To compute the thickness of the deformed shape a projection coupling operator is used to define the variable *th*. Such an operator can integrate any expression across a domain in the spatial frame. Here integrating '1' across the blank returns its deformed thickness.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Plasticity/die\_forming

---

### *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 Axisymmetric**.
- 2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 3 Click **Add**.
- 4 In the **Added physics interfaces** tree, select **Solid Mechanics (solid)**.
- 5 Click **Study**.
- 6 In the **Select Study** tree, select **Preset Studies>Stationary**.
- 7 Click **Done**.

#### **GLOBAL DEFINITIONS**

##### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
sigma_y0	124[MPa]	1.24E8 Pa	Initial yield stress
U_p	40[mm]	0.04 m	Punch displacement
R0	0.69	0.69	Lankford's coefficient r0
R45	0.69	0.69	Lankford's coefficient r45
R90	0.69	0.69	Lankford's coefficient 90
F	$R0 / (R90 * (R0 + 1)) / \text{sigma\_y0}^2$	3.8483E-17 m <sup>2</sup> ·s <sup>4</sup> /kg <sup>2</sup>	Hill's coefficient
G	$1 / (R0 + 1) / \text{sigma\_y0}^2$	3.8483E-17 m <sup>2</sup> ·s <sup>4</sup> /kg <sup>2</sup>	Hill's coefficient
H	$R0 / (R0 + 1) / \text{sigma\_y0}^2$	2.6553E-17 m <sup>2</sup> ·s <sup>4</sup> /kg <sup>2</sup>	Hill's coefficient
L	0	0	Hill's coefficient
M	0	0	Hill's coefficient
N	0	0	Hill's coefficient
p	0 [mm]	0 m	Parameter

## GEOMETRY 1

Set unit of geometry to millimeters (mm).

- 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**.

### Rectangle 1 (r1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 66.5.
- 4 In the **Height** text field, type 4.
- 5 Locate the **Position** section. In the **r** text field, type 106.5.
- 6 In the **z** text field, type 2.

### Circle 1 (c1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 18.
- 4 Locate the **Position** section. In the **r** text field, type 106.5.
- 5 In the **z** text field, type 20.
- 6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	4

### Rectangle 2 (r2)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 4.
- 4 In the **Height** text field, type 6.
- 5 Locate the **Position** section. In the **r** text field, type 88.5.
- 6 In the **z** text field, type 20.

### Circle 2 (c2)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 20.
- 4 Locate the **Position** section. In the **r** text field, type 72.5.
- 5 In the **z** text field, type 26.
- 6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	4

### 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 Domains 1 and 3–5 only.

5 On the object **c2**, select Domains 1–3 and 5 only.

6 Click **Build Selected**.

#### *Rectangle 3 (r3)*

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 72.5.

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

5 Locate the **Position** section. In the **r** text field, type 0.

6 In the **z** text field, type 42.

#### *Mirror 1 (mir1)*

1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the objects **r1**, **del1(1)**, and **r2** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Point on Line of Reflection** section. In the **r** text field, type 173.

#### *Rectangle 4 (r4)*

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 4.

4 In the **Height** text field, type 16.

5 Locate the **Position** section. In the **r** text field, type 253.5.

6 In the **z** text field, type 26.

7 Click **Build All Objects**.

Remove the internal boundaries while making union.

#### *Union 1 (uni1)*

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 Click in the **Graphics** window and then press Ctrl+A to select all objects.

4 In the **Settings** window for **Union**, locate the **Union** section.

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

6 Click **Build All Objects**.

### Rectangle 5 (r5)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 71.
- 4 In the **Height** text field, type 4.
- 5 Locate the **Position** section. In the **z** text field, type -44.
- 6 In the **r** text field, type 102.

### Circle 3 (c3)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 20.
- 4 Locate the **Position** section. In the **r** text field, type 102.
- 5 In the **z** text field, type -24.
- 6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	4

### Rectangle 6 (r6)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 4.
- 4 In the **Height** text field, type 8.
- 5 Locate the **Position** section. In the **z** text field, type -24.
- 6 In the **r** text field, type 82.

### Circle 4 (c4)

- 1 In the **Model Builder** window, under **Component 1 (comp 1)>Geometry 1** right-click **Circle 3 (c3)** and choose **Duplicate**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 16.
- 4 Locate the **Position** section. In the **r** text field, type 70.
- 5 In the **z** text field, type -16.

### *Rectangle 7 (r7)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 70.
- 4 In the **Height** text field, type 4.
- 5 Locate the **Position** section. In the **z** text field, type -4.
- 6 Click **Build All Objects**.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

### *Delete Entities 2 (del2)*

- 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 **c3**, select Domains 1 and 3–5 only.
- 5 On the object **c4**, select Domains 1–3 and 5 only.
- 6 Click **Build Selected**.

### *Mirror 2 (mir2)*

- 1 On the **Geometry** toolbar, click **Transforms** and choose **Mirror**.
- 2 In the **Settings** window for **Mirror**, locate the **Input** section.
- 3 Select the **Keep input objects** check box.
- 4 Select the objects **r5**, **del2(1)**, **del2(2)**, **r6**, and **r7** only.
- 5 Locate the **Point on Line of Reflection** section. In the **r** text field, type 173.

### *Rectangle 8 (r8)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Position** section.
- 3 In the **r** text field, type 346.
- 4 In the **z** text field, type -4.
- 5 Locate the **Size and Shape** section. In the **Width** text field, type 40.
- 6 In the **Height** text field, type 4.
- 7 Click **Build Selected**.
- 8 Click **Build Selected**.

Remove the internal boundaries while making union.

### *Union 2 (uni2)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
- 2 Click the **Zoom Extents** button on the **Graphics** toolbar.
- 3 Select the objects **r8**, **r5**, **mir2(1)**, **del2(1)**, **mir2(2)**, **del2(2)**, **mir2(3)**, **mir2(4)**, **r6**, **r7**, and **mir2(5)** only.
- 4 In the **Settings** window for **Union**, locate the **Union** section.
- 5 Clear the **Keep interior boundaries** check box.
- 6 Click **Build Selected**.

### *Rectangle 9 (r9)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 80.
- 4 In the **Height** text field, type 20.
- 5 Locate the **Position** section. In the **r** text field, type 300.
- 6 In the **z** text field, type 2.
- 7 Click **Build Selected**.

### *Rectangle 10 (r10)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 380.
- 4 In the **Height** text field, type 2.
- 5 Click **Build All Objects**.
- 6 Click **Build Selected**.

### *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 From the **Pair type** list, choose **Contact pair**.
- 5 On the **Geometry** toolbar, click **Build All**.

## DEFINITIONS

### *Interpolation 1 (int1)*

On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.

## GLOBAL DEFINITIONS

### *Interpolation 1 (int1)*

- 1 In the **Settings** window for **Interpolation**, type Prescribed Punch Displacement in the **Label** text field.
- 2 Locate the **Definition** section. In the **Function name** text field, type U\_punch.
- 3 In the table, enter the following settings:

t	f(t)
0	0
1	-U_p
2	0

- 4 Locate the **Units** section. In the **Arguments** text field, type 1.
- 5 In the **Function** text field, type m.

### *Interpolation 2 (int2)*

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, type Hardening Function in the **Label** text field.
- 3 Locate the **Definition** section. In the **Function name** text field, type sigma\_h.
- 4 In the table, enter the following settings:

t	f(t)
0	0
0.02	43
0.05	76
0.1	103
0.15	115
0.2	127
0.3	129
0.4	129.3
0.5	129.4

- 5 Locate the **Units** section. In the **Arguments** text field, type 1.
- 6 In the **Function** text field, type MPa.

#### *Integration 1 (intop1)*

On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

### **DEFINITIONS**

#### *Integration 1 (intop1)*

- 1 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 2 From the **Geometric entity level** list, choose **Boundary**.
- 3 Select Boundaries 30, 32, 34, 36, 41–43, 45, and 47 only.
- 4 In the **Operator name** text field, type intop.
- 5 In the **Label** text field, type Integration over punch boundary.

Create explicit selections for the contact boundaries and various domains.

#### *Explicit 1*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 30, 32, 34, 36, 41–43, 45, and 47 only.
- 5 In the **Label** text field, type contact\_punch.

#### *Explicit 2*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type contact\_die in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 3, 5, 7, 9, 10, 13, 15, 18, 20, 22, and 23 only.

#### *Explicit 3*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type contact\_holder in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 50 only.

Select the boundary of the blank which is in contact with the punch and the holder.

#### *Explicit 4*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type contact\_blank\_up in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 27 only.

Select the boundary of the blank which is in contact with the die.

#### *Explicit 5*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type contact\_blank\_down in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 26 only.

#### *Explicit 6*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 Select Domain 1 only.
- 3 In the **Settings** window for **Explicit**, type die in the **Label** text field.

#### *Explicit 7*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 Select Domain 3 only.
- 3 In the **Settings** window for **Explicit**, type punch in the **Label** text field.

#### *Explicit 8*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type holder in the **Label** text field.
- 3 Select Domain 4 only.

#### *Explicit 9*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 Select Domain 2 only.
- 3 In the **Settings** window for **Explicit**, type blank in the **Label** text field.  
Create the variable to evaluate the blank thickness after the punch release.

#### *General Projection 1 (genproj1)*

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **General Projection**.
- 2 Select Domain 2 only.

- 3 In the **Settings** window for **General Projection**, locate the **Source Map** section.
- 4 In the **r-expression** text field, type R.
- 5 In the **z-expression** text field, type Z.
- 6 Locate the **Destination Map** section. In the **r-expression** text field, type R.

*Variables 1*

- 1 Right-click **General Projection 1 (genproj1)** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
th	genproj1(1)	m	Thickness

- 4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
- 5 Select Domain 2 only.

*Union 1*

- 1 On the **Definitions** toolbar, click **Union**.
- 2 In the **Settings** window for **Union**, type die\_and\_holder in the **Label** text field.
- 3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
- 4 In the **Add** dialog box, in the **Selections to add** list, choose **die** and **holder**.
- 5 Click **OK**.

Modify the contact pairs by activating **Manual control of selection** option. Make sure that the blank is destination in all contact pairs.

- 6 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Contact Pair 1 (ap1)**.
- 7 In the **Settings** window for **Pair**, locate the **Pair Type** section.
- 8 Select the **Manual control of selections** check box.
- 9 Locate the **Source Boundaries** section. From the **Selection** list, choose **contact\_die**.
- 10 Locate the **Destination Boundaries** section. From the **Selection** list, choose **contact\_blank\_down**.
- 11 In the **Label** text field, type die\_blank.
- 12 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Contact Pair 2 (ap2)**.

- 13 In the **Settings** window for **Pair**, locate the **Pair Type** section.
- 14 Select the **Manual control of selections** check box.
- 15 Locate the **Source Boundaries** section. From the **Selection** list, choose **contact\_punch**.
- 16 Locate the **Destination Boundaries** section. From the **Selection** list, choose **contact\_blank\_up**.
- 17 In the **Label** text field, type `punch_blank`.
- 18 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Contact Pair 3 (ap3)**.
- 19 In the **Settings** window for **Pair**, locate the **Pair Type** section.
- 20 Select the **Manual control of selections** check box.
- 21 Locate the **Source Boundaries** section. From the **Selection** list, choose **contact\_holder**.
- 22 Locate the **Destination Boundaries** section. From the **Selection** list, choose **contact\_blank\_up**.
- 23 In the **Label** text field, type `holder_blank`.

## MATERIALS

On the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

### ADD MATERIAL

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-In>Structural steel**.
- 3 Click **Add to Component** in the window toolbar.

### ADD MATERIAL

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-In>Aluminum**.
- 3 Click **Add to Component** in the window toolbar.

## MATERIALS

*Aluminum (mat2)*

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 From the **Selection** list, choose **blank**.

## **SOLID MECHANICS (SOLID)**

### *Linear Elastic Material 1*

Set the blank domain with an elastoplastic material model.

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

### *Plasticity 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Plasticity**.
- 2 In the **Settings** window for **Plasticity**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **blank**.
- 4 Locate the **Plasticity Model** section. From the **Plasticity model** list, choose **Large plastic strains**.
- 5 From the **Yield function F** list, choose **Hill orthotropic plasticity**.
- 6 From the **Specify** list, choose **Hill's coefficients**.
- 7 Find the **Isotropic hardening model** subsection. From the list, choose **User defined**.

### *Contact 1*

- 1 On the **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Contact**.
- 2 In the **Settings** window for **Contact**, locate the **Contact Pressure Method** section.
- 3 From the list, choose **Penalty**.
- 4 Locate the **Pair Selection** section. In the **Pairs** list, choose **die\_blank (ap1)**, **punch\_blank (ap2)**, and **holder\_blank (ap3)**.

Fix the die and holder.

### *Fixed Constraint 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **Fixed Constraint**.
- 2 In the **Settings** window for **Fixed Constraint**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **die\_and\_holder**.

Set prescribed displacements for the punch.

### *Prescribed Displacement 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Displacement**.
- 2 In the **Settings** window for **Prescribed Displacement**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **punch**.

- 4 Locate the **Prescribed Displacement** section. Select the **Prescribed in r direction** check box.
- 5 Select the **Prescribed in z direction** check box.
- 6 In the  $u_{0z}$  text field, type U\_punch (p).

## MATERIALS

*Aluminum (mat2)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat2)**.
- 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
Hill's coefficients	{Hillcoefficients1, Hillcoefficients2, Hillcoefficients3, Hillcoefficients4, Hillcoefficients5, Hillcoefficients6}	{F, G, H, L, M, N}	$m^2 \cdot s^4 / kg^2$	Elastoplastic material model
Hardening function	sigmagh	sigma_h(solid.epe)	Pa	Elastoplastic material model
Density	rho	2700[kg/m <sup>3</sup> ]	kg/m <sup>3</sup>	Basic
Young's modulus	E	70e9[Pa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.33	l	Young's modulus and Poisson's ratio
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	l	Basic
Heat capacity at constant pressure	Cp	900[J/(kg*K)]	J/(kg*K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	238[W/(m*K)]	W/(m*K)	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	3.774e7[S/m]	S/m	Basic

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilon <sub>r_iso</sub> ; epsilon <sub>r_ii</sub> = epsilon <sub>r_iso</sub> , epsilon <sub>r_ij</sub> = 0	1		Basic
Coefficient of thermal expansion	alpha <sub>iso</sub> ; alpha <sub>ii</sub> = alpha <sub>iso</sub> , alpha <sub>ij</sub> = 0	23e-6[1 / K]	1/K	Basic
Murnaghan third-order elastic moduli	l	- 2.5e11 [Pa ]	N/m <sup>2</sup>	Murnaghan
Murnaghan third-order elastic moduli	m	- 3.3e11 [Pa ]	N/m <sup>2</sup>	Murnaghan
Murnaghan third-order elastic moduli	n	- 3.5e11 [Pa ]	N/m <sup>2</sup>	Murnaghan
Lamé parameter $\lambda$	lamLame	5.1e10 [Pa ]	N/m <sup>2</sup>	Lamé parameters
Lamé parameter $\mu$	muLame	2.6e10 [Pa ]	N/m <sup>2</sup>	Lamé parameters

## MESH I

Mesh the curved boundaries more densely compared to the flat boundaries. Mesh blank finely as it is **destination** in all contact pairs.

### *Mapped I*

- 1 In the **Model Builder** window, under **Component I (comp1)** right-click **Mesh I** and choose **Mapped**.
- 2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **blank**.

### *Distribution I*

- 1 Right-click **Component I (comp1)>Mesh I>Mapped I** and choose **Distribution**.
- 2 Click the **Zoom Box** button on the **Graphics** toolbar.
- 3 Select Boundary 25 only.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.
- 5 In the **Settings** window for **Distribution**, locate the **Distribution** section.

6 In the **Number of elements** text field, type 2.

#### *Distribution 2*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundary 27 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 400.

#### *Edge 1*

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Edge**.  
Zoom in and out to select the following boundaries.
- 2 Select Boundaries 18, 20, 22, 23, 43, 45, and 47 only.

#### *Size 1*

- 1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Extremely fine**.
- 4 Click the **Custom** button.
- 5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 6 In the associated text field, type 1.
- 7 Select the **Minimum element size** check box.
- 8 In the associated text field, type 0.001.

#### *Mapped 2*

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.
- 2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 4 only.

#### *Distribution 1*

- 1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 2** and choose **Distribution**.
- 2 Select Boundary 49 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 1.

#### *Distribution 2*

- 1 Right-click **Mapped 2** and choose **Distribution**.

- 2 Select Boundary 51 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 10.

*Size 1*

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.
- 2 Right-click **Free Triangular 1** and choose **Size**.
- 3 In the **Settings** window for **Size**, locate the **Element Size** section.
- 4 From the **Predefined** list, choose **Finer**.
- 5 Click **Build All**.

**STUDY 1**

Click on **Show Default Solver** in order to customize the solver settings. **Constant Newton** method is selected as nonlinear method from the **Fully Coupled** node. **Constant** predictor is chosen from the **Parametric** node.

*Step 1: Stationary*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.
- 3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 4 Click **Add**.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p (Parameter)	range(0, 2e-3, 0.9) range(0.901, 1e-3, 1) range(1.0002, 1e-4, 2)	

*Solution 1 (sol1)*

- 1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.
- 2 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.
- 3 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Constant (Newton)**.
- 4 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.
- 5 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Parametric**.

6 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

7 Select the **Tuning of step size** check box.

8 In the **Minimum step size** text field, type 0.00001.

9 In the **Maximum step size** text field, type 0.005.

10 From the **Predictor** list, choose **Constant**.

Add a stop condition to prevent computation after full release. The stop condition is controlled by the gap distance between punch and blank.

11 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Parametric** and choose **Stop Condition**.

12 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

13 Click **Add**.

14 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

15 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
comp1.intop(comp1.incontact_ap2)<1E-6	true	√	Stop expression 1

Click on **Get Initial Value** in order to generate the default plots, so that they can be modified, and can be used for visualization while solving.

*Stress (solid)*

On the **Study** toolbar, click **Get Initial Value**.

## RESULTS

*Stress (solid)*

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

2 From the **Frame** list, choose **Spatial (r, phi, z)**.

*Plastic strain*

1 In the **Model Builder** window, expand the **Stress (solid)** node.

2 Right-click **Plastic strain** and choose **Disable**.

*Contact pressure*

In the **Model Builder** window, under **Results>Stress (solid)** right-click **Contact pressure** and choose **Disable**.

## STUDY 1

### *Step 1: Stationary*

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, click to expand the **Results while solving** section.
- 3 Locate the **Results While Solving** section. Select the **Plot** check box.
- 4 From the **Update at** list, choose **Steps taken by solver**.  
Now compute the solution.
- 5 On the **Study** toolbar, click **Compute**.

The default plot shows the von Mises stress after release as in [Figure 3](#):

## RESULTS

### *Stress (solid)*

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **2D Plot Group**, type Residual stress in the **Label** text field.
- 3 On the **Residual stress** toolbar, click **Plot**.

The following steps create the plot in [Figure 1](#):

### *Revolution 2D 1*

- 1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Revolution 2D 1**.
- 2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution layers** section.
- 3 Locate the **Revolution Layers** section. In the **Start angle** text field, type 0.
- 4 In the **Revolution angle** text field, type 360.

### *Stress, 3D (solid)*

- 1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.
- 2 In the **Settings** window for **3D Plot Group**, type Deformed shape, 3D in the **Label** text field.
- 3 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

### *Plastic strain*

- 1 In the **Model Builder** window, expand the **Results>Deformed shape, 3D** node.
- 2 Right-click **Plastic strain** and choose **Disable**.

### *Contact pressure*

In the **Model Builder** window, under **Results>Deformed shape, 3D** right-click **Contact pressure** and choose **Disable**.

### *Surface 1*

- 1 In the **Model Builder** window, under **Results>Deformed shape, 3D** click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `solid.epeGp`.
- 4 Select the **Description** check box.
- 5 In the associated text field, type `Deformed shape, 3D`.
- 6 In the **Unit** field, type `.`
- 7 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 8 From the **Color** list, choose **Gray**.
- 9 On the **Deformed shape, 3D** toolbar, click **Plot**.

The following steps create the plot in [Figure 4](#):

### *2D Plot Group 3*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type `Effective plastic strain` in the **Label** text field.
- 3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (r, phi, z)**.

### *Surface 1*

- 1 Right-click **Effective plastic strain** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `solid.epe`.

### *Deformation 1*

- 1 Right-click **Results>Effective plastic strain>Surface 1** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 Select the **Scale factor** check box.
- 4 In the associated text field, type `1`.
- 5 On the **Effective plastic strain** toolbar, click **Plot**.

The following steps create the plot in [Figure 5](#):

#### *2D Plot Group 4*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type **Blank thickness** in the **Label** text field.
- 3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (r, phi, z)**.

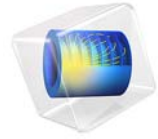
#### *Surface 1*

- 1 Right-click **Blank thickness** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type **th**.

#### *Deformation 1*

- 1 Right-click **Results>Blank thickness>Surface 1** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 Select the **Scale factor** check box.
- 4 In the associated text field, type **1**.
- 5 On the **Blank thickness** toolbar, click **Plot**.





# Combining Elastoplastic and Creep Material Models

## Introduction

---

this example shows how to combine different types of material nonlinearity, such as creep and elastoplasticity. In this specific example you perform a stress and nonlinear strain analysis on a thick cylinder under a nonproportional loading: an initial temperature increase followed by a fluctuating pressure applied to the internal surface of the cylinder.

This load case involves two different nonlinear material behaviors: elastoplasticity and creep. The plastic behavior is introduced during the temperature increase and the rapid change of the pressure sign. The creep behavior develops under the constant pressure load applied to the pipe for a sufficiently long period of time.

The original model is a NAFEMS benchmark model described in *Selected Benchmarks for Material Non-Linearity* (Ref. 1). The COMSOL Multiphysics solutions are compared with the reference data.

## Model Definition

---

The physical geometry of the problem consists of a hollow cylinder with a 0.16 m inner radius and a 0.25 m outer radius. For simplicity, the study is performed on a 2D plane using an axial symmetry assumption, reducing the model geometry to a rectangle.

The benchmark setup takes the cylinder to be infinitely long so that it can be modeled using a plane-strain assumption in the pipe axis plane. This assumption can easily be implemented by constraining the axial displacement in the whole geometry.

### MATERIAL PROPERTIES

- Isotropic material with  $E = 2.2 \cdot 10^7$  Pa,  $\nu = 0.3$ ,  $\alpha = 1.85 \cdot 10^{-5}$  K<sup>-1</sup>
- Elastoplastic with initial yield stress,  $\sigma_{y0} = 9900$  Pa,
- Nonlinear isotropic hardening with stress-strain curve according to the table below:

Plastic strain	Stress (Pa)
3.9e-4	12500
9.5e-4	15200
2.95e-3	17500
6.15e-3	20000

This nonlinear hardening function  $\sigma_h(\epsilon_{pe})$  is defined in the Materials node by an interpolation function.

- Creep constitutive law:

$$\dot{\epsilon}^c = 1e^{-26} \sigma^{5.25} t^{-0.1} \quad (1)$$

where  $\dot{\epsilon}^c$  is the equivalent creep strain increment,  $\sigma$  is the equivalent stress, and  $t$  is the time. In Equation 1, the stress is given in Pa and the time in seconds. To follow the benchmark example, the creep behavior starts after 2 s.

### LOAD HISTORY

- The internal pressure step up from 0 to 3600 Pa in 1 s at  $t = 1$  s and change sign within 1 s at  $t = 999$  s as illustrated in Figure 1 below.

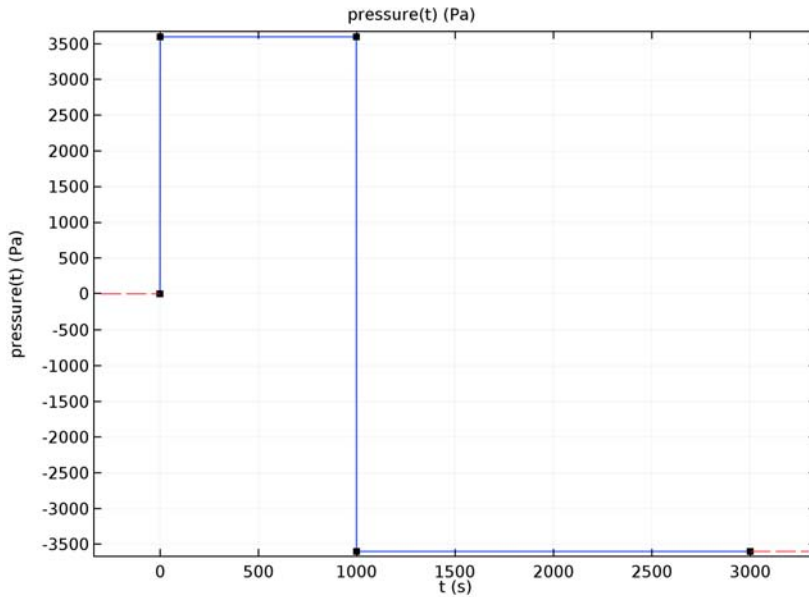


Figure 1: Internal pressure versus time (blue line). The red line is the default extrapolation of the pressure outside the defined domain.

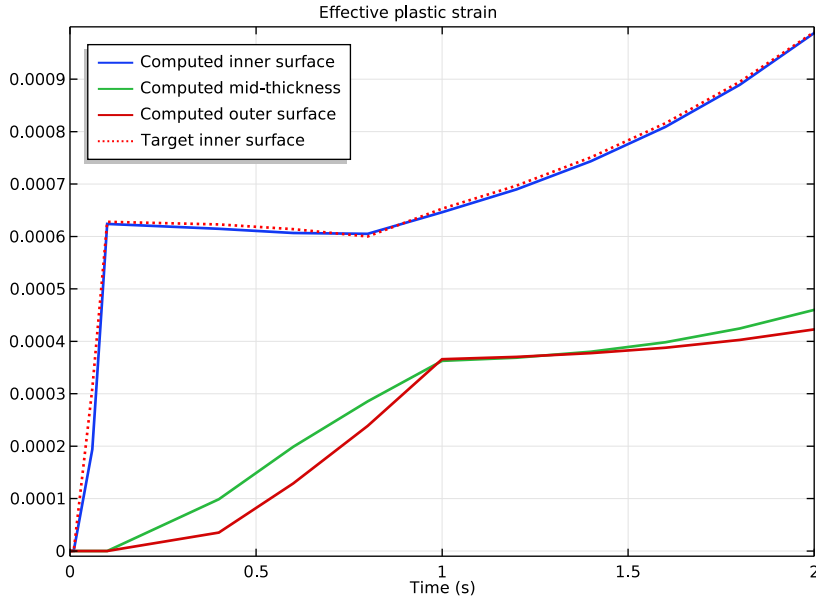
- The temperature transient is defined by:

$$T = \begin{cases} 0 & t = 0 \\ 500t \left( \frac{r-0.07}{0.09} \right) & 0 \leq t \leq 0.1 \\ 50t \left( \frac{r-0.16}{0.09} \right) & 0.1 \leq t \leq 1 \\ 50 & t > 1 \end{cases}$$

where both time and radius,  $r$ , are given in default units.

## Results and Discussion

In [Figure 2](#) you can see the increase of the effective plastic strain at an early time ( $t < 2$  s).

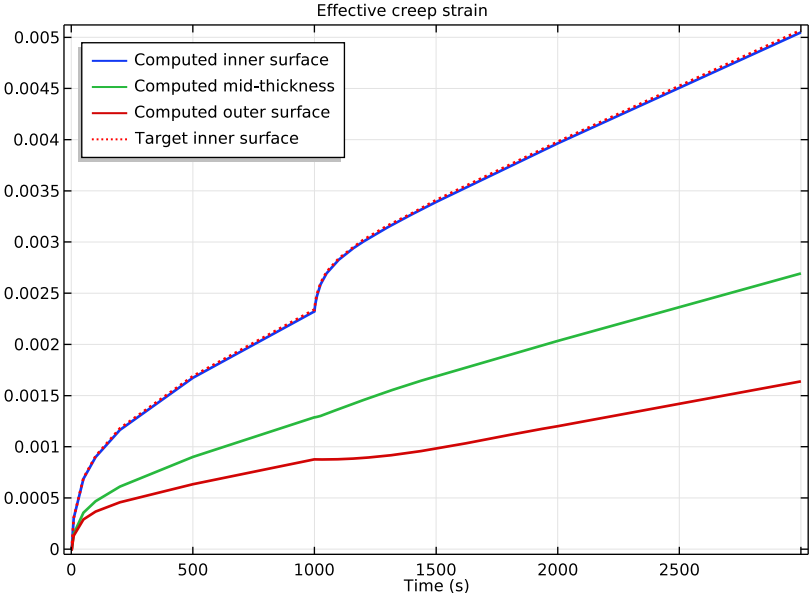


*Figure 2: Time history of effective plastic strain when the cylinder is affected by the temperature gradient.*

The dotted curve corresponds to the target data for the benchmark purpose. You can see that the computed solution is in fairly good agreement with the targeted solution.

The initial temperature gradient creates thermal stresses in the cylinder. The stress varies through the cross section and is high enough to develop plastic strain at the inner surface.

In the considered time interval, stress is significantly higher at the inner surface than in the center or at the outer radius. After 0.1 s, the thermal load at the inner radius is removed but it is still present in the rest of the cylinder, where the material starts to deform plastically. After 1 s, the temperature is constant in the material and in combination with the mechanical load the plasticity develops but at a lower rate. After 2 s the material relaxes and creep takes over; see [Figure 3](#).



*Figure 3: Time history of effective creep strain.*

The change in pressure at  $t = 999$  s has the most significant effect on the creep strain at the inner wall.

[Figure 4](#), [Figure 5](#), and [Figure 6](#) show, respectively, the axial, hoop, and von Mises stress at the inner radius.

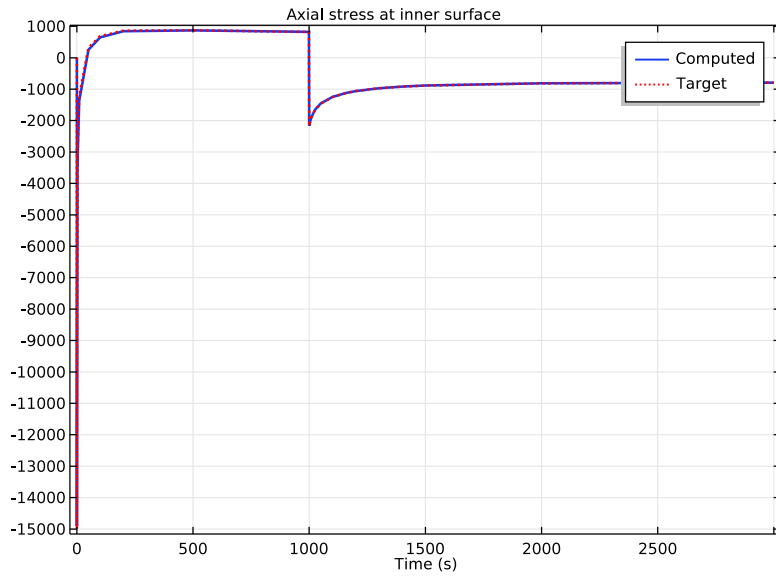


Figure 4: Time history of axial stress at the inner radius.

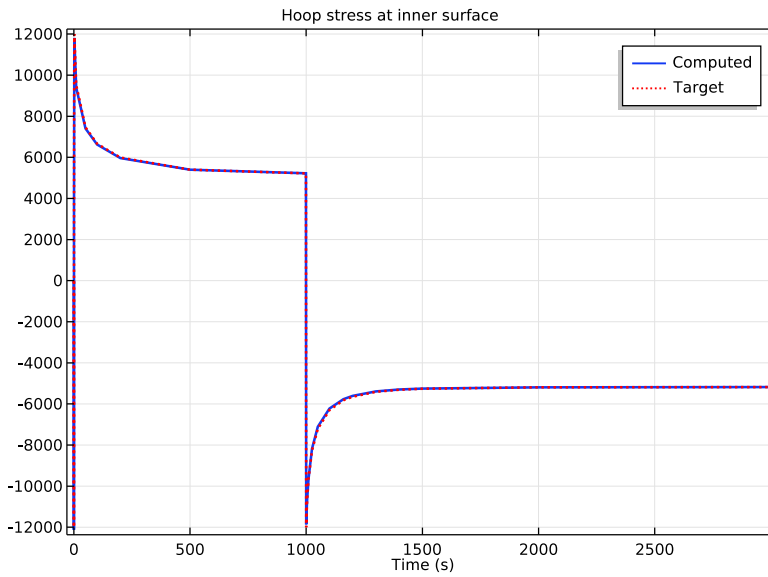
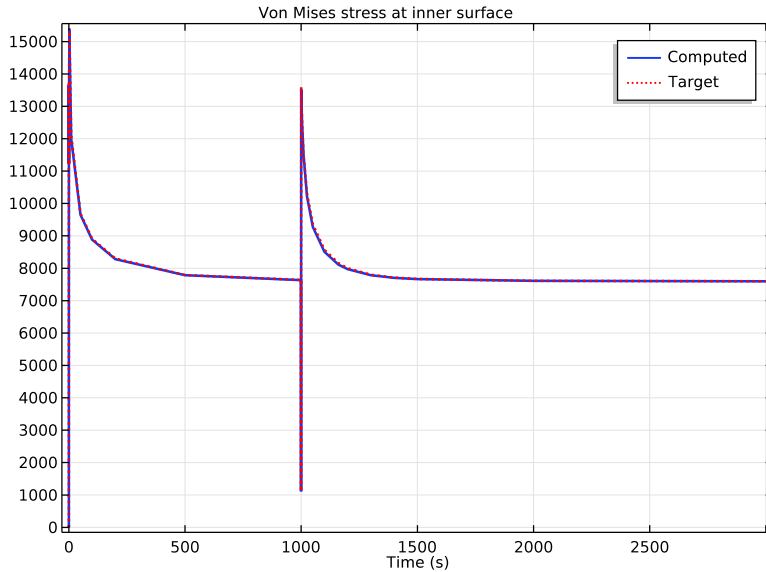


Figure 5: Time history of hoop stress at the inner radius.



*Figure 6: Time history of von Mises stress at the inner radius.*

The axial and hoop stresses both change direction when the inner pressure is reversed. The asymptotic behavior is caused by the creep of the material. The effective stress, on the other hand, cannot be negative. When the pressure is changed within 1 s, the von Mises stress first decreases, pressure decreases to zero, and then increases, the pressure decreases further to a nonzero negative pressure. The peak is caused by the creep strains, which needs instantaneous reverse due to the pressure change.

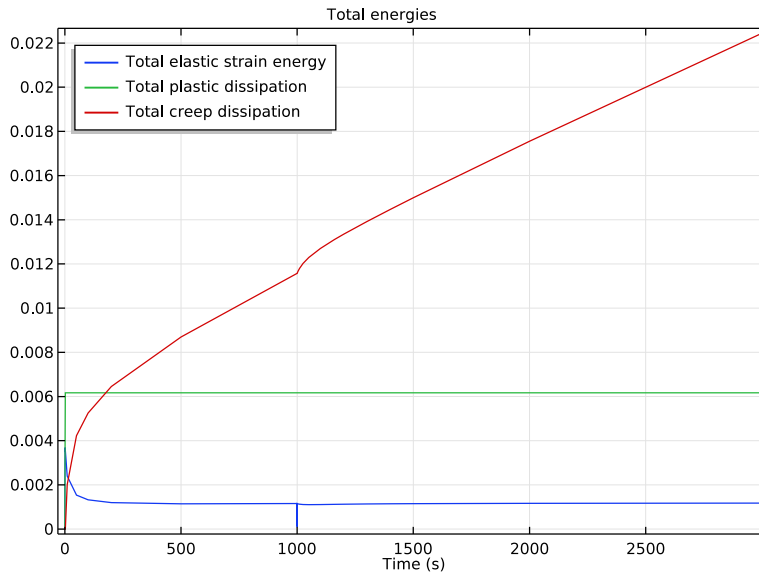


Figure 7: Elastic strain energy, plastic dissipation and creep dissipation.

### Notes About the COMSOL Implementation

In COMSOL Multiphysics you can combine several nonlinear models for the same material. Different material models are available as feature nodes to add to the Linear Elastic Material node; see Figure 8.

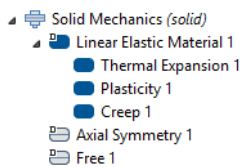


Figure 8: Multiple material models added to a default linear elastic material.

COMSOL Multiphysics adds all strain contributions of each node to the total strain.

The examined creep behavior can be modeled with a built-in Norton-Bailey model with a time-hardening option. To normalize the effective stress in Pa and the time in hours, set the reference stress  $\sigma_r = 1$  Pa and the reference time  $t_r = 1$  s. Time offset,  $t_0$ , is here set to 1 ms to avoid the singularity that occurs at  $t = 0$ . To set the COMSOL model identical to

the benchmark model, the creep strain is added only after  $t=2$  s. To do this multiply the creep rate coefficient,  $A$ , with the logical expression,  $(t > 2)$ .

When computing the creep material model, the time steps taken by the solver must be small when the creep rate is high and large when the creep strain rate is low. Because the time step can be difficult to predict, you can let the solver control what time steps to store.

Note that the benchmark (Ref. 1) utilizes a very coarse mesh. Due to the interpolation from Gauss points, this coarse mesh results in decreasing effective plastic strains, as shown around  $0.1 < t < 1$  s in Figure 2.

### *Reference*

---

1. D. Linkens, *Selected Benchmarks For Material Non-Linearity, volume 2*, NAFEMS, 1993.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/Creep/elastoplastic\_creep

---

### *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 Axisymmetric**.
- 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 **Preset Studies>Time Dependent**.
- 6** Click **Done**.

## GLOBAL DEFINITIONS

### Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
m	0.9	0.9	Power law exponent

## DEFINITIONS

### Interpolation 1 (int1)

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type pressure.
- 4 In the table, enter the following settings:

t	f(t)
0	0
1	0
2	3600
999	3600
1000	-3600
3000	-3600

- 5 Locate the **Units** section. In the **Arguments** text field, type s.
- 6 In the **Function** text field, type Pa.
- 7 Click **Plot**.

### Analytic 1 (an1)

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Analytic**.
- 2 In the **Settings** window for **Analytic**, type T in the **Function name** text field.
- 3 Locate the **Definition** section. In the **Expression** text field, type  $500 * t^{((r-7e-2)/9e-2)} * (t \leq 0.1) + 50 * t^{((r-0.16)/9e-2)} * (t > 0.1) * (t \leq 1) + 50 * (t > 1)$ .
- 4 In the **Arguments** text field, type t, r.
- 5 Locate the **Units** section. In the **Arguments** text field, type s, m.

6 In the **Function** text field, type K.

#### *Interpolation 2 (int2)*

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 From the **Data source** list, choose **File**.
- 4 Click **Browse**.
- 5 Browse to the model's Application Libraries folder and double-click the file `elastoplastic_creep_target.txt`.
- 6 In the **Number of arguments** text field, type 1.
- 7 Click **Import**.
- 8 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
sz_target	1
sphi_target	2
mises_target	3
epe_target	4
ece_target	5

### **GEOMETRY I**

#### *Rectangle 1 (r1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type  $9e-2$ .
- 4 In the **Height** text field, type  $9e-3$ .
- 5 Locate the **Position** section. In the **r** text field, type 0.16.
- 6 Click **Build All Objects**.

### **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 **Structural Transient Behavior** section.
- 3 From the list, choose **Quasi-static**.

### *Linear Elastic Material I*

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

### *Thermal Expansion I*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.
- 2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.
- 3 In the  $T$  text field, type  $T(t, r)$ .
- 4 Locate the **Thermal Expansion Properties** section. In the  $T_{\text{ref}}$  text field, type 0.

### *Linear Elastic Material I*

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

### *Plasticity I*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Plasticity**.
- 2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.
- 3 Find the **Isotropic hardening model** subsection. From the list, choose **User defined**.

### *Linear Elastic Material I*

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

### *Creep I*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Creep**.
- 2 In the **Settings** window for **Creep**, locate the **Creep Data** section.
- 3 From the **Material model** list, choose **Norton-Bailey**.

In order to fulfill the benchmark requirements the creep material model is activated only after  $t = 2$  s.

- 4 From the  $A$  list, choose **User defined**. In the associated text field, type  $1e-26/m*(t>2)$ .
- 5 From the  $\sigma_{\text{ref}}$  list, choose **User defined**. In the associated text field, type 1.
- 6 From the  $n$  list, choose **User defined**. In the associated text field, type 5.25.
- 7 In the  $m$  text field, type  $m$ .
- 8 In the  $t_{\text{shift}}$  text field, type  $1e-3$ .
- 9 In the  $t_{\text{ref}}$  text field, type 1.

### *Linear Elastic Material I*

Activate the computation of the dissipated energy.

- 1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.
- 2 In the **Model Builder** window, under **Component 1 (comp 1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.
- 3 In the **Settings** window for **Linear Elastic Material**, click to expand the **Energy dissipation** section.
- 4 Locate the **Energy Dissipation** section. Select the **Calculate dissipated energy** check box.

#### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 Specify the  $\mathbf{F}_A$  vector as

pressure(t)	r
0	z

#### *Prescribed Displacement 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Displacement**.
- 2 Click in the **Graphics** window and then press Ctrl+A to select all domains.
- 3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
- 4 Select the **Prescribed in z direction** check box.
- 5 Right-click **Prescribed Displacement 1** and choose **Blank Material**.

## **MATERIALS**

#### *Material 1 (mat1)*

- 1 In the **Settings** window for **Material**, locate the **Material Contents** section.
- 2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	22 [MPa]	Pa	Basic
Poisson's ratio	nu	0.3		Basic
Density	rho	0	kg/m <sup>3</sup>	Basic
Coefficient of thermal expansion	alpha	18.5e-6	1/K	Basic

Property	Variable	Value	Unit	Property group
Initial yield stress	sigmags	sigma_yield (0)	Pa	Elastoplastic material model
Hardening function	sigmagh	sigma_yield (epe)- sigma_yield (0)	Pa	Elastoplastic material model

- 3 In the **Model Builder** window, expand the **Material I (mat1)** node, then click **Elastoplastic material model (ElastoplasticModel)**.
- 4 In the **Settings** window for **Property Group**, locate the **Output Properties and Model Inputs** section.
- 5 Find the **Quantities** subsection. In the tree, select **Model Inputs>Effective Plastic Strain**.
- 6 Click **Add**.

#### *Interpolation 1 (int1)*

- 1 Right-click **Component I (comp1)>Materials>Material I (mat1)>Elastoplastic material model (ElastoplasticModel)** and choose **Functions>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type `sigma_yield`.
- 4 In the table, enter the following settings:

t	f(t)
0	9900
3.9e-4	12500
9.5e-4	15200
2.95e-3	17500
6.15e-3	20000

- 5 Locate the **Units** section. In the **Arguments** text field, type 1.
- 6 In the **Function** text field, type Pa.

## **MESH 1**

#### *Distribution 1*

- 1 In the **Model Builder** window, under **Component I (comp1)** right-click **Mesh 1** and choose **Mapped**.
- 2 Right-click **Mapped 1** and choose **Distribution**.
- 3 Select Boundary 1 only.

- 4 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 5 In the **Number of elements** text field, type 1.

#### *Distribution 2*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 10.
- 5 Click **Build All**.

### **STUDY 1**

#### *Step 1: Time Dependent*

- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 In the **Times** text field, type 0 0.01 0.06 0.1 range(0.4, 0.2, 2) 10 50 100 200 500 range(999, 0.2, 1000) 1002 1004 1010 1025 1050 1100 1160 range(1200, 100, 1500) 2000 3000.
- 3 From the **Tolerance** list, choose **User controlled**.
- 4 In the **Relative tolerance** text field, type 1e-3.

#### *Solution 1 (sol1)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
- 3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

Ensure that results are given at exactly the same times as in the benchmark reference.

- 4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.
- 5 Select the **Initial step** check box.
- 6 In the associated text field, type 0.01.  
Setting the initial step ensures a good calculation of creep strain at  $t = 0$ .
- 7 On the **Study** toolbar, click **Compute**.

## RESULTS

### *Stress (solid)*

Follow the steps below to obtain [Figure 2](#).

### *Cut Point 2D 1*

- 1 On the **Results** toolbar, click **Cut Point 2D**.
- 2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.
- 3 In the **R** text field, type 0.16 0.205 0.25.
- 4 In the **Z** text field, type 5e-3.

### *ID Plot Group 3*

- 1 On the **Results** toolbar, click **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Effective Plastic Strain in the **Label** text field.
- 3 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.
- 4 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 6 In the **Title** text area, type Effective plastic strain.

### *Point Graph 1*

- 1 Right-click **Effective Plastic Strain** and choose **Point Graph**.
- 2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Effective plastic strain**.
- 3 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type 2.
- 4 Click to expand the **Legends** section. Select the **Show legends** check box.
- 5 From the **Legends** list, choose **Manual**.
- 6 In the table, enter the following settings:

---

#### **Legends**

---

Computed inner surface

---

Computed mid-thickness

---

Computed outer surface

---

### Global 1

- 1 In the **Model Builder** window, under **Results** right-click **Effective Plastic Strain** and choose **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
epe_target(t)		

- 4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
- 5 From the **Color** list, choose **Red**.
- 6 In the **Width** text field, type 2.
- 7 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
- 8 In the table, enter the following settings:

Legends
Target inner surface

- 9 On the **Effective Plastic Strain** toolbar, click **Plot**.  
Display development of plasticity in first 2 s as in the benchmark example.

### Effective Plastic Strain

- 1 In the **Model Builder** window, under **Results** click **Effective Plastic Strain**.
- 2 In the **Settings** window for **ID Plot Group**, click to expand the **Axis** section.
- 3 Select the **Manual axis limits** check box.
- 4 In the **x minimum** text field, type 0.
- 5 In the **x maximum** text field, type 2.
- 6 On the **Effective Plastic Strain** toolbar, click **Plot**.

You can obtain [Figure 3](#) by running the following instructions.

### Effective Plastic Strain 1

- 1 Right-click **Results>Effective Plastic Strain** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Effective Creep Strain in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type Effective creep strain.
- 4 Locate the **Axis** section. Clear the **Manual axis limits** check box.

### *Point Graph 1*

- 1 In the **Model Builder** window, expand the **Results>Effective Creep Strain** node, then click **Point Graph 1**.
- 2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>solid.eceGp - Effective creep strain**.

### *Global 1*

- 1 In the **Model Builder** window, under **Results>Effective Creep Strain** click **Global 1**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
ece_target(t)		

- 4 On the **Effective Creep Strain** toolbar, click **Plot**.

To get [Figure 4](#), follow the instruction below.

### *Cut Point 2D 2*

- 1 On the **Results** toolbar, click **Cut Point 2D**.
- 2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.
- 3 In the **R** text field, type 0.16.
- 4 In the **Z** text field, type 5e-3.

### *Effective Creep Strain 1*

- 1 In the **Model Builder** window, under **Results** right-click **Effective Creep Strain** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Axial Stress in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type Axial stress at inner surface.
- 4 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 2**.
- 5 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

### *Point Graph 1*

- 1 In the **Model Builder** window, expand the **Results>Axial Stress** node, then click **Point Graph 1**.
- 2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>**

**Stress (Gauss points)>Stress tensor, Gauss-point evaluation (spatial frame)>solid.sGpz - Stress tensor, Gauss-point evaluation, z component.**

3 Locate the **Legends** section. In the table, enter the following settings:

<b>Legends</b>
Computed

*Global 1*

- 1 In the **Model Builder** window, under **Results>Axial Stress** click **Global 1**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

<b>Expression</b>	<b>Unit</b>	<b>Description</b>
sz_target (t)		

4 In the table, enter the following settings:

<b>Legends</b>
Target

5 On the **Axial Stress** toolbar, click **Plot**.

Follow the instructions below to get [Figure 5](#).

*Axial Stress 1*

- 1 In the **Model Builder** window, under **Results** right-click **Axial Stress** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Hoop Stress in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type Hoop stress at inner surface.

*Point Graph 1*

- 1 In the **Model Builder** window, expand the **Results>Hoop Stress** node, then click **Point Graph 1**.
- 2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress (Gauss points)>Stress tensor, Gauss-point evaluation (spatial frame)>solid.sGpphi - Stress tensor, Gauss-point evaluation, phi component**.

*Global 1*

- 1 In the **Model Builder** window, under **Results>Hoop Stress** click **Global 1**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
sphi_target(t)		

4 On the **Hoop Stress** toolbar, click **Plot**.

Follow the instructions below to get [Figure 6](#).

#### *Hoop Stress I*

- 1 In the **Model Builder** window, under **Results** right-click **Hoop Stress** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Von Mises Stress in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type Von Mises stress at inner surface.

#### *Point Graph I*

- 1 In the **Model Builder** window, expand the **Results>Von Mises Stress** node, then click **Point Graph I**.
- 2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress (Gauss points)>solid.misesGp - von Mises stress, Gauss-point evaluation**.

#### *Global I*

- 1 In the **Model Builder** window, under **Results>Von Mises Stress** click **Global I**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
mises_target(t)		

4 On the **Von Mises Stress** toolbar, click **Plot**.

#### *ID Plot Group 8*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Title** section.
- 3 From the **Title type** list, choose **Manual**.
- 4 In the **Label** text field, type Total Energies.
- 5 Locate the **Title** section. In the **Title** text area, type Total energies.

#### *Global I*

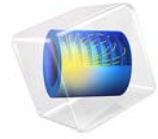
- 1 Right-click **Total Energies** and choose **Global**.

- 2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Global>solid.Ws\_tot - Total elastic strain energy**.
- 3 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Global>solid.Wp\_tot - Total plastic dissipation**.
- 4 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Global>solid.Wc\_tot - Total creep dissipation**.

#### *Total Energies*

- 1 In the **Model Builder** window, under **Results** click **Total Energies**.
- 2 In the **Settings** window for **ID Plot Group**, click to expand the **Legend** section.
- 3 From the **Position** list, choose **Upper left**.
- 4 On the **Total Energies** toolbar, click **Plot**.





# Elastoplastic Analysis of Holed Plate

## Introduction

---

In this example you analyze a perforated plate loaded into the plastic regime. In addition to the original problem, which you can find in section 7.10 of *The Finite Element Method* by O.C. Zienkiewicz (Ref. 1), you can also study the unloading of the plate.

The model also shows how to apply an external hardening function based on an interpolated stress-strain curve.

## Model Definition

---

Figure 1 shows the plate's geometry. Due to the double symmetry of the geometry you only need to analyze a quarter of the plate.

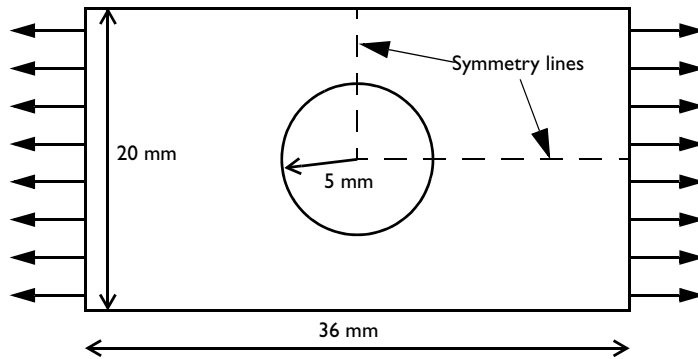


Figure 1: The plate geometry.

Because the plate is thin and the loads are in plane, you can assume a plane stress condition.

### MATERIAL

- Elastic properties:  $E = 70000$  MPa and  $\nu = 0.2$ .
- Plastic properties: Yield stress 243 MPa and a linear isotropic hardening with tangent modulus 2171 MPa.

### CONSTRAINTS AND LOADS

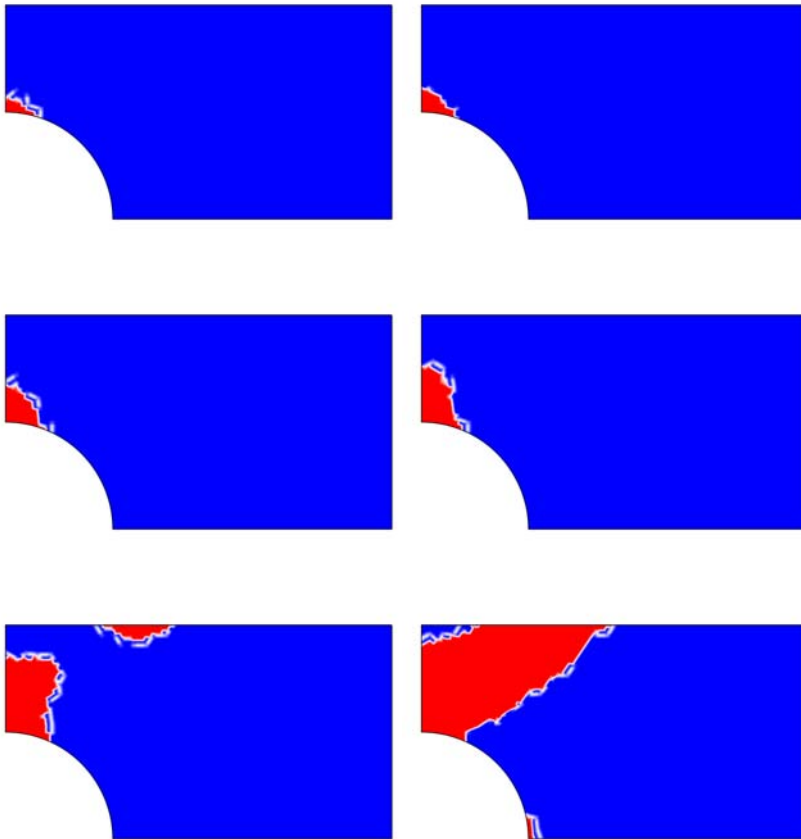
- Symmetry plane constraints are applied on the left most vertical boundary and the lower horizontal boundary.

- The right vertical edge is subjected to a stress, which increases from zero to a maximum value of 133.65 MPa and then is released again. The peak value is selected so that the mean stress over the section through the hole is 10% above the yield stress ( $=1.1 \cdot 243 \cdot (20-10)/20$ ).

### *Results and Discussion*

---

Figure 2 shows the development of the plastic region. The parameter values are 0.59, 0.68, 0.78, 0.88, 0.98, and 1.08. These values are proportional to the load with parameter value 1.0 corresponding to the yield limit as average stress over the cross section through the hole. For a material without strain hardening, the structure would thus have collapsed before reaching the final load level. Because an elastoplastic solution is load-path dependent, it is important not to use too large steps in the load parameter when you anticipate a plastic flow. Usually you can take one large step up to the elastic limit, as this example shows. Moreover, reversed plastic flow can occur during the unloading. This is why this study uses small parameter steps at the end of the parameter range.



*Figure 2: Development of plastic region (red) for parameter values 0.59, 0.68, 0.78, 0.88, 0.98 and 1.08.*

### *Modeling with COMSOL Multiphysics*

---

In this example there are two studies where the only difference is in how the hardening data of the plasticity model is entered. In the first study, you give the data in the most natural way, since a linear hardening can be entered directly using the tangent modulus.

In the second study it is shown how to proceed when you have a tabulated data from a general tensile test. Note that in metal plasticity, the hardening function  $\sigma_h$  to be entered

is the stress added to the initial yield stress  $\sigma_{ys0}$  as function of the effective plastic strain  $\epsilon_{pe}$ . Thus, the function must always pass the point  $(0,0)$ . If your tabulated data contains total stress versus total strain, the hardening function must thus be written as

$$\sigma_h(\epsilon_{pe}) = \sigma_{tab}(\epsilon_{pe} + \sigma_e/E) - \sigma_{ys0}$$

here,  $\sigma_e$  is the effective (von Mises) stress,  $E$  is the Young's modulus, and  $\sigma_{tab}$  is an interpolated function of your tabulated data. Figure 4 shows the linear elastic and plastic regions.

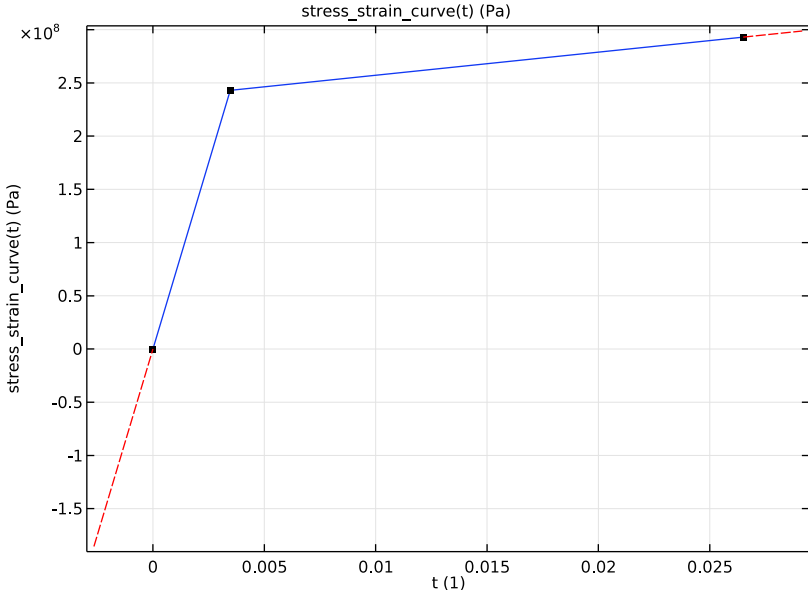


Figure 3: The interpolated stress-strain curve shows both the elastic and hardening regions.

The results show in Figure 4 and Figure 5 are in good agreement. In Study 1, isotropic hardening is generated with an isotropic tangent modulus  $E_{Tiso} = 2.171$  GPa, and in

Study 2 it is generated with interpolated hardening function data, which mimics the isotropic tangent modulus from Study 1.

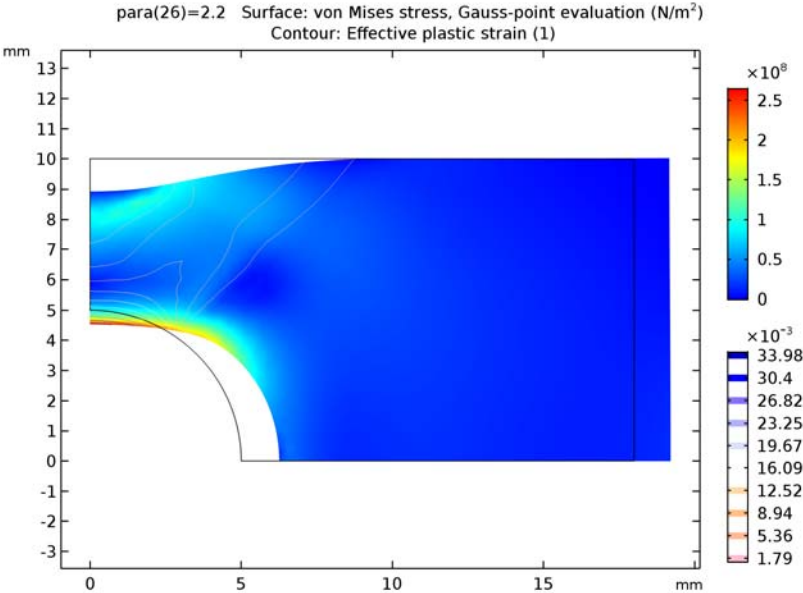


Figure 4: Deformation and von Mises stress for parameter value 2.2. The hardening was implemented with isotropic tangent modulus  $E_{Tiso} = 2.171$  GPa.

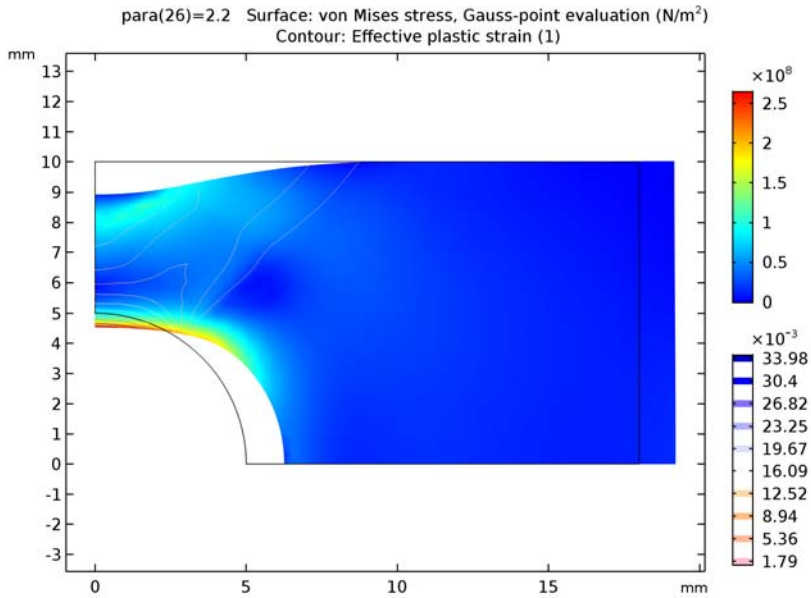


Figure 5: Deformation and von Mises stress for parameter value 2.2. The hardening was implemented with the interpolated hardening function.

### Reference

1. O.C. Zienkiewicz and R.L. Taylor, *The Finite Element Method*, 4th ed., McGraw-Hill, 1991.

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Plasticity/elastoplastic\_plate

### 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 **Preset Studies>Stationary**.
- 6 Click **Done**.

## GEOMETRY 1

Begin by changing the length unit to millimeters.

- 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**.

### *Rectangle 1 (r1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 18.
- 4 In the **Height** text field, type 10.
- 5 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

### *Circle 1 (c1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 5.
- 4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

### *Difference 1 (dif1)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **r1** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 5 Select the object **c1** only.
- 6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

Add a solver parameter for controlling the load expression.

## GLOBAL DEFINITIONS

### Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 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	Horizontal load parameter

## DEFINITIONS

### Interpolation 1 (int1)

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type loadfunc.
- 4 In the table, enter the following settings:

t	f(t)
0	0
1.1	133.65
2.2	0

- 5 Locate the **Units** section. In the **Arguments** text field, type 1.
- 6 In the **Function** text field, type MPa.
- 7 Click **Plot**.

The interpolation function defines the load function.

## 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 **2D Approximation** section.
- 3 From the list, choose **Plane stress**.
- 4 Locate the **Thickness** section. In the  $d$  text field, type 10[mm].

### *Linear Elastic Material 1*

In the **Model Builder** window, expand the **Solid Mechanics (solid)** node, then click **Linear Elastic Material 1**.

### *Plasticity 1*

On the **Physics** toolbar, click **Attributes** and choose **Plasticity**.

## **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:

<b>Property</b>	<b>Variable</b>	<b>Value</b>	<b>Unit</b>	<b>Property group</b>
Young's modulus	E	70e9	Pa	Basic
Poisson's ratio	nu	0.2	l	Basic
Density	rho	7850	kg/m <sup>3</sup>	Basic
Initial yield stress	sigmags	243e6	Pa	Elastoplastic material model
Isotropic tangent modulus	Et	2.171e9	Pa	Elastoplastic material model

## **SOLID MECHANICS (SOLID)**

### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 1 and 3 only.

### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 4 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the  $\mathbf{F}_A$  vector as

loadfunc(para)	x
0	y

### MESH I

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.
- 2 Right-click **Mesh 1** and choose **More Operations>Refine**.
- 3 In the **Settings** window for **Mesh**, click **Build All**.  
The mesh should consist of around 1300 elements.

### STUDY I

*Step 1: Stationary*

Set up an auxiliary continuation sweep for the para parameter.

- 1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.
- 3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 4 Click **Add**.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list
para (Horizontal load parameter)	0 range(0.44,0.05,0.59) range(0.63,0.05,1.08) range(1.1,0.2,1.9) range(1.95,0.05,2.2)

With these settings, the edge load you defined earlier increases from zero to a maximum value of 133.65 MPa and is then released.

- 6 In the **Model Builder** window, click **Study 1**.
- 7 In the **Settings** window for **Study**, type **Isotropic Hardening** in the **Label** text field.
- 8 On the **Home** toolbar, click **Compute**.

### RESULTS

*Stress (solid)*

The default plot shows the von Mises stress with contours of effective plastic strain for the final parameter value.

- 1 In the **Model Builder** window, click **Stress (solid)**.
- 2 In the **Settings** window for **2D Plot Group**, type `Stress`, `Isotropic Hardening` in the **Label** text field.

Visualize the plastic zone using a Boolean expression `solid.epe>0` which is 1 in the plastic region and 0 elsewhere.

#### *2D Plot Group 2*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type `Plastic Region` in the **Label** text field.

#### *Surface 1*

- 1 Right-click **Plastic Region** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `solid.epe>0`.
- 4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

#### *Plastic Region*

- 1 In the **Model Builder** window, under **Results** click **Plastic Region**.
- 2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 3 From the **Parameter value (para)** list, choose **0.59**.
- 4 On the **Plastic Region** toolbar, click **Plot**.

The plot in the Graphics window should now look like that in the upper-left panel of [Figure 2](#).

- 5 Repeat steps 3, 4 and 5 for **Parameter value (para)** 0.68, 0.78, 0.88, 0.98, and 1.08 to reproduce the remaining subplots in [Figure 2](#).

### *Hardening with Experimental Stress-Strain Curve.*

---

To fit the first study case, the stress-strain curve will be very simple, an elastic slope of 70 GPa and then a plastic slope of 2.171 GPa. Only three points are needed to define the stress-strain curve in this simple case.

#### **DEFINITIONS**

##### *Interpolation 2 (int2)*

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `stress_strain_curve`.

4 In the table, enter the following settings:

t	f(t)
0	0
243e6/70e9	243e6
243e6/70e9+50e6/2.171e9	243e6+50e6

5 Locate the **Units** section. In the **Arguments** text field, type 1.

6 In the **Function** text field, type Pa.

7 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Linear**.

8 Click **Plot**.

Add a variable for controlling the hardening.

#### *Variables 1*

1 On the **Home** toolbar, click **Variables** and choose **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
hardening	<code>max(0, stress_strain_curve(solid.epe+solid.mises/solid.E)-solid.sigmags)</code>	Pa	Hardening function

## **SOLID MECHANICS (SOLID)**

### *Linear Elastic Material 1*

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

### *Plasticity 2*

1 On the **Physics** toolbar, click **Attributes** and choose **Plasticity**.

Use hardening function data and add the hardening function to the material properties.

2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.

3 Find the **Isotropic hardening model** subsection. From the list, choose **User defined**.

## MATERIALS

*Material 1 (mat1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 1 (mat1)**.
- 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
Hardening function	sigmagh	hardening	Pa	Elastoplastic material model

## ADD STUDY

- 1 On 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 **Preset Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.
- 5 On 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
para (Horizontal load parameter)	0 range(0.44,0.05,0.59) range(0.63,0.05,1.08) range(1.1,0.2,1.9) range(1.95,0.05,2.2)

The solver settings are the same as in the previous study.

- 5 In the **Model Builder** window, click **Study 2**.
- 6 In the **Settings** window for **Study**, type Interpolated Hardening in the **Label** text field.
- 7 On the **Home** toolbar, click **Compute**.

## RESULTS

### *Stress (solid)*

The default plot of the second study shows the von Mises stress and contour of effective plastic strain for the final parameter value.

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **2D Plot Group**, type Stress, Interpolated Hardening in the **Label** text field.

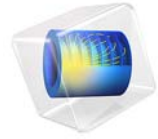
The analysis is now finished. If you want to store this model and re-use it later, you will need to disable the second plasticity feature for the first study.

## ISOTROPIC HARDENING

### *Step 1: Stationary*

- 1 In the **Model Builder** window, under **Isotropic Hardening** 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)>Linear Elastic Material 1>Plasticity 2**.
- 5 Click **Disable**.





# 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.

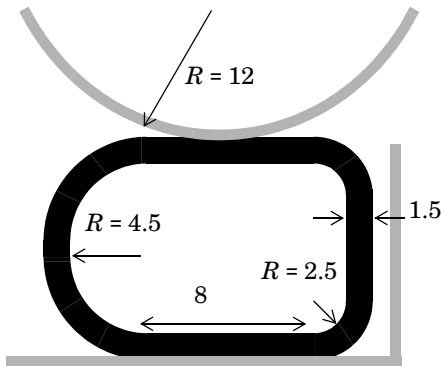


Figure 1: Model geometry.

The model describes a cross section of the seal assuming plane strain. The thickness in the out-of-plane direction is 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 Gauss' theorem, and converting the original surface integral to a contour integral:

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

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  $A_0 = 123.63 \text{ mm}^2$ . The constant  $\gamma$  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**

- 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 continuation solver as the negative  $y$  displacement. It starts with a gap of 0 mm and is lowered 4 mm.

## 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.

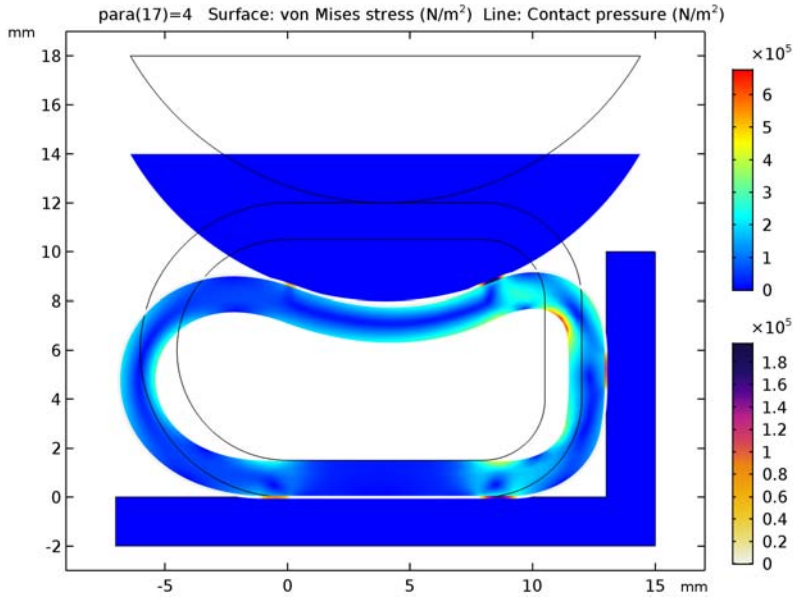
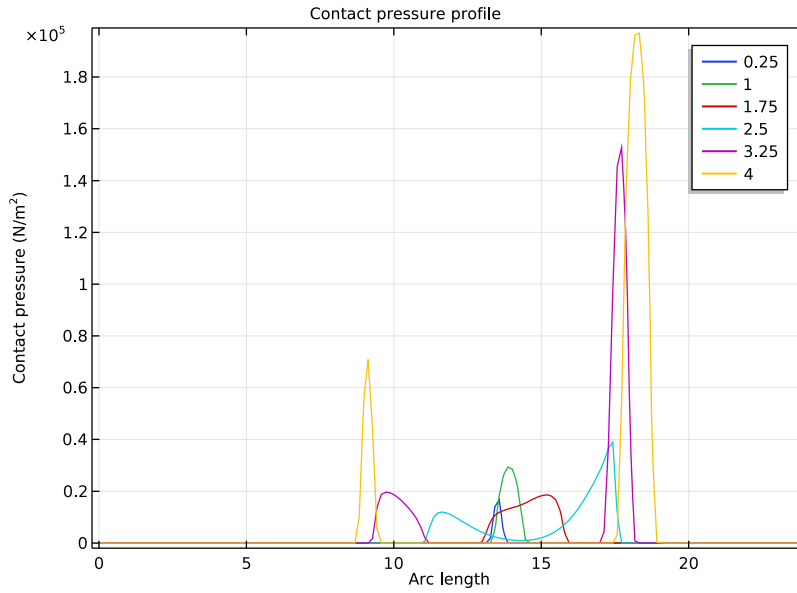


Figure 2: Seal deformation at 4 mm indentation without internal pressure.

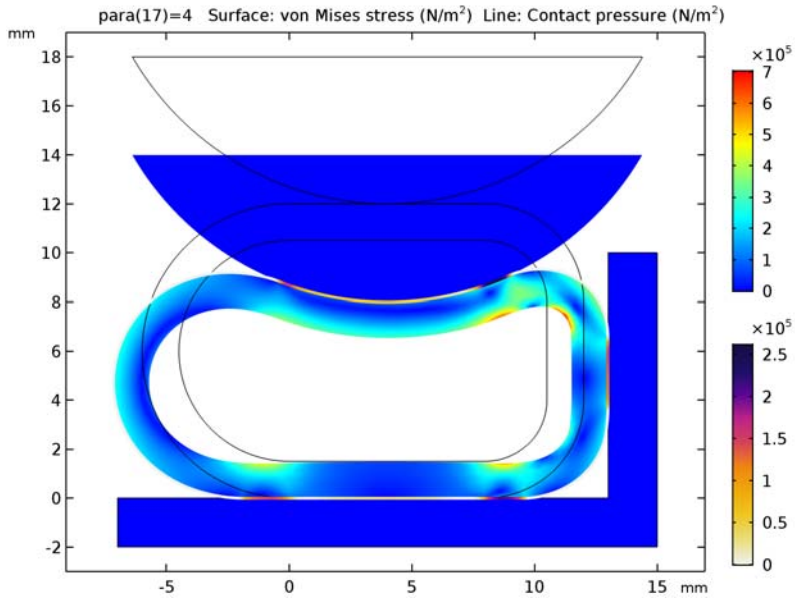
Figure 3 shows the corresponding contact pressure plot. The detachment region appears first at indentation just over 2.5 mm and grows as the indentation increases further. The actual contact areas are reduced to two spots at the sides.

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.



*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.

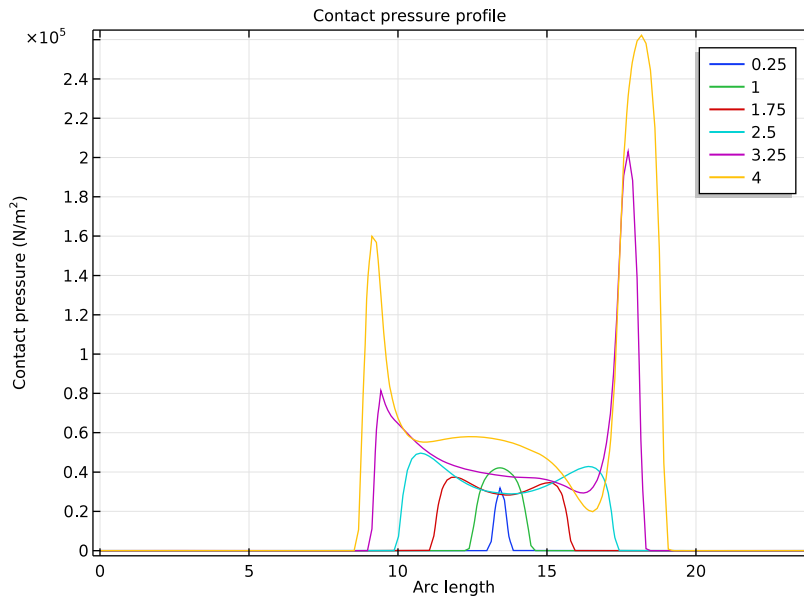


Figure 5: The contact pressure distribution for different indentations with the internal pressure taken into account.

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.

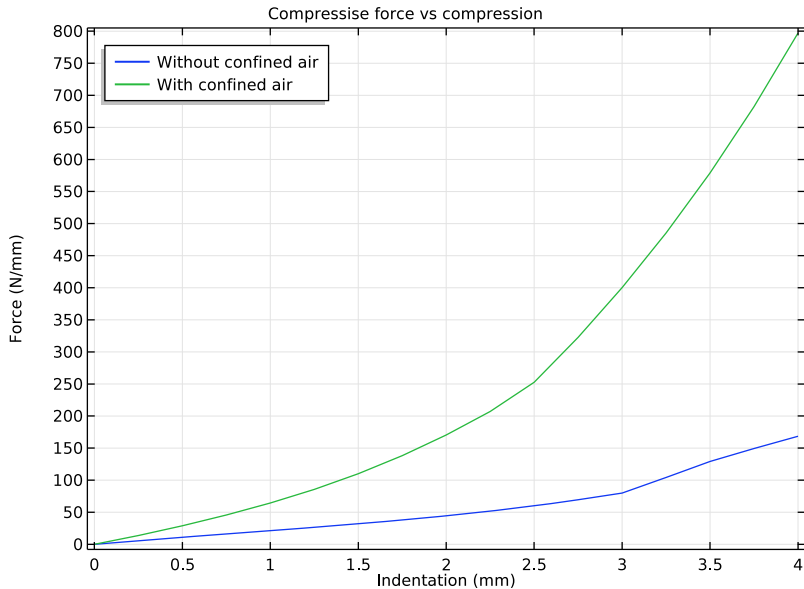


Figure 6: Force per unit length versus compression with and without internal pressure.

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.

---

**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 **Preset Studies>Stationary**.
- 6 Click **Done**.

## GEOMETRY 1

- 1 In the **Model Builder** window, under **Component 1 (comp 1)** click **Geometry 1**.
- 2 In the **Settings** window for **Geometry**, locate the **Units** section.
- 3 From the **Length unit** list, choose **mm**.

Create the outer profile of the seal.

### *Bézier Polygon 1 (b1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.
- 2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
- 3 Find the **Added segments** subsection. Click **Add Quadratic**.
- 4 Find the **Control points** subsection. In row **1**, set **x** to 8.
- 5 In row **2**, set **x** to 12.
- 6 In row **3**, set **x** to 12 and **y** to 4.
- 7 Find the **Added segments** subsection. Click **Add Linear**.
- 8 Find the **Control points** subsection. In row **2**, set **y** to 8.
- 9 Find the **Added segments** subsection. Click **Add Quadratic**.
- 10 Find the **Control points** subsection. In row **2**, set **y** to 12.
- 11 In row **3**, set **x** to 8 and **y** to 12.
- 12 Find the **Added segments** subsection. Click **Add Linear**.
- 13 Find the **Control points** subsection. In row **2**, set **x** to 0.
- 14 Find the **Added segments** subsection. Click **Add Quadratic**.
- 15 Find the **Control points** subsection. In row **2**, set **x** to -6.
- 16 In row **3**, set **x** to -6 and **y** to 6.
- 17 Find the **Added segments** subsection. Click **Add Quadratic**.
- 18 Find the **Control points** subsection. In row **2**, set **y** to 0.

- 19 In row **3**, set **x** to 0 and **y** to 0.
- 20 Find the **Added segments** subsection. Click **Add Linear**.
- 21 Find the **Control points** subsection. Click **Close Curve**.
- 22 Click **Build Selected**.
- 23 Right-click **Bézier Polygon 1 (b1)** and choose **Duplicate**.

Create the Inner profile of the seal.

*Bézier Polygon 2 (b2)*

- 1 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
- 2 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 1 (quadratic)**.
- 3 Find the **Control points** subsection. In row **1**, set **y** to 1.5.
- 4 In row **2**, set **x** to 10.5 and **y** to 1.5.
- 5 In row **3**, set **x** to 10.5.
- 6 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 2 (linear)**.
- 7 Find the **Control points** subsection. In row **2**, set **x** to 10.5.
- 8 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 3 (quadratic)**.
- 9 Find the **Control points** subsection. In row **2**, set **x** to 10.5 and **y** to 10.5.
- 10 In row **3**, set **y** to 10.5.
- 11 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 4 (linear)**.
- 12 Find the **Control points** subsection. In row **2**, set **y** to 10.5.
- 13 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 5 (quadratic)**.
- 14 Find the **Control points** subsection. In row **2**, set **x** to -4.5 and **y** to 10.5.
- 15 In row **3**, set **x** to -4.5.
- 16 Find the **Added segments** subsection. In the **Added segments** list, select **Segment 6 (quadratic)**.
- 17 Find the **Control points** subsection. In row **2**, set **x** to -4.5 and **y** to 1.5.
- 18 In row **3**, set **y** to 1.5.

**19** Find the **Added segments** subsection. In the **Added segments** list, select **Segment 7 (linear)**.

**20** Find the **Control points** subsection. In row **2**, set **y** to 1.5.

**21** Click **Build Selected**.

#### *Difference 1 (dif1)*

**1** On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

**2** Select the object **b1** only.

**3** In the **Settings** window for **Difference**, locate the **Difference** section.

**4** Find the **Objects to subtract** subsection. Select the **Active** toggle button.

**5** Select the object **b2** only.

**6** Locate the **Selections of Resulting Entities** section. Click **New**.

**7** In the **New Cumulative Selection** dialog box, type Seal in the **Name** text field.

**8** Click **OK**.

**9** In the **Settings** window for **Difference**, click **Build Selected**.

#### *Explicit Selection 1 (sel1)*

**1** On 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** On the object **dif1**, select one of the boundaries on the inside of the seal.

#### *Explicit Selection 2 (sel2)*

**1** On 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** On the object **dif1**, select one of the boundaries on the outside of the seal

#### *Explicit Selection 3 (sel3)*

**1** On 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 `dif1`, select **Boundary 1** only.

Create the support.

#### *Bézier Polygon 3 (b3)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.
- 2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
- 3 Find the **Added segments** subsection. Click **Add Linear**.
- 4 Find the **Control points** subsection. In row **1**, set **x** to -7 and **y** to -2.
- 5 In row **2**, set **x** to 15 and **y** to -2.
- 6 Find the **Added segments** subsection. Click **Add Linear**.
- 7 Find the **Control points** subsection. In row **2**, set **y** to 10.
- 8 Find the **Added segments** subsection. Click **Add Linear**.
- 9 Find the **Control points** subsection. In row **2**, set **x** to 13.
- 10 Find the **Added segments** subsection. Click **Add Linear**.
- 11 Find the **Control points** subsection. In row **2**, set **y** to 0.
- 12 Find the **Added segments** subsection. Click **Add Linear**.
- 13 Find the **Control points** subsection. In row **2**, set **x** to -7.
- 14 Find the **Added segments** subsection. Click **Add Linear**.
- 15 Find the **Control points** subsection. Click **Close Curve**.
- 16 Locate the **Selections of Resulting Entities** section. Click **New**.
- 17 In the **New Cumulative Selection** dialog box, type `Rigid base` in the **Name** text field.
- 18 Click **OK**.
- 19 In the **Settings** window for **Bézier Polygon**, click **Build Selected**.

Create the indenter.

#### *Circle 1 (c1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 12.

- 4 Locate the **Position** section. In the **x** text field, type 4.
- 5 In the **y** text field, type 24.
- 6 Right-click **Circle 1 (c1)** and choose **Build Selected**.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

#### *Rectangle 1 (r1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 22.
- 4 In the **Height** text field, type 6.
- 5 Locate the **Position** section. In the **x** text field, type -7.
- 6 In the **y** text field, type 12.
- 7 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

#### *Intersection 1 (int1)*

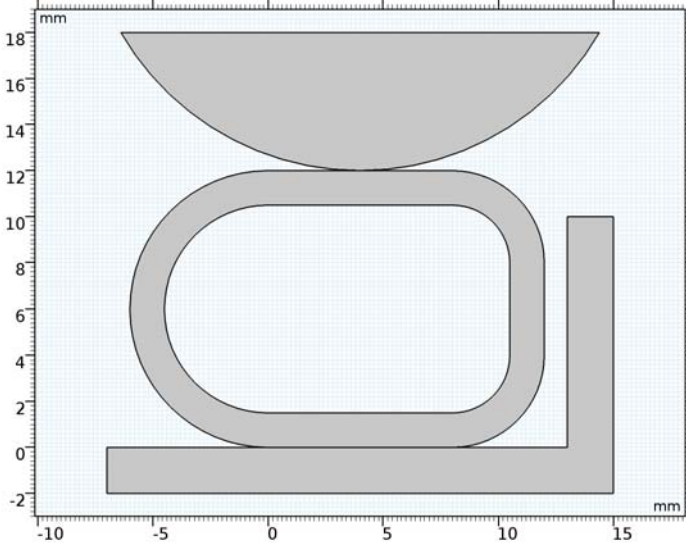
- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.
- 2 Select the objects **r1** and **c1** only.
- 3 In the **Settings** window for **Intersection**, locate the **Selections of Resulting Entities** section.
- 4 Click **New**.
- 5 In the **New Cumulative Selection** dialog box, type **Indentor** in the **Name** text field.
- 6 Click **OK**.
- 7 Right-click **Intersection 1 (int1)** and choose **Build Selected**.
- 8 Click the **Zoom Extents** button on the **Graphics** toolbar.

#### *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 Right-click **Component 1 (comp1)**>**Geometry 1**>**Form Union (fin)** and choose **Build Selected**.

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, under **Global Definitions** click **Parameters**.
- 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

## DEFINITIONS

### Contact Pair 1 (p1)

- 1 On 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 **Indentor**.

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

#### *Contact Pair 2 (p2)*

- 1 On 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.

### **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 50[mm].

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

#### *Hyperelastic Material 1*

- 1 On 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 **Seal**.
- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney-Rivlin, two parameters**.
- 5 In the  $\kappa$  text field, type 1e4[MPa].

#### *Fixed Constraint 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **Fixed Constraint**.
- 2 In the **Settings** window for **Fixed Constraint**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Rigid base**.

#### *Prescribed Displacement 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **Prescribed Displacement**.
- 2 In the **Settings** window for **Prescribed Displacement**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Indentor**.

4 Locate the **Prescribed Displacement** section. Select the **Prescribed in x direction** check box.

Leave the value in the  $u_0$  text field at 0.

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

6 In the  $u_{0y}$  text field, type  $-\text{para} \cdot 1$  [mm].

#### *Contact 1*

1 On the **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Contact**.

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

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

4 Locate the **Penalty Factor** section. From the **Penalty factor control** list, choose **Manual tuning**.

5 From the **Use relaxation** list, choose **Conditional**.

6 In the **Suppression criterion** field, type  $\text{para} > 0.3$

During the initial compression step, when contact is first established, it is reasonable to use the default, stable, approach with relaxation of the penalty factor. For the rest of the analysis, this is not necessary, and it is less time consuming to use a somewhat more aggressive scheme.

#### *Friction 1*

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

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

3 In the  $\mu_{\text{stat}}$  text field, type 0.2.

#### *Contact 2*

1 On the **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Contact**.

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

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

4 Locate the **Penalty Factor** section. From the **Tuned for** list, choose **Speed**.

#### *Friction 1*

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

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

3 In the  $\mu_{\text{stat}}$  text field, type 0.2.

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

### Spring Foundation 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.
- 2 In the **Settings** window for **Spring Foundation**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Glued seal boundary**.
- 4 Locate the **Spring** section. From the list, choose **Diagonal**.
- 5 In the  $k_A$  table, enter the following settings:

1e10	0
0	2e10

## DEFINITIONS

### Integration 1 (intop1)

- 1 On the **Definitions** toolbar, click **Component 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 On 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
EnclosedArea	AreaInt(-x*solid.nx)	m <sup>2</sup>	
int_p	0.1[MPa]*((123.63[mm <sup>2</sup> ]/EnclosedArea) <sup>1.4-1</sup> )	Pa	

## SOLID MECHANICS (SOLID)

### Boundary Load 1

- 1 On 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.

## 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 **Geometric Entity Selection** section.
- 3 From the **Selection** list, choose **Seal**.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1100 [ kg/m <sup>3</sup> ]	kg/m <sup>3</sup>	Basic
Model parameters	C10	0.37 [MPa]	Pa	Mooney-Rivlin
Model parameters	C01	0.11 [MPa]	Pa	Mooney-Rivlin

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.

## ADD MATERIAL

- 1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Built-In>Structural steel**.
- 4 Click **Add to Component** in the window toolbar.
- 5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

## MATERIALS

### *Structural steel (mat2)*

Select Domains 1 and 2 only.

Add an integration operator to get reaction force for postprocessing.

## DEFINITIONS

### *Integration 2 (intop2)*

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, type `IndentInt` in the **Operator name** text field.
- 3 Select Domain 2 only.
- 4 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

## MESH 1

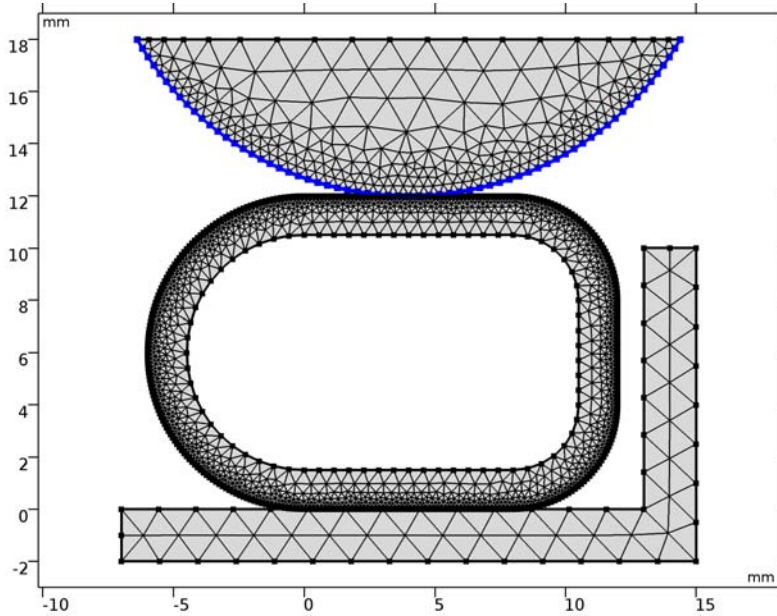
### *Size 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.
- 2 Right-click **Free Triangular 1** and choose **Size**.
- 3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 4 From the **Geometric entity level** list, choose **Boundary**.
- 5 From the **Selection** list, choose **Outer seal boundary**.
- 6 Locate the **Element Size** section. Click the **Custom** button.
- 7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 8 In the associated text field, type 0.15.

### *Size 2*

- 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 Select Boundaries 8 and 9 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 text field, type 0.4.

**8 Click Build All.**



**STUDY I**

*Step 1: Stationary*

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

- 1** In the **Model Builder** window, expand the **Study I** node, then click **Step 1: Stationary**.
- 2** In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3** Select the **Modify physics tree and variables 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.

- 6** Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 7** Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Vertical displacement parameter)	range (0, 0.25, 4)	

9 In the **Model Builder** window, click **Study I**.

10 In the **Settings** window for **Study**, type Study: without Pressure in the **Label** text field.

*Solution I (sol1)*

On the **Study** toolbar, click **Show Default Solver**.

### **STUDY: WITHOUT PRESSURE**

*Solution I (sol1)*

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.

1 In the **Model Builder** window, expand the **Solution I (sol1)** node.

2 In the **Model Builder** window, expand the **Study: without Pressure>Solver Configurations>Solution I (sol1)>Dependent Variables I** node, then click **Displacement field (material and geometry frames) (comp1.u)**.

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

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

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

Variable name	Scaling
Displacement field (Material) (comp1.u)	$1e-3$
Auxiliary pressure (comp1.solid.pw)	$1e5$
Contact pressure (comp1.solid.Tn_upper)	$1e5$
Contact pressure (comp1.solid.Tn_lower)	$1e5$
Friction force (Spatial) (comp1.solid.Tt_upper)	$1e4$
Friction force (Spatial) (comp1.solid.Tt_lower)	$1e4$

*Step 1: Stationary*

1 In the **Model Builder** window, under **Study: without Pressure** click **Step 1: Stationary**.

- 2 In the **Settings** window for **Stationary**, click to expand the **Results while solving** section.
- 3 Locate the **Results While Solving** section. Select the **Plot** check box.
- 4 On the **Study** toolbar, click **Compute**.

## RESULTS

### *Stress (solid)*

- 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 Click the **Zoom Extents** button on 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.

### *ID Plot Group 2*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID 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 (1-17)** text field, type **range(2,3,17)**.
- 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 13, 17, and 23** 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>Solid Mechanics>Contact>Contact 1>solid.cnt1.Tn - Contact pressure**.
- 4 Click to expand the **Legends** section. Select the **Show legends** check box.
- 5 On 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 On 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 **Preset Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.
- 5 On 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)	range (0,0.25,4)	

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

### *Stress (solid)*

On the **Study** toolbar, click **Get Initial Value**.

## STUDY: WITH PRESSURE

### *Step 1: Stationary*

- 1 In the **Settings** window for **Stationary**, click to expand the **Results while solving** section.
- 2 Locate the **Results While Solving** section. Select the **Plot** check box.
- 3 From the **Plot group** list, choose **Stress (solid)**.
- 4 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 (material and geometry frames) (comp1.u)**.

- 2 In the **Settings** window for **Field**, locate the **Scaling** section.
- 3 In the **Scale** text field, type  $1e-3$ .
- 4 Scale the fields in the same way as you did in Study 1.

Variable name	Scaling
Displacement field (Material) (comp l.u)	$1e-3$
Auxiliary pressure (comp l.solid.pw)	$1e5$
Contact pressure (comp l.solid.Tn_upper)	$1e5$
Contact pressure (comp l.solid.Tn_lower)	$1e5$
Friction force (Spatial) (comp l.solid.Tt_upper)	$1e4$
Friction force (Spatial) (comp l.solid.Tt_lower)	$1e4$

- 5 On the **Study** toolbar, click **Compute**.

## RESULTS

### *Stress (solid)*

- 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 Click the **Zoom Extents** button on 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 without Confined Air 1*

- 1 In the **Model Builder** window, under **Results** right-click **Contact Pressure without Confined Air** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type **Contact Pressure with Confined Air** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study: with Pressure/ Solution 2 (sol2)**.
- 4 On 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 indenter.

### *ID Plot Group 5*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.

- 2 In the **Settings** window for **ID Plot Group**, type **Compressive Force vs Compression** in the **Label** text field.
- 3 On the **Compressive Force vs Compression** toolbar, click **Global**.

*Global 1*

- 1 In the **Model Builder** window, under **Results>Compressive Force vs Compression** click **Global 1**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
IndentInt(-solid.RFy/solid.d)	N/m	Without confined air

*Global 2*

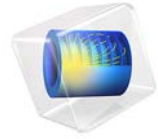
- 1 Right-click **Results>Compressive Force vs Compression>Global 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study: with Pressure/Solution 2 (sol2)**.
- 4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
IndentInt(-solid.RFy/solid.d)	N/m	With confined air

*Compressive Force vs Compression*

- 1 In the **Model Builder** window, under **Results** click **Compressive Force vs Compression**.
- 2 In the **Settings** window for **ID Plot Group**, click to expand the **Title** section.
- 3 From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type **Compressive force vs compression**.
- 5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 6 Select the **y-axis label** check box.
- 7 In the **x-axis label** text field, type **Indentation (mm)**.
- 8 In the **y-axis label** text field, type **Force (N/mm)**.
- 9 Locate the **Legend** section. From the **Position** list, choose **Upper left**.  
Compare with the plot shown in [Figure 6](#).





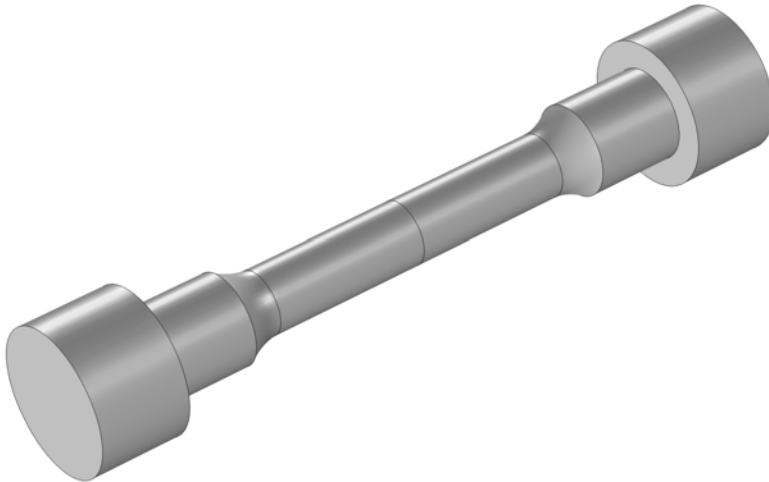
# Lemaitre-Chaboche Viscoplastic Model

## *Introduction*

---

Most metals and alloys undergo viscoplastic deformation at high temperatures. In case of cyclic loading, a constitutive law with both isotropic and kinematic hardening is necessary to describe effects such as ratcheting, cyclic softening/hardening, and stress relaxation. The Lemaitre-Chaboche viscoplastic model combines isotropic hardening with nonlinear kinematic hardening to model these effects.

This tutorial model demonstrates the uniaxial deformation of an indium IN 100 test specimen submitted to cyclic tension-compression loading at high temperature, as described in [Ref. 1](#)



*Figure 1: Specimen used for the uniaxial tension-compression tests.*

## *Model Definition*

---

The test specimen consists of a small cylinder with a thin central section to ensure uniform stress and strain distributions, furnished with thick extremities to ease the mounting in a universal testing machine. The loading for the tension-compression cycle is directed in the axial direction. For shortening the computation time, half of the specimen is modeled in a 2D axisymmetric geometry.

## GOVERNING EQUATIONS

The strain tensor consists of the sum of the elastic strain tensor  $\epsilon_{el}$  and the viscoelastic strain tensor  $\epsilon_{vp}$ :

$$\epsilon = \epsilon_{el} + \epsilon_{vp}$$

The constitutive law between stress and strain is given by Hooke's law

$$\sigma = \mathbf{C}:(\epsilon - \epsilon_{vp})$$

The viscoplastic strain tensor is calculated by Lemaitre-Chaboche viscoplastic rule

$$\dot{\epsilon}_{vp} = \frac{3}{2} \dot{\epsilon}_{vpe} \frac{\sigma' - \sigma_b'}{J_2(\sigma - \sigma_b)}$$

here,  $\sigma_b$  is the back-stress tensor, derived by the nonlinear kinematic hardening rule; and  $\sigma'$  and  $\sigma_b'$  are the deviatoric parts of the stress and back-stress tensors  $\sigma$  and  $\sigma_b$ .

The effective viscoplastic strain rate is defined by the expression

$$\dot{\epsilon}_{vpe} = \max\left(A \left(\frac{J_2(\sigma - \sigma_b) - \sigma_y}{\sigma_{ref}}\right)^n, 0\right)$$

here,  $A$  is the viscoplastic rate coefficient,  $\sigma_{ref}$  is a reference stress,  $n$  is the stress exponent, and  $\sigma_y$  is the yield stress given by the nonlinear isotropic hardening rule.

## MIXED HARDENING

The isotropic hardening model represents the change in yield stress as a function of the effective viscoplastic strain. Lemaitre and Chaboche (Ref. 1) derived a nonlinear isotropic hardening relation of the type

$$\sigma_y = \sigma_{ys0} + \sigma_{sat} \left(1 - e^{-\beta \epsilon_{vpe}}\right)$$

here,  $\sigma_{ys0}$  is the initial yield stress,  $\epsilon_{vpe}$  is the effective viscoplastic strain, and  $\sigma_{sat}$  and  $\beta$  are material parameters.

The kinematic hardening rule represents the translation of the yield surface as a function of the back-stress  $\sigma_b$  and the effective viscoplastic strain  $\epsilon_{vp}$ . Lemaitre and Chaboche (Ref. 1) derived a nonlinear kinematic hardening relation where the back-stress tensor is calculated from the ordinary differential equation

$$\dot{\sigma}_b = \frac{2}{3} C_k \dot{\epsilon}_{vp} - \gamma_k \dot{\epsilon}_{vpe} \sigma_b$$

here  $C_k$  and  $\gamma_k$  are material parameters. The kinematic hardening parameter  $\gamma_k$  also depends exponentially on the effective viscoplastic strain:

$$\gamma_k = \gamma_s + (\gamma_0 - \gamma_s) e^{-\beta_k \epsilon_{vpe}}$$

here  $\gamma_s$ ,  $\gamma_0$ , and  $\beta_k$  are material parameters.

### MATERIAL DATA

The material parameters of the indium alloy IN 100 at a temperature of 900 K are given in [Ref. 1](#).

TABLE 1: MATERIAL PARAMETERS, IN 100 AT 900 K.

PARAMETER	
$K$	490 MPa.s <sup>1/n</sup>
$n$	9
$\sigma_{y0}$	60 MPa
$\sigma_{y0} + \sigma_{sat}$	25 MPa
$b$	200
$C_k$	362,500 MPa
$\gamma_0$	1,200
$\gamma_s$	1,540
$\beta_k$	1,000

In [Ref. 1](#), the parameter  $K$  is given in units of MPa.s<sup>1/n</sup>. The parameters  $K$ ,  $A$ , and  $\sigma_{ref}$  are related by the expression

$$\frac{A}{(\sigma_{ref})^n} = \frac{1}{K^n}$$

Use the values  $A = 1/s$  and  $\sigma_{ref} = 490$  MPa to obtain similar results as in given in [Ref. 1](#).

### **STRESS VS TIME**

After the first tensile cycle the axial stress (blue line in [Figure 2](#)) increases linearly, exceeding the yield strength (cyan line). Because of viscous effects, the onset of plasticity starts once the viscous stress (green line) reaches the yield strength.

At this point, the relation between stress and strain is no longer linear, kinematic/isotropic hardening evolves, and the back-stress (red line) increases.

When the maximum allowed tensile strain is reached, the prescribed axial velocity turns negative to prescribe axial unloading. The axial stress then linearly decreases, and the back stress remains constant to account for kinematic hardening and the Bauschinger effect. The yield strength in compression after a tensile cycle is lower than the initial yield strength.

In the compressive cycle, the viscoplastic flow starts earlier and lasts for a longer period. When the maximum allowed compressive strain is reached, the prescribed axial velocity becomes positive to prescribe tensile unloading. This procedure is repeated for each load cycle.

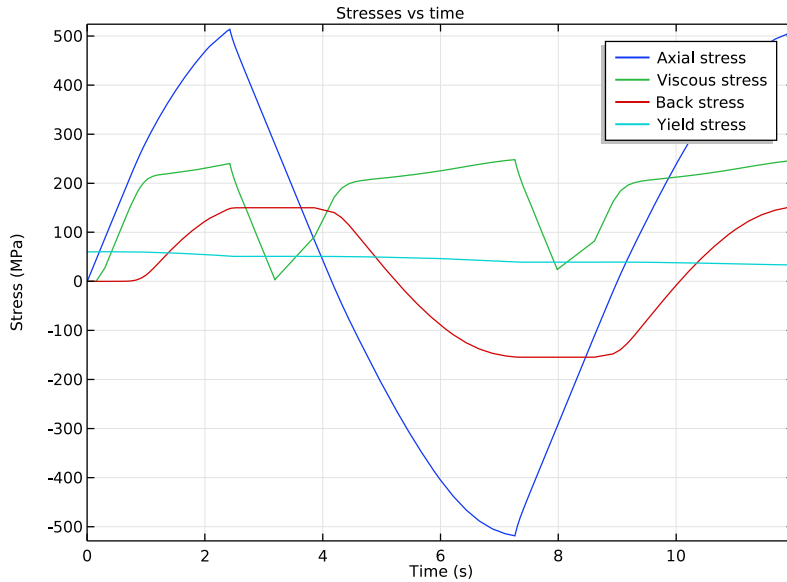


Figure 2: Evolution of the axial stress variables versus time.

Four studies are performed to prescribe four different types of loading cycles: prescribed symmetric strain, prescribed symmetric stress, prescribed nonsymmetric strain, and prescribed nonsymmetric stress. The stress-strain graph for each type of loading cycle highlights material properties of the Lemaitre-Chaboche viscoplastic model.

### SYMMETRIC CYCLES

Symmetric loading aims to show the stabilization effect of the hysteresis cycle; this emphasizes the softening or hardening effect during cyclic loadings for a given hardening rule.

For a symmetric prescribed axial strain cycle (Figure 3), there is a periodic response to the periodic load and a stabilized state is reached after few cycles. A closer view shows that the stress amplitude decreases during the first cycles due to material softening.

Similar effect can be seen in a symmetric prescribed stress cycle (Figure 4), but more cycles are needed to reach a stabilized state as compared to the prescribed strain loading; also, the axial strain amplitude increases with the number of cycles.

This is an effect of the isotropic hardening rule. With kinematic hardening only, the stabilized state is reached after the first cycle of prescribed stress. Thus, a mixed hardening formulation is needed to model both cyclic softening/hardening and Bauschinger effects.

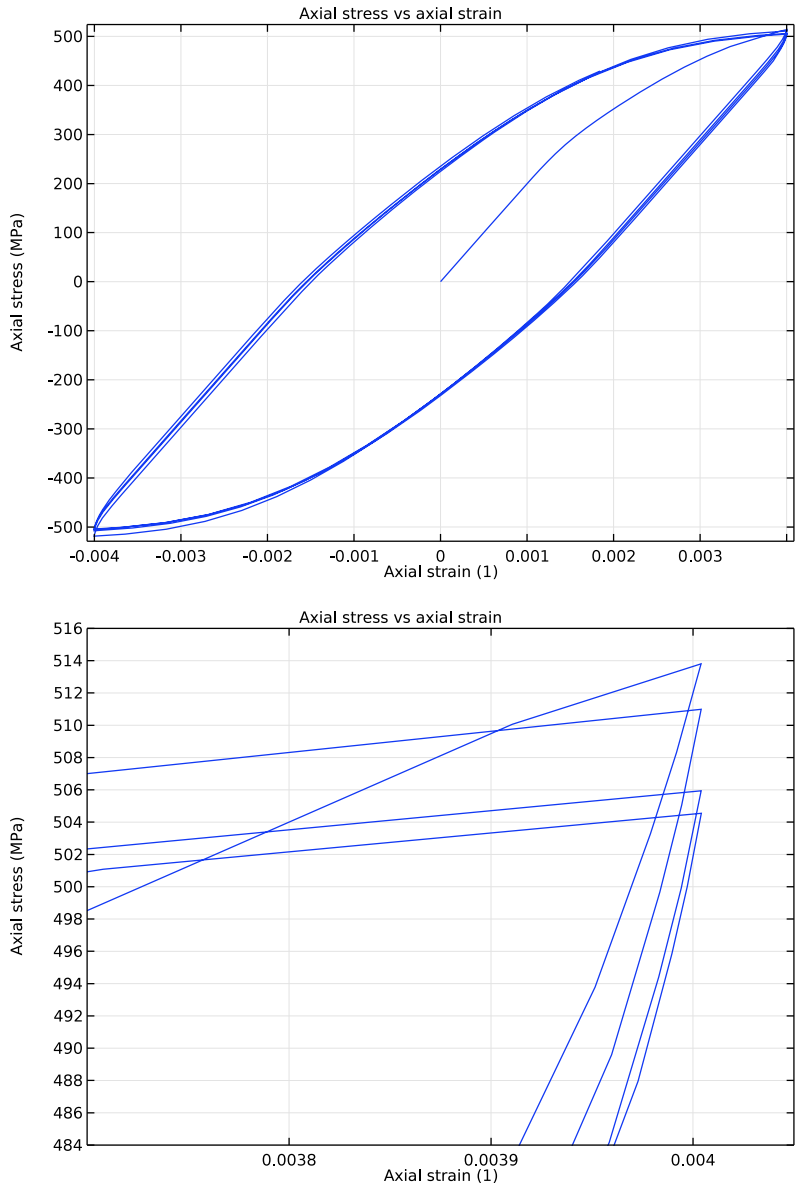


Figure 3: Axial stress vs axial strain for a symmetric prescribed strain loading.

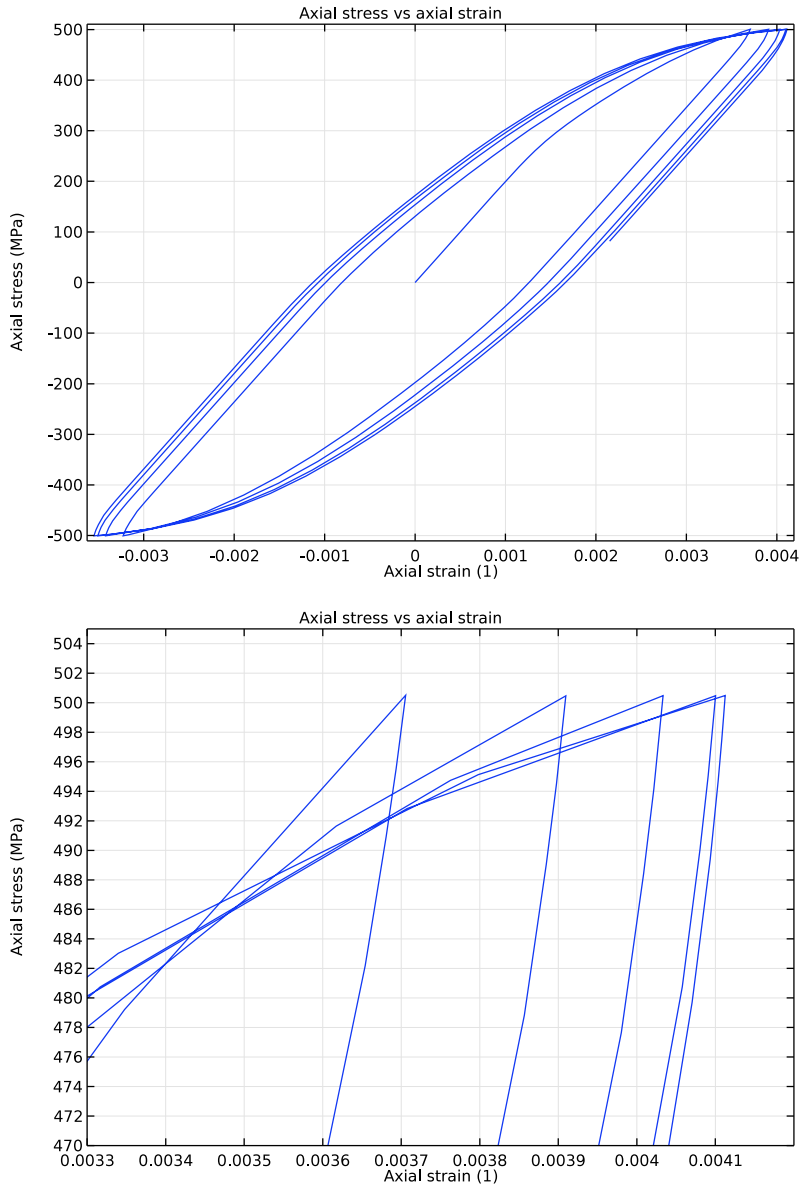


Figure 4: Axial stress vs axial strain for a symmetric prescribed stress loading.

## NONSYMMETRIC STRAIN CYCLES

When the average strain is not zero in a prescribed strain cycle (Figure 5), the initial asymmetry in the axial stress gradually disappears over the cycles: This happens due to the stress relaxation effect, which is observed in many alloys.

Once a stabilized cycle is reached, the tensile and compressive stresses are equal in absolute value. This is an effect of the nonlinear kinematic hardening rule. Applying linear kinematic hardening only (or isotropic hardening) would not allow to observe the relaxation of the mean stress, so a nonlinear kinematic hardening rule is needed to model stress relaxation.

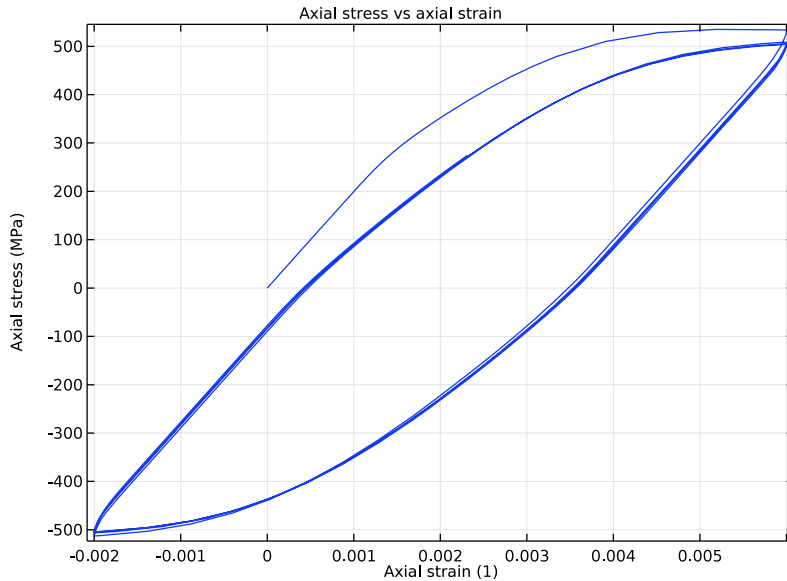


Figure 5: Axial stress vs axial strain for nonsymmetric prescribed strain loading.

## NONSYMMETRIC STRESS CYCLES

The gradual deformation that occurs in tension-compression cycles when the mean stress is non-zero is called the ratcheting effect. The ratcheting effect is maximal when the lower stress limit is  $-\sigma_{ys0}$ . This behavior is possible to observe thanks to the nonlinear kinematic hardening rule.

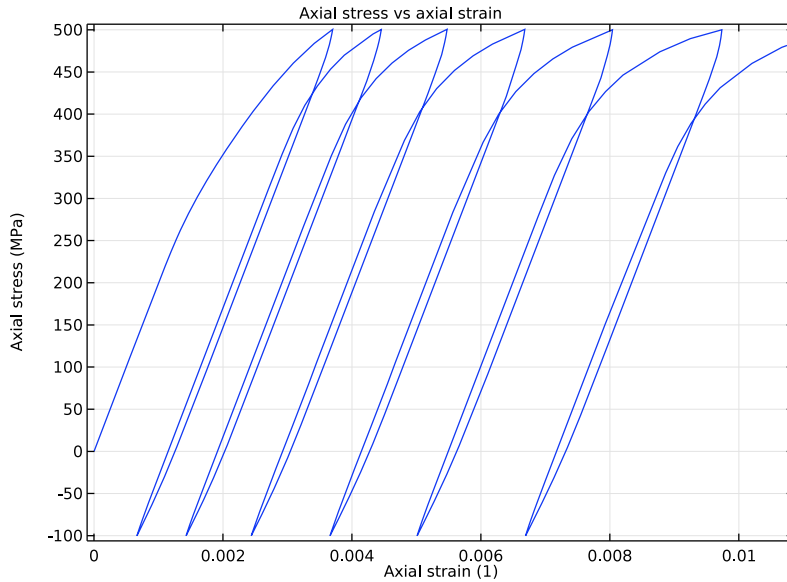


Figure 6: Axial stress vs axial strain for a nonsymmetric prescribed stress loading.

### Notes About the COMSOL Implementation

Several yield functions and plastic potentials are available in the Viscoplasticity feature. Use von Mises effective stress to reproduce the results described in [Ref. 1](#).

Solve four studies to account for the different types of load cycles: the load cycle can be symmetric or nonsymmetric, and the control can be done by prescribing either the axial strain or the axial stress.

Apply a constant axial velocity on one end of the test specimen to avoid using complicate loading functions. Multiply this axial velocity by 1 to represent axial tension, or by  $-1$  to represent axial compression.

To achieve this, define a discrete state called `LoadingType` in an **Event** interface. Add **Indicator States** in the **Event** interface to define tension and compression limits based on

the stress or strain state. For instance, for a symmetric loading cycle with a prescribed strain of 0.4 % the following **Indicator States** are used:

---

Tension	intop1(solid.e133)-0.004
Compression	intop1(solid.e133)+0.004

---

Then add two **Implicit Event** nodes to define the discrete state `LoadingType`: when `Tension > 0`, then `LoadingType` is set to `-1`, and when `Compression < 0`, `LoadingType` is set to `1`.

### *Reference*

---

1. J. Lemaitre and J.-L. Chaboche, *Mécanique des matériaux solides*, 2<sup>nd</sup> ed., Dunod, 2001.

---

**Application Library path:** `Nonlinear_Structural_Materials_Module/Viscoplasticity/lemaitre_chaboche_viscoplastic_model`

---

### *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 Axisymmetric**.
- 2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 3 Click **Add**.
- 4 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Events (ev)**.
- 5 Click **Add**.
- 6 Click **Study**.
- 7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.
- 8 Click **Done**.

## GLOBAL DEFINITIONS

### Parameters

Set the parameter that will be needed in the model.

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
e0t	$1e-3[1/s]$	0.001 1/s	Prescribed strain rate
L0	$(35/2+14+10)$ [mm]	0.0415 m	Half length

Add a step function to apply load smoothly.

### Step 1 (step1)

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Step**.
- 2 In the **Settings** window for **Step**, locate the **Parameters** section.
- 3 In the **Location** text field, type 0.005.
- 4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 0.01.
- 5 Click **Plot**.

## GEOMETRY I

- 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**.

### Rectangle 1 (r1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 7/2.
- 4 In the **Height** text field, type 35/2.

### Rectangle 2 (r2)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 5.

- 4 In the **Height** text field, type 10.
- 5 Locate the **Position** section. In the **z** text field, type 21.5.

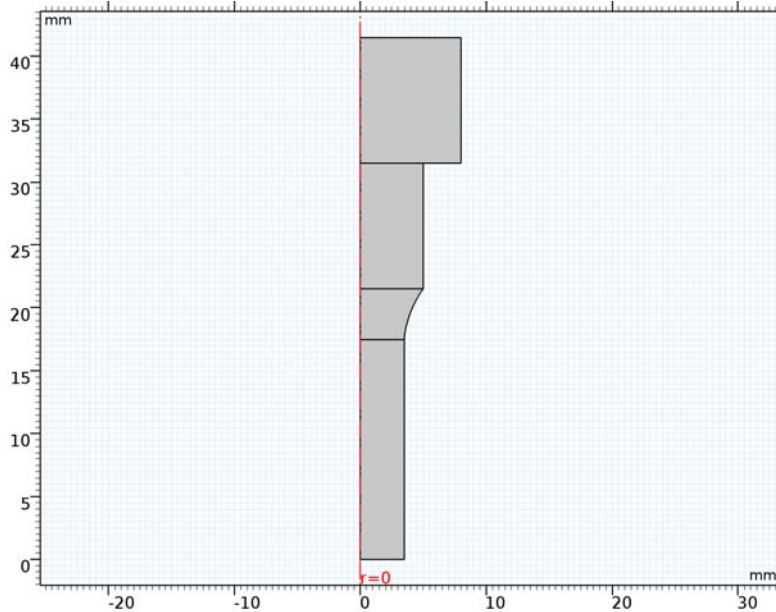
*Rectangle 3 (r3)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **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 10.
- 5 Locate the **Position** section. In the **z** text field, type 31.5.

*Bézier Polygon 1 (b1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.
- 2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
- 3 Find the **Added segments** subsection. Click **Add Quadratic**.
- 4 Find the **Control points** subsection. In row **1**, set **r** to 3.5.
- 5 In row **1**, set **z** to 17.5.
- 6 In row **2**, set **r** to 3.5.
- 7 In row **2**, set **z** to 19.5.
- 8 In row **3**, set **r** to 5.
- 9 In row **3**, set **z** to 21.5.
- 10 Find the **Added segments** subsection. Click **Add Linear**.
- 11 Find the **Control points** subsection. In row **2**, set **r** to 0.
- 12 Find the **Added segments** subsection. Click **Add Linear**.
- 13 Find the **Control points** subsection. In row **2**, set **z** to 17.5.
- 14 Find the **Added segments** subsection. Click **Add Linear**.
- 15 Find the **Control points** subsection. Click **Close Curve**.

**16 Click Build All Objects.**



Add conditions to toggle the boundary conditions between tension and compression. This is controlled by the **Events** interface. First create an integration operator to get variable from a point.

*Integration 1 (intop1)*

On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

**DEFINITIONS**

*Integration 1 (intop1)*

- 1 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 2 From the **Geometric entity level** list, choose **Point**.
- 3 Select Point 6 only.
- 4 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

**EVENTS (EV)**

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

*Discrete States 1*

- 1 On the **Physics** toolbar, click **Global** and choose **Discrete States**.

- 2 In the **Settings** window for **Discrete States**, locate the **Discrete States** section.
- 3 In the table, enter the following settings:

Name	Initial value (u0)	Description
LoadingType	1	

The LoadingType variable is 1 when tension is applied and -1 when compression is applied.

The first indicator state is used to control the loading with strain. The tension and compression limits are symmetric.

#### *Indicator States 1*

- 1 On the **Physics** toolbar, click **Global** and choose **Indicator States**.
- 2 In the **Settings** window for **Indicator States**, type Indicator States: Strain, Symmetric in the **Label** text field.
- 3 Locate the **Indicator Variables** section. In the table, enter the following settings:

Name	$g(v,vt,vtt,t)$
Tension	$\text{intop1}(\text{solid.e133}) - 0.004$
Compression	$\text{intop1}(\text{solid.e133}) + 0.004$

The second indicator state is used to control the loading with stress. The tension and compression limits are symmetric.

#### *Indicator States 2*

- 1 On the **Physics** toolbar, click **Global** and choose **Indicator States**.
- 2 In the **Settings** window for **Indicator States**, type Indicator States: Stress, Symmetric in the **Label** text field.
- 3 Locate the **Indicator Variables** section. In the table, enter the following settings:

Name	$g(v,vt,vtt,t)$
Tension	$\text{intop1}(\text{solid.s133}) - 500[\text{MPa}]$
Compression	$\text{intop1}(\text{solid.s133}) + 500[\text{MPa}]$

The third indicator state is used to control the loading with strain. The tension and compression limits are not symmetric

#### *Indicator States 3*

- 1 On the **Physics** toolbar, click **Global** and choose **Indicator States**.

2 In the **Settings** window for **Indicator States**, type Indicator States: Strain, Nonsymmetric in the **Label** text field.

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

Name	$g(v,vt,vtt,t)$
Tension	$\text{intop1}(\text{solid.e133}) - 0.006$
Compression	$\text{intop1}(\text{solid.e133}) + 0.002$

The fourth indicator state is used to control the loading with stress. The tension and compression limits are not symmetric

#### *Indicator States 4*

1 On the **Physics** toolbar, click **Global** and choose **Indicator States**.

2 In the **Settings** window for **Indicator States**, type Indicator States: Stress, Nonsymmetric in the **Label** text field.

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

Name	$g(v,vt,vtt,t)$
Tension	$\text{intop1}(\text{solid.s133}) - 500[\text{MPa}]$
Compression	$\text{intop1}(\text{solid.s133}) + 100[\text{MPa}]$

## **EVENTS (EV)**

### *Implicit Event 1*

1 On the **Physics** toolbar, click **Global** and choose **Implicit Event**.

2 In the **Settings** window for **Implicit Event**, locate the **Event Conditions** section.

3 In the **Condition** text field, type Tension>0.

4 Locate the **Reinitialization** section. In the table, enter the following settings:

Variable	Expression
LoadingType	-1

### *Implicit Event 2*

1 On the **Physics** toolbar, click **Global** and choose **Implicit Event**.

2 In the **Settings** window for **Implicit Event**, locate the **Event Conditions** section.

3 In the **Condition** text field, type Compression<0.

4 Locate the **Reinitialization** section. In the table, enter the following settings:

Variable	Expression
LoadingType	1

#### **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 **Structural Transient Behavior** section.
- 3 From the list, choose **Quasi-static**.

##### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 Select Boundary 2 only.

##### *Prescribed Velocity 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Velocity**.
- 2 Select Boundary 9 only.
- 3 In the **Settings** window for **Prescribed Velocity**, locate the **Prescribed Velocity** section.
- 4 Select the **Prescribed in z direction** check box.
- 5 In the  $v_z$  text field, type  $e0t*L0*step1(t[1/s])*LoadingType$ .

##### *Linear Elastic Material 1*

Add viscoplasticity with combined isotropic and kinematic hardening.

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

##### *Viscoplasticity 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Viscoplasticity**.
- 2 Select Domains 1–3 only.
- 3 In the **Settings** window for **Viscoplasticity**, locate the **Viscoplasticity Model** section.
- 4 From the **Viscoplasticity model** list, choose **Chaboche**.
- 5 Find the **Isotropic hardening model** subsection. From the list, choose **Voce**.
- 6 Find the **Kinematic hardening model** subsection. From the list, choose **Armstrong-Frederick**.
- 7 From the  $\gamma_k$  list, choose **User defined**.

The kinematic hardening parameter is non-linear. Add a variable to set its expression.

## DEFINITIONS

### *Variables 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
gamma0	1200		Initial kinematic hardening parameter
gammas	1540		Saturation kinematic hardening parameter
beta	1000		Kinematic hardening parameter exponent
gammak	$\text{gammas} + (\text{gamma0} - \text{gammas}) * \exp(-\text{beta} * \text{solid.evpe})$		Kinematic hardening parameter

## SOLID MECHANICS (SOLID)

### *Viscoplasticity 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1** click **Viscoplasticity 1**.
- 2 In the **Settings** window for **Viscoplasticity**, locate the **Viscoplasticity Model** section.
- 3 Find the **Kinematic hardening model** subsection. In the  $\gamma_k$  text field, type gammak.

Set the material properties.

## 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
Young's modulus	E	200 [ GPa ]	Pa	Basic
Poisson's ratio	nu	0.3	l	Basic
Density	rho	7500	kg/m <sup>3</sup>	Basic
Viscoplastic rate coefficient	A_cha	1	l/s	Chaboche viscoplasticity
Reference stress	sigRef_cha	490 [ MPa ]	N/m <sup>2</sup>	Chaboche viscoplasticity
Stress exponent	n_cha	9	l	Chaboche viscoplasticity
Initial yield stress	sigmags	60 [ MPa ]	Pa	Elastoplastic material model
Saturation flow stress	sigma_voc	-35 [ MPa ]	Pa	Voce
Saturation exponent	beta_voc	200	l	Voce
Kinematic hardening modulus	Ck	362.5 [ GPa ]	Pa	Armstrong-Frederick

Create a mapped mesh.

## MESH 1

### *Distribution 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
- 2 Right-click **Mapped 1** and choose **Distribution**.
- 3 Select Boundary 8 only.
- 4 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 5 In the **Number of elements** text field, type 4.

### *Distribution 2*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundary 12 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 2.

### *Distribution 3*

- 1 Right-click **Mapped I** and choose **Distribution**.
- 2 Select Boundary 10 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 16.
- 5 Select Boundary 1 only.

### *Distribution 4*

- 1 Right-click **Mapped I** and choose **Distribution**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 4.

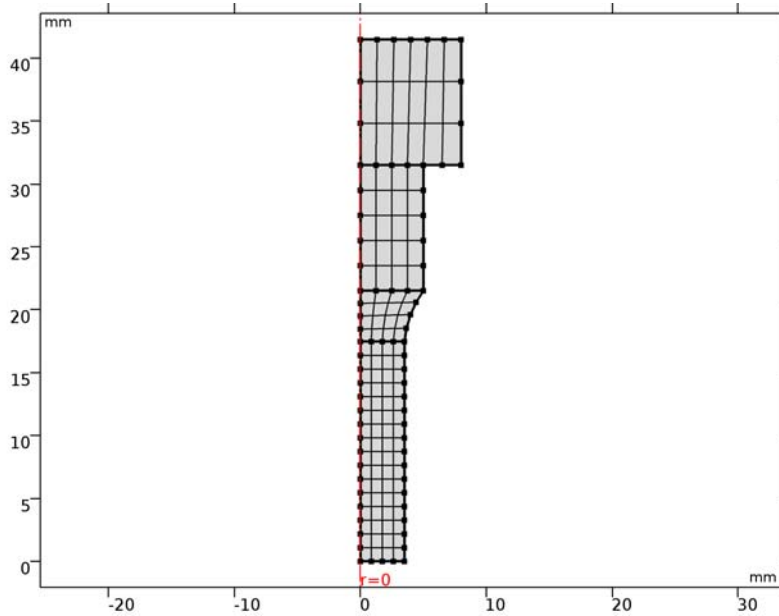
### *Distribution 5*

- 1 Right-click **Mapped I** and choose **Distribution**.
- 2 Select Boundary 5 only.

### *Distribution 6*

- 1 Right-click **Mapped I** and choose **Distribution**.
- 2 Select Boundary 7 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 3.

5 Click **Build All**.



## STUDY I

*Step 1: Time Dependent*

- 1 In the **Model Builder** window, under **Study I** click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 In the **Times** text field, type 0 40.  
For the first study, control the loading with symmetric strain cycles.
- 4 In the **Model Builder** window, click **Step 1: Time Dependent**.
- 5 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.
- 6 Select the **Modify physics tree and variables for study step** check box.
- 7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Events (ev)> Indicator States: Stress, Symmetric, Component 1 (comp1)>Events (ev)> Indicator States: Strain, Nonsymmetric**, and **Component 1 (comp1)>Events (ev)> Indicator States: Stress, Nonsymmetric**.
- 8 Click **Disable**.

### *Solution 1 (sol1)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
- 3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.
- 4 Locate the **Time Stepping** section. In the **Event tolerance** text field, type 0.001.
- 5 Click to expand the **Output** section. From the **Times to store** list, choose **Steps taken by solver**.

Only start and end time are defined in the time step node. The solver sets the time steps automatically. Store results for all time steps.

- 6 Clear the **Store time derivatives** check box.
- 7 In the **Model Builder** window, click **Study 1**.
- 8 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 9 Clear the **Generate default plots** check box.

Plot the loading cycles during the computation. For that, get initial values to create an empty plot group.

### *Study 1*

On the **Study** toolbar, click **Get Initial Value**.

## **RESULTS**

### *ID Plot Group 1*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Stress vs Strain, Prescribed Symmetric Strain in the **Label** text field.
- 3 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 4 In the associated text field, type Axial strain (1).
- 5 Select the **y-axis label** check box.
- 6 In the associated text field, type Axial stress (MPa).
- 7 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 8 In the **Title** text area, type Axial stress vs axial strain.

### *Point Graph 1*

- 1 Right-click **Stress vs Strain, Prescribed Symmetric Strain** and choose **Point Graph**.

- 2 Select Point 6 only.
- 3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor, local coordinate system>solid.sl33 - Stress tensor, local coordinate system, 33 component**.
- 4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 6 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Strain>Strain tensor, local coordinate system>solid.el33 - Strain tensor, local coordinate system, 33 component**.

## STUDY I

### *Step 1: Time Dependent*

- 1 In the **Model Builder** window, under **Study I** click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, click to expand the **Results while solving** section.
- 3 Locate the **Results While Solving** section. Select the **Plot** check box.
- 4 On the **Home** toolbar, click **Compute**.

## RESULTS

The first plot group should be like in [Figure 3](#).

Add a plot group to reproduce [Figure 2](#).

### *ID Plot Group 2*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Stresses vs Time, Prescribed Symmetric Strain** in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type **Stresses vs time**.

### *Point Graph 1*

- 1 Right-click **Stresses vs Time, Prescribed Symmetric Strain** and choose **Point Graph**.
- 2 Select Point 6 only.
- 3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>**

**Stress>Stress tensor, local coordinate system>solid.sl33 - Stress tensor, local coordinate system, 33 component.**

- 4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 6 Click to expand the **Legends** section. Select the **Show legends** check box.
- 7 From the **Legends** list, choose **Manual**.
- 8 In the table, enter the following settings:

---

**Legends**

---

Axial stress

---

*Point Graph 2*

- 1 Right-click **Results>Stresses vs Time, Prescribed Symmetric Strain>Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `max(0, solid.l1emm1.vp11.Fyield)`.
- 4 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

---

Viscous stress

---

*Point Graph 3*

- 1 Right-click **Results>Stresses vs Time, Prescribed Symmetric Strain>Point Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `solid.l1emm1.vp11.S1_back33`.
- 4 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

---

Back stress

---

*Point Graph 4*

- 1 Right-click **Results>Stresses vs Time, Prescribed Symmetric Strain>Point Graph 3** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `solid.l1emm1.vp11.sY`.

4 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

---

Yield stress

---

*Stresses vs Time, Prescribed Symmetric Strain*

- 1 In the **Model Builder** window, under **Results** click **Stresses vs Time, Prescribed Symmetric Strain**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Plot Settings** section.
- 3 Select the **y-axis label** check box.
- 4 In the associated text field, type **Stress (MPa)**.
- 5 On the **Stresses vs Time, Prescribed Symmetric Strain** toolbar, click **Plot**.

Add a study to compute loading controlled by symmetric stress cycles.

**ADD STUDY**

- 1 On 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 **Preset Studies> Time Dependent**.
- 4 Click **Add Study** in the window toolbar.
- 5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

**STUDY 2**

*Step 1: Time Dependent*

- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 In the **Times** text field, type **0 40**.
- 3 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Events (ev)> Indicator States: Strain, Symmetric, Component 1 (comp1)>Events (ev)> Indicator States: Strain, Nonsymmetric**, and **Component 1 (comp1)>Events (ev)> Indicator States: Stress, Nonsymmetric**.
- 5 Click **Disable**.
- 6 In the **Model Builder** window, click **Study 2**.
- 7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

#### *Solution 2 (sol2)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.
- 3 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.
- 4 In the **Event tolerance** text field, type 0.001.
- 5 Locate the **Output** section. From the **Times to store** list, choose **Steps taken by solver**.
- 6 Clear the **Store time derivatives** check box.

#### *Study 2*

On the **Study** toolbar, click **Get Initial Value**.

## **RESULTS**

#### *Stress vs Strain, Prescribed Symmetric Strain 1*

- 1 In the **Model Builder** window, under **Results** right-click **Stress vs Strain, Prescribed Symmetric Strain** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type **Stress vs Strain, Prescribed Symmetric Stress** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

#### *Stresses vs Time, Prescribed Symmetric Strain 1*

- 1 In the **Model Builder** window, under **Results** right-click **Stresses vs Time, Prescribed Symmetric Strain** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type **Stresses vs Time, Prescribed Symmetric Stress** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

## **STUDY 2**

#### *Step 1: Time Dependent*

- 1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, click to expand the **Results while solving** section.
- 3 Locate the **Results While Solving** section. Select the **Plot** check box.
- 4 From the **Plot group** list, choose **Stress vs Strain, Prescribed Symmetric Stress**.

5 On the **Study** toolbar, click **Compute**.

## RESULTS

### *Stress vs Strain, Prescribed Symmetric Stress*

Add a study to compute loading controlled by nonsymmetric strain cycles.

## ADD STUDY

- 1 On the **Study** 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 **Preset Studies> Time Dependent**.
- 4 Click **Add Study** in the window toolbar.
- 5 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

## STUDY 3

### *Step 1: Time Dependent*

- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 In the **Times** text field, type 0 40.
- 3 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component I (comp1)>Events (ev)> Indicator States: Strain, Symmetric, Component I (comp1)>Events (ev)> Indicator States: Stress, Symmetric**, and **Component I (comp1)>Events (ev)> Indicator States: Stress, Nonsymmetric**.
- 5 Click **Disable**.

### *Solution 3 (sol3)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver I**.
- 3 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.
- 4 In the **Event tolerance** text field, type 0.001.
- 5 Locate the **Output** section. From the **Times to store** list, choose **Steps taken by solver**.
- 6 Clear the **Store time derivatives** check box.
- 7 In the **Model Builder** window, click **Study 3**.

- 8 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 9 Clear the **Generate default plots** check box.

### *Study 3*

On the **Study** toolbar, click **Get Initial Value**.

## **RESULTS**

### *Stress vs Strain, Prescribed Symmetric Strain I*

- 1 In the **Model Builder** window, right-click **Stress vs Strain, Prescribed Symmetric Strain** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type *Stress vs Strain, Prescribed Nonsymmetric Strain* in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 3/Solution 3 (sol3)**.

### *Stresses vs Time, Prescribed Symmetric Strain I*

- 1 In the **Model Builder** window, under **Results** right-click **Stresses vs Time, Prescribed Symmetric Strain** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type *Stresses vs Time, Prescribed Nonsymmetric Strain* in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 3/Solution 3 (sol3)**.

## **STUDY 3**

### *Step 1: Time Dependent*

- 1 In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Results While Solving** section.
- 3 Select the **Plot** check box.
- 4 From the **Plot group** list, choose **Stress vs Strain, Prescribed Nonsymmetric Strain**.
- 5 On the **Study** toolbar, click **Compute**.

## **RESULTS**

### *Stress vs Strain, Prescribed Nonsymmetric Strain*

Add a study to compute loading controlled by nonsymmetric stress cycles.

## **ADD STUDY**

- 1 On the **Study** 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 **Preset Studies>Time Dependent**.
- 4 Click **Add Study** in the window toolbar.
- 5 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

#### STUDY 4

##### *Step 1: Time Dependent*

- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 In the **Times** text field, type 0 30.
- 3 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component I (comp I)>Events (ev)>Indicator States: Strain, Symmetric, Component I (comp I)>Events (ev)>Indicator States: Stress, Symmetric**, and **Component I (comp I)>Events (ev)>Indicator States: Strain, Nonsymmetric**.
- 5 Click **Disable**.

##### *Solution 4 (sol4)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Time-Dependent Solver I**.
- 3 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.
- 4 In the **Event tolerance** text field, type 0.001.
- 5 Locate the **Output** section. From the **Times to store** list, choose **Steps taken by solver**.
- 6 Clear the **Store time derivatives** check box.
- 7 In the **Model Builder** window, click **Study 4**.
- 8 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 9 Clear the **Generate default plots** check box.

##### *Study 4*

On the **Study** toolbar, click **Get Initial Value**.

## RESULTS

### *Stress vs Strain, Prescribed Symmetric Strain I*

- 1 In the **Model Builder** window, under **Results** right-click **Stress vs Strain, Prescribed Symmetric Strain** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type **Stress vs Strain, Prescribed Nonsymmetric Stress** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 4/Solution 4 (sol4)**.

### *Stresses vs Time, Prescribed Symmetric Strain I*

- 1 In the **Model Builder** window, under **Results** right-click **Stresses vs Time, Prescribed Symmetric Strain** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type **Stresses vs Time, Prescribed Nonsymmetric Stress** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 4/Solution 4 (sol4)**.

## STUDY 4

### *Step 1: Time Dependent*

- 1 In the **Model Builder** window, under **Study 4** click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Results While Solving** section.
- 3 Select the **Plot** check box.
- 4 From the **Plot group** list, choose **Stress vs Strain, Prescribed Nonsymmetric Stress**.
- 5 On the **Study** toolbar, click **Compute**.

## RESULTS

### *Stress vs Strain, Prescribed Nonsymmetric Stress*

Plot the effective viscoplastic strain on the whole geometry

### *Revolution 2D I*

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.
- 2 In the **Settings** window for **Revolution 2D**, click to expand the **Advanced** section.
- 3 Select the **Define variables** check box.

## RESULTS

### *Mirror 3D I*

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

- 2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.
- 3 From the **Plane** list, choose **XY-planes**.

#### *3D Plot Group 9*

- 1 On the **Results** toolbar, click **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type **Effective Viscoplastic Strain** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Mirror 3D 1**.

#### *Surface 1*

- 1 Right-click **Effective Viscoplastic Strain** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>solid.evpeGp - Effective viscoplastic strain**.

#### *Surface 2*

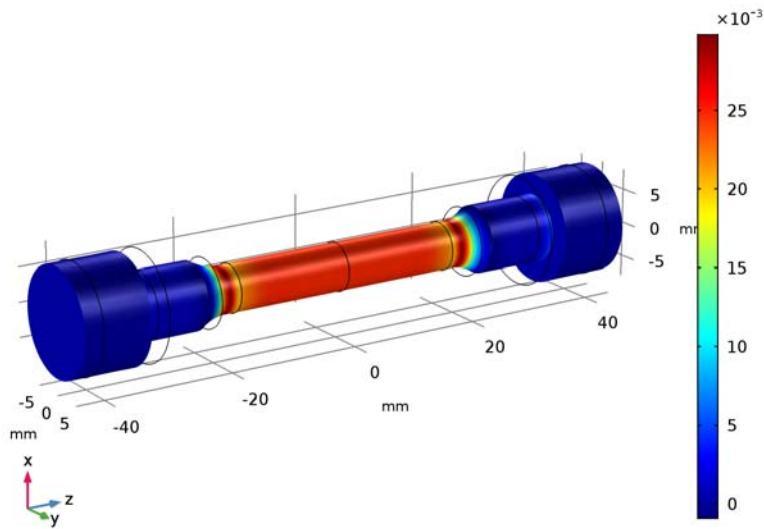
- 1 Right-click **Results>Effective Viscoplastic Strain>Surface 1** and choose **Deformation**.
- 2 In the **Model Builder** window, under **Results** right-click **Effective Viscoplastic Strain** and choose **Surface**.
- 3 In the **Settings** window for **Surface**, locate the **Expression** section.
- 4 In the **Expression** text field, type 0.
- 5 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 6 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

#### *Filter 1*

- 1 Right-click **Results>Effective Viscoplastic Strain>Surface 2** and choose **Deformation**.
- 2 Right-click **Surface 2** and choose **Filter**.
- 3 In the **Settings** window for **Filter**, locate the **Element Selection** section.

4 In the **Logical expression for inclusion** text field, type  $z \geq 31.5$  [mm].

Time=40 s Surface: Effective viscoplastic strain (1)



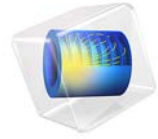
#### *Effective Viscoplastic Strain*

Add an animation to show the increase of effective viscoplastic strain over time.

#### *Animation 1*

- 1 On the **Results** toolbar, click **Animation** and choose **Player**.
- 2 In the **Settings** window for **Animation**, click **Show Frame**.
- 3 In the **Label** text field, type Effective Viscoplastic Strain.
- 4 Locate the **Scene** section. From the **Subject** list, choose **Effective Viscoplastic Strain**.
- 5 Locate the **Frames** section. In the **Number of frames** text field, type 50.
- 6 Right-click **Effective Viscoplastic Strain** and choose **Play**.





# Pressurized Orthotropic Container

## *Introduction*

---

A container made of rolled steel is subjected to an internal overpressure. As an effect of the manufacturing method, one of the three material principal directions—the out-of-plane direction—has a higher yield stress than the other two. Hill’s orthotropic plasticity is used to model the differences in yield strength. The model also shows how to define and use curvilinear coordinates aligned with the principal directions of the material, which in this case follow the contours of the container.

## *Model Definition*

---

The container has the shape of a cylinder capped by two torispherical heads (also called Klöpper head). The cylinder has an internal radius of  $R_1 = 24$  cm, a height of  $h = 80$  cm, and its thickness is  $t = 2$  cm. The torispherical head is made out of three parts: the crown, the knuckle and the flange. The crown has an internal radius of  $R_c = 43.2$  cm, the knuckle has an internal radius of  $R_k = 5.2$  cm, and the straight flange is  $s = 7$  cm in height, see [Figure 1](#).

Because of 2D axial symmetry and reflection symmetry, it is sufficient to model a quarter of the container; see [Figure 1](#). The red lines define the rotation symmetry axis and the reflection symmetry axis.

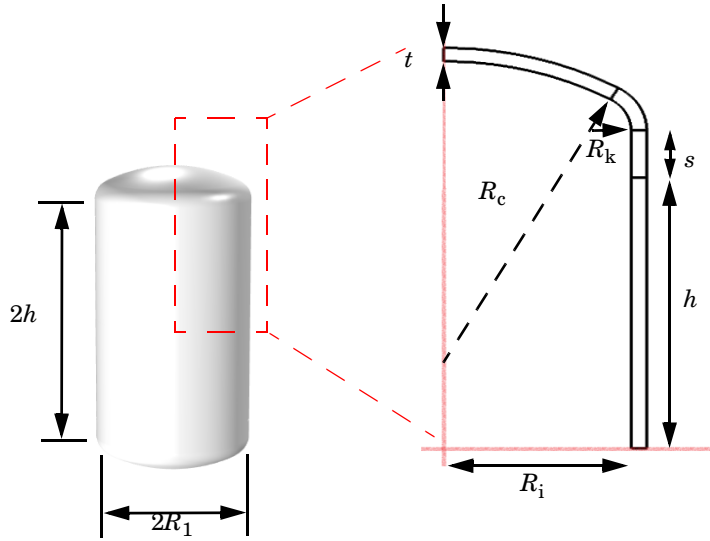


Figure 1: Schematic description of the container geometry and dimensions.

#### MATERIAL MODEL

The elastoplastic material is defined by a Young's modulus,  $E = 205$  GPa and a Poisson's ratio,  $\nu = 0.28$ . Hill's orthotropic plasticity governs the yielding, with the yield stress components given by

$$\begin{bmatrix} \sigma_{ys1} \\ \sigma_{ys2} \\ \sigma_{ys3} \\ \tau_{ys23} \\ \tau_{ys31} \\ \tau_{ys12} \end{bmatrix} = \begin{bmatrix} 381 \\ 381 \\ 450 \\ 240 \\ 240 \\ 220 \end{bmatrix} \text{MPa}$$

There is no hardening, so the material is perfectly plastic. The numbers in the subscripts denote the principal material directions, as indicated in the following section.

#### MATERIAL ORIENTATION

The rolled steel sheet has better mechanical properties in the out-of-plane direction, direction 3. To account for this anisotropy, use a special coordinate system that follows the component shape; see [Figure 2](#).

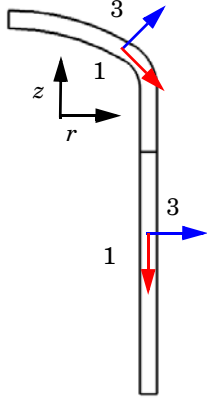


Figure 2: Orientation of local material coordinate system. The second principal direction is oriented in the circumferential direction, perpendicular to the  $rz$ -plane.

The curvilinear coordinates interface is used to compute an orthogonal system of coordinates able to follow the container's shape.

### Results and Discussion

An approximate analytical solution can be obtained for the cylindrical part of the container. The principal stresses in the center of the container can be estimated from the internal radius  $R_i$ , the wall thickness  $t$  and internal pressure  $p$ :

$$\begin{aligned}\sigma_1 &= p \frac{R_i}{2t} \\ \sigma_2 &= p \frac{R_i}{t} \\ \sigma_3 &= -p\end{aligned}\tag{1}$$

Following Hill's criterion, the yielding occurs when

$$F(\sigma_2 - \sigma_3)^2 + G(\sigma_3 - \sigma_1)^2 + H(\sigma_1 - \sigma_2)^2 = 1$$

or replacing by the expressions in [Equation 1](#)

$$p^2 \left[ F \left( \frac{R_i}{t} + 1 \right)^2 + G \left( 1 + \frac{R_i}{2t} \right)^2 + H \left( \frac{R_i}{2t} - \frac{R_i}{t} \right)^2 \right] = 1$$

The material parameters,  $F = G = 2.47 \cdot 10^{-18} \text{ 1/Pa}^2$  and  $H = 4.42 \cdot 10^{-18} \text{ 1/Pa}^2$ , give the analytical onset of yielding in the center of the cylinder at  $p = 37.8 \text{ MPa}$ .

Given the curvature of the knuckle, the material in the torispherical head undergoes plastic deformation below this onset pressure.

Figure 3 shows the von Mises stress at 10% yielded volume, which happens when the inner pressure reaches 31.4 MPa. For isotropic steel with yield stress of 381 MPa, the 10% yielded volume is reached when  $p = 29 \text{ MPa}$ . Therefore, with orthotropic steel, the pressure needed to reach 10% yielded volume is about 8% higher than when using isotropic steel.

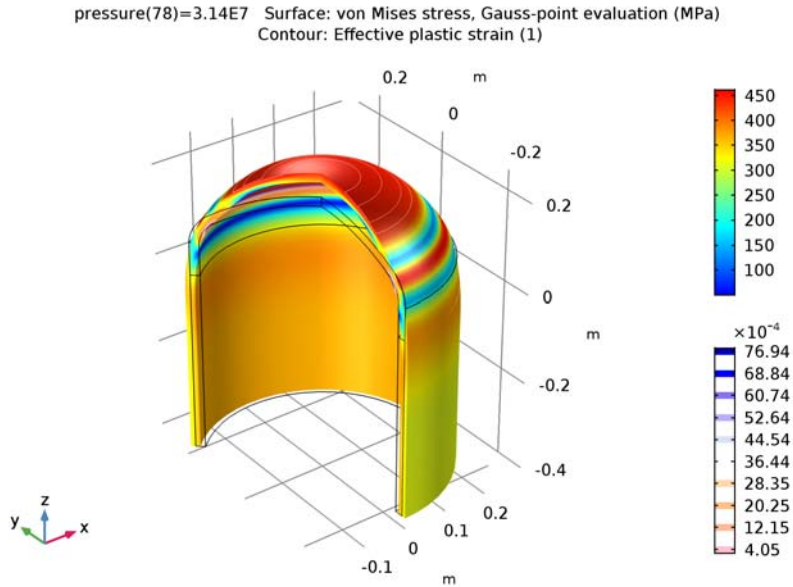


Figure 3: Effective stress at the onset of yielding.

### Notes About the COMSOL Implementation

Hill orthotropic plasticity is available in COMSOL as a built-in option under the Plasticity feature, where either Hill's coefficients or initial yield stresses can be given. The yield strength values can also be specified in the material node.

A coordinate system that follows the geometrical shape is created by the Curvilinear Coordinates interface. For axisymmetric geometries, the new base vectors,  $x_1$  and  $x_3$ , are computed for the geometry. In a case of geometric nonlinearity, the coordinates  $R$  and  $Z$

define the positions with respect to the initial configuration (*material frame*) whereas  $r$  and  $z$  define the positions with respect to the deformed configuration (*spatial frame*). Generally, material properties are defined in terms of the initial configuration. To assign the new base vector system to the component, select **Curvilinear System** from the **Coordinate system** list in the **Settings** window for **Linear Elastic Material**.

Figure 4 visualizes the base vector system defined by the curvilinear system using Arrow Surface plots. The red arrows denote the direction 1, while the blue arrows denote direction 3. The out-of-plane direction is used as direction 2 (not plotted)

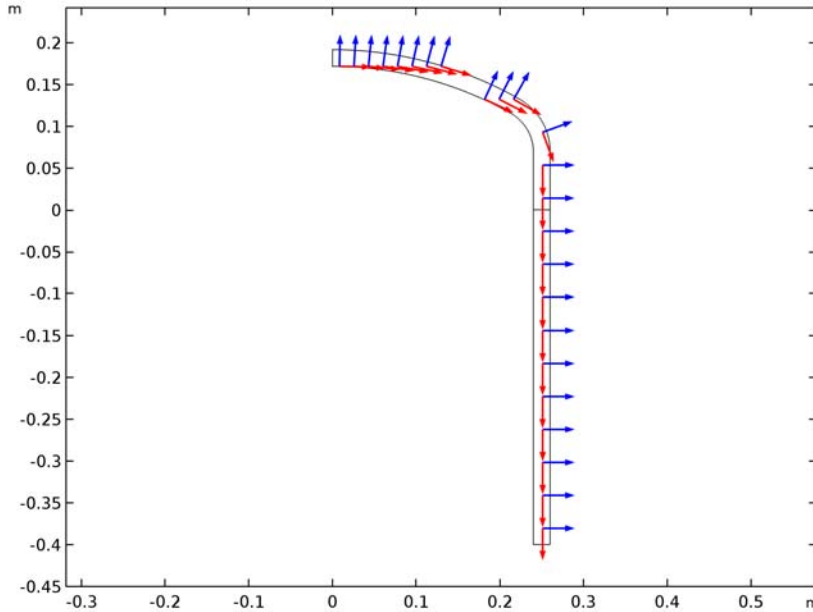


Figure 4: Orientation of the curvilinear system.

A stop condition is added to the parametric solver, so that the simulation stops when 10% of the material has exceeded the yield limit. Unless you are performing a failure analysis, it is not necessary to compute the whole plastic history, and the stop condition saves much computation time from being spent in the strongly nonlinear regime.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Plasticity/orthotropic\_container

---

## *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 Axisymmetric**.
- 2 In the **Select Physics** tree, select **Mathematics>Curvilinear Coordinates (cc)**.
- 3 Click **Add**.
- 4 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 5 Click **Add**.
- 6 Click **Study**.
- 7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
- 8 Click **Done**.

### **GLOBAL DEFINITIONS**

Load the parameters from file.

#### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `orthotropic_container_parameters.txt`.

### **DEFINITIONS**

#### *Integration 1 (intop1)*

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Selection** list, choose **All domains**.
- 4 Locate the **Advanced** section. From the **Frame** list, choose **Material (R, PHI, Z)**.

#### *Variables 1*

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
y_vol	$\text{intop1}(\text{solid.epeGp}>0)/\text{intop1}(1)$		Yielded volume fraction

## GEOMETRY I

### Circle 1 (c1)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type Rc.
- 4 In the **Sector angle** text field, type alpha.
- 5 Locate the **Position** section. In the **z** text field, type sf - (Rc-hi).
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type 90-alpha.

### Circle 2 (c2)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type Rc+th.
- 4 In the **Sector angle** text field, type alpha.
- 5 Locate the **Position** section. In the **z** text field, type sf - (Rc-hi).
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type 90-alpha.

### Difference 1 (dif1)

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 In the **Settings** window for **Difference**, type Crown in the **Label** text field.
- 3 Select the object **c2** only.
- 4 Locate the **Difference** section. Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 5 Select the object **c1** only.

### Circle 3 (c3)

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type Rk+th.

- 4 In the **Sector angle** text field, type 90-alpha.
- 5 Locate the **Position** section. In the **r** text field, type  $R_i - R_k$ .
- 6 In the **z** text field, type  $sf$ .
- 7 Right-click **Circle 3 (c3)** and choose **Build Selected**.

*Circle 4 (c4)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type  $R_k$ .
- 4 In the **Sector angle** text field, type 90-alpha.
- 5 Locate the **Position** section. In the **r** text field, type  $R_i - R_k$ .
- 6 In the **z** text field, type  $sf$ .
- 7 Right-click **Circle 4 (c4)** and choose **Build Selected**.

*Difference 2 (dif2)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **c3** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 5 Select the object **c4** only.
- 6 Right-click **Difference 2 (dif2)** and choose **Build Selected**.
- 7 In the **Model Builder** window, click **Difference 2 (dif2)**.
- 8 In the **Settings** window for **Difference**, type **Knuckle** in the **Label** text field.

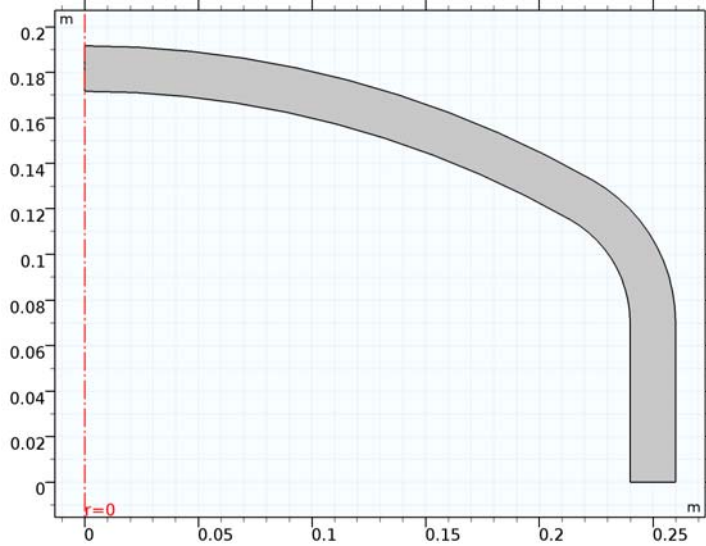
*Rectangle 1 (r1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, type **Flange** in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Width** text field, type  $th$ .
- 4 In the **Height** text field, type  $sf$ .
- 5 Locate the **Position** section. In the **r** text field, type  $R_i$ .
- 6 Click **Build All Objects**.

*Union 1 (uni1)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
- 2 In the **Settings** window for **Union**, type **Klopper Head** in the **Label** text field.

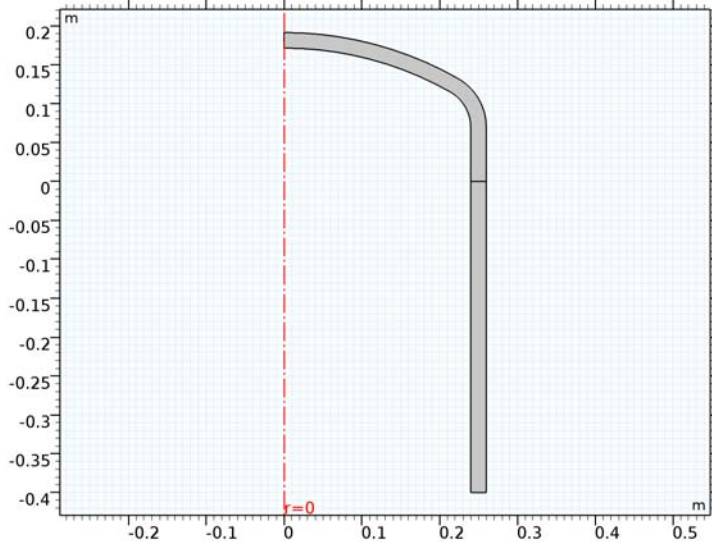
- 3 Click in the **Graphics** window and then press Ctrl+A to select all objects.
- 4 Locate the **Union** section. Clear the **Keep interior boundaries** check box.
- 5 Click **Build All Objects**.



*Rectangle 2 (r2)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, type Cylinder in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Width** text field, type th.
- 4 In the **Height** text field, type hcyl.
- 5 Locate the **Position** section. In the **r** text field, type Ri.
- 6 In the **z** text field, type -hcyl.

7 Click **Build All Objects**.



### **CURVILINEAR COORDINATES (CC)**

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Curvilinear Coordinates (cc)**.
- 2 In the **Settings** window for **Curvilinear Coordinates**, locate the **Settings** section.
- 3 Select the **Create base vector system** check box.

#### *Diffusion Method 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **Diffusion Method**.
- 2 In the **Settings** window for **Diffusion Method**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **All domains**.

#### *Inlet 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Inlet**.
- 2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
- 3 Click **Clear Selection**.
- 4 Select Boundary 1 only.

#### *Diffusion Method 1*

In the **Model Builder** window, under **Component 1 (comp1)**>**Curvilinear Coordinates (cc)** click **Diffusion Method 1**.

#### *Outlet 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Outlet**.
- 2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.
- 3 Click **Clear Selection**.
- 4 Select Boundary 3 only.

#### **SOLID MECHANICS (SOLID)**

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

#### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 Select Boundary 3 only.

#### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 In the **Settings** window for **Boundary Load**, type Boundary Load Pressure in the **Label** text field.
- 3 Select Boundaries 2, 4, 8, and 10 only.
- 4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.
- 5 In the  $p$  text field, type pressure.

#### *Linear Elastic Material 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Solid Mechanics (solid)** click **Linear Elastic Material 1**.
- 2 In the **Settings** window for **Linear Elastic Material**, locate the **Coordinate System Selection** section.
- 3 From the **Coordinate system** list, choose **Curvilinear System (cc) (cc\_cs)**.

#### *Plasticity 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Plasticity**.
- 2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.
- 3 From the **Yield function F** list, choose **Hill orthotropic plasticity**.
- 4 Find the **Isotropic hardening model** subsection. From the list, choose **Perfectly plastic**.
- 5 Find the **Kinematic hardening model** subsection. From the list, choose **No kinematic hardening**.

## ADD MATERIAL

- 1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Built-In>Steel AISI 4340**.
- 4 Click **Add to Component** in the window toolbar.

## MATERIALS

### *Steel AISI 4340 (mat1)*

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 From the **Selection** list, choose **All domains**.
- 3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Initial tensile and shear yield stresses	ys	{381e6, 381e6, 450e6, 240e6, 240e6, 220e6}	N/m <sup>2</sup>	Elastoplastic material model

- 4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

## MESH 1

### *Distribution 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
- 2 Right-click **Mapped 1** and choose **Distribution**.
- 3 Select Boundary 1 only.

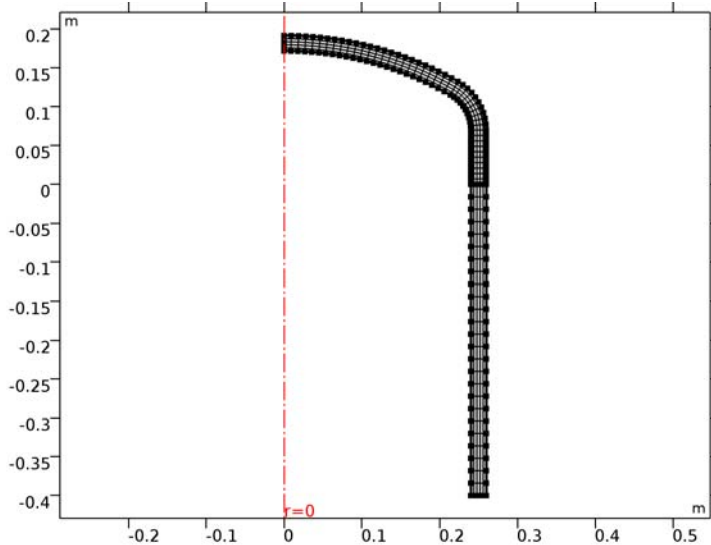
### *Distribution 2*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundaries 4, 7, 10, and 11 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution properties** list, choose **Fixed number of elements**.
- 5 In the **Number of elements** text field, type 10.

### *Distribution 3*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

- 3 From the **Distribution properties** list, choose **Fixed number of elements**.
- 4 In the **Number of elements** text field, type 25.
- 5 Select Boundaries 2, 6, 8, and 9 only.
- 6 Click **Build All**.



The mesh should consist of 350 quadrilateral elements, with 5 element through thickness. Finer elements are created at the knuckle since stress gradients are expected there.

## STUDY 1

### *Step 1: Stationary*

- 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 In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

### *Step 2: Stationary 2*

- 1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 In the table, clear the **Solve for** check box for the **Curvilinear Coordinates (cc)** interface.  
Set up an auxiliary continuation sweep for the **pressure** parameter.

- 4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 5 Click **Add**.
- 6 In the table, enter the following settings:

Parameter name	Parameter value list
pressure (Internal pressure)	range (16e6, 2e5, 36e6)

*Solution 1 (sol1)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.  
Introduce a stop condition to stop the solver.
- 3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** node.
- 4 Right-click **Parametric 1** and choose **Stop Condition**.
- 5 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.
- 6 Click **Add**.
- 7 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
0.1-comp1.y_vol	Negative (<0)	√	Stop expression 1

Specify that the solution is to be stored both before and after the stop condition is reached.

- 8 Locate the **Output at Stop** section. From the **Add solution** list, choose **Steps before and after stop**.
- 9 On the **Study** toolbar, click **Compute**.
- 10 Click the **Zoom Extents** button on the **Graphics** toolbar.

## RESULTS

*Coordinate system (cc)*

- 1 In the **Model Builder** window, under **Results** click **Coordinate system (cc)**.
- 2 In the **Settings** window for **2D Plot Group**, type Material Principal Direction in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

### 2D Plot Group 5

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type Plastic Strain 2D in the **Label** text field.

### Surface 1

- 1 Right-click **Plastic Strain 2D** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Effective plastic strain**.
- 3 On the **Plastic Strain 2D** toolbar, click **Plot**.  
The onset of plasticity can be investigated by evaluating the volume of the material which has exceeded the yield stress.

### 1D Plot Group 6

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
- 2 In the **Settings** window for **1D Plot Group**, type Yielded Volume in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.
- 4 Find the **Type and data** subsection. Clear the **Type** check box.
- 5 Clear the **Unit** check box.

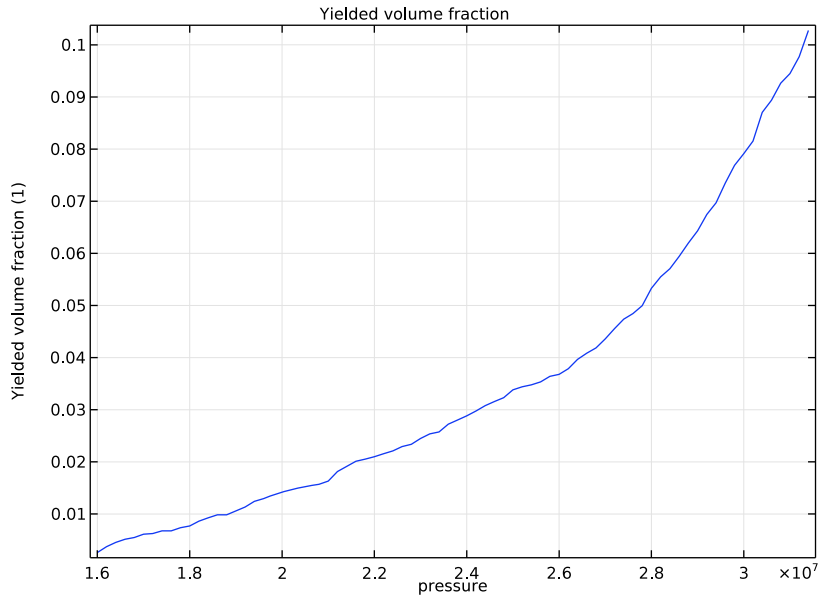
### Global 1

- 1 Right-click **Yielded Volume** and choose **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
y_vol	1	Yielded volume fraction

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

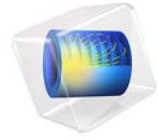
5 On the **Yielded Volume** toolbar, click **Plot**.



#### *Surface 1*

- 1 In the **Model Builder** window, expand the **Results>Stress, 3D (solid)** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.
- 4 On the **Stress, 3D (solid)** toolbar, click **Plot**.





# Polynomial Hyperelastic Model

## Introduction

---

This example shows how you can implement a user defined hyperelastic material, using the strain density energy function. The model used is a general Mooney-Rivlin hyperelastic material model defined by a polynomial.

For such a material model, the strain energy density function has the following expression:

$$W = \sum_{i,j=0}^n C_{i,j} (\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + \frac{1}{2} K (J_{e1} - 1)^2$$

Here  $\bar{I}_1$  and  $\bar{I}_2$  are the first and second invariant of the left isochoric Cauchy-Green deformation tensor,  $J_{e1}$  is the elastic Jacobian,  $C_{ij}$  are coefficients in the polynomial, and  $K$  is the bulk modulus.

In this example, you see two material models based on the above expression: a two-term equation and a five-term equation. The two-term Mooney-Rivlin material model implementation is then validated with the results obtained with the predefined Mooney-Rivlin hyperelastic material.

## Model Definition

---

A simple geometry is used consisting of a single block of the hyperelastic material as shown in [Figure 1](#). The block is fixed at one face and loaded with an uniform normal load of 1 MPa at the opposite face. Due to symmetry, only one quarter of the geometry is represented.

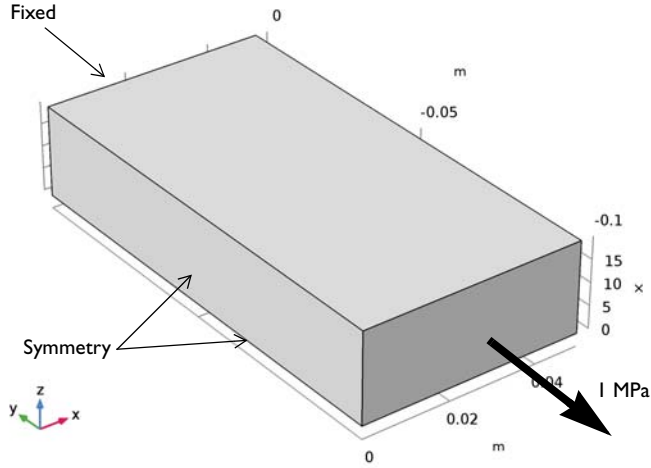


Figure 1: Model geometry with boundary conditions and loads.

The two-term Mooney-Rivlin material model is defined by the following strain energy density:

$$W_{\text{siso}} = C_{1,0}(\bar{I}_1 - 3) + C_{0,1}(\bar{I}_2 - 3)$$

$$W_{\text{svol}} = \frac{1}{2}\kappa(J_{\text{el}} - 1)^2$$

The five-term Mooney-Rivlin material model is defined by the following strain energy density:

$$W_{\text{siso}} = \begin{cases} C_{1,0}(\bar{I}_1 - 3) + C_{0,1}(\bar{I}_2 - 3) + C_{2,0}(\bar{I}_1 - 3)^2 + \\ C_{0,2}(\bar{I}_2 - 3)^2 + C_{1,1}(\bar{I}_1 - 3)(\bar{I}_2 - 3) \end{cases}$$

$$W_{\text{svol}} = \frac{1}{2}\kappa(J_{\text{el}} - 1)^2$$

---

**Note:** Both the two-term and the five-term Mooney-Rivlin material model are available in the Hyperelastic Model feature node.

---

## Results and Discussion

Figure 2 shows the  $y$ -component of the second Piola-Kirchhoff stress along the center axis through the block. You can see that the results from the two-term polynomial equation model perfectly matches the results of the built-in Mooney-Rivlin material.

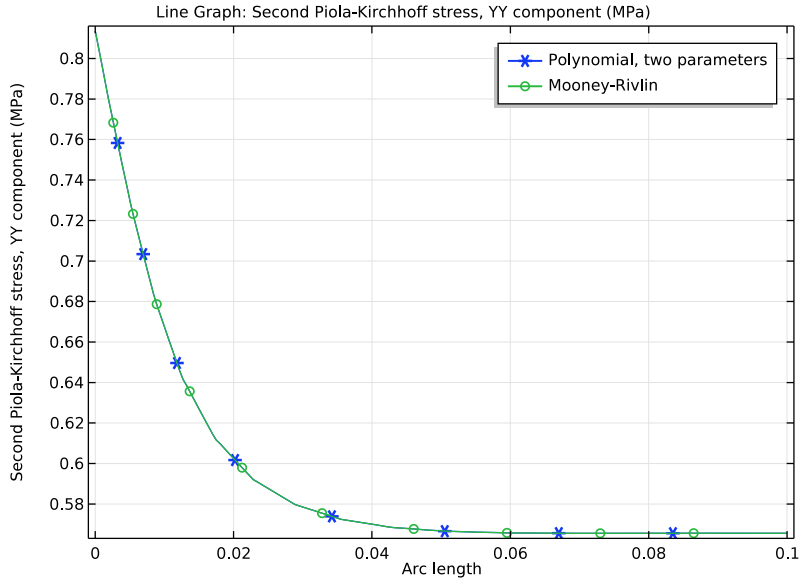


Figure 2: Stress plot ( $\mathcal{Y}$ -component of the second Piola-Kirchhoff stress) along the geometry length.

Figure 3 shows the von Mises stress distribution in the geometry with the two-term Mooney-Rivlin material.

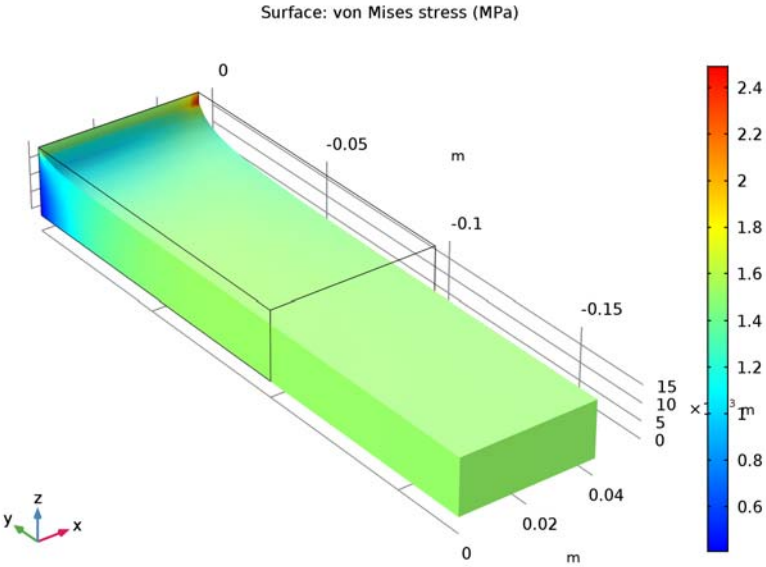
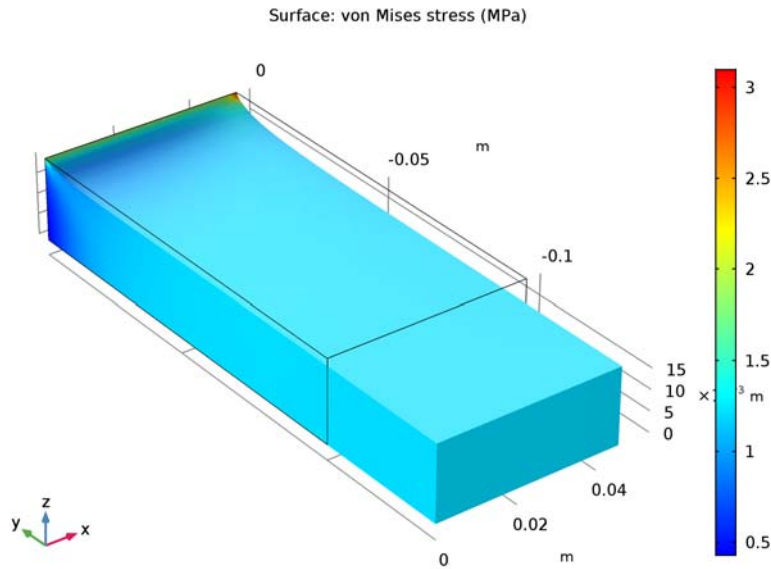


Figure 3: von Mises stress distribution for the two-term polynomial hyperelastic material model.

Figure 4 shows the von Mises stress distribution in the geometry with the five-term Mooney-Rivlin material model. Note the difference in deformation: the five-term polynomial model has a significantly smaller deformation than the two-term model.



*Figure 4: von Mises stress distribution for the five-term polynomial hyperelastic material model.*

In the second case, you can see that the stress in the region far away from the fixed end is significantly lower than in the first case. The reason is that the total load is the same in both cases, but the area reduction is much larger with the more flexible two-term material. The von Mises effective stress is computed from the Cauchy stress, which is based on force per current area.

### *Notes About the COMSOL Implementation*

Instead of using the predefined hyperelastic material model, you manually define the material in the hyperelastic material model feature node's **Settings** window. In the **Hyperelastic Material Model** section, select **User defined** from the **Material model** list.

For nearly incompressible materials, the strain energy density is defined using a separation of the isochoric strain energy density and the volumetric strain energy density.

When you use a hyperelastic material in your model, all studies automatically become geometrically nonlinear.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Hyperelasticity/polynomial\_hyperelastic

---

### *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 **3D**.
- 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 **Preset Studies>Stationary**.
- 6 Click **Done**.

#### **GLOBAL DEFINITIONS**

##### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

<b>Name</b>	<b>Expression</b>	<b>Value</b>	<b>Description</b>
kappa	3[MPa]	3E6 Pa	Bulk modulus
C01	0.5[MPa]	5E5 Pa	Polynomial coefficient C01
C10	0.1[MPa]	1E5 Pa	Polynomial coefficient C10
C11	0.15[MPa]	1.5E5 Pa	Polynomial coefficient C11
C20	0.2[MPa]	2E5 Pa	Polynomial coefficient C20
C02	0.4[MPa]	4E5 Pa	Polynomial coefficient C02

## GEOMETRY I

### Block 1 (blk1)

- 1 On the **Geometry** toolbar, click **Block**.
- 2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 0.1.
- 4 In the **Depth** text field, type 0.05.
- 5 In the **Height** text field, type 0.02.
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.
- 7 Click **Build All Objects**.
- 8 Click the **Zoom Extents** button on the **Graphics** toolbar.

## DEFINITIONS

### Variables 1

- 1 On the **Home** toolbar, click **Variables** and choose **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
Wsiso_MR2	$C10*(\text{solid.I1CIe1-3})+C01*(\text{solid.I2CIe1-3})$		Isochoric strain energy density, Mooney-Rivlin two parameters
Wsiso_MR5	$Wsiso\_MR2+C20*(\text{solid.I1CIe1-3})^2+C02*(\text{solid.I2CIe1-3})^2+C11*(\text{solid.I1CIe1-3})*(\text{solid.I2CIe1-3})$		Isochoric strain energy density, Mooney-Rivlin five parameters
Wsvol	$0.5*kappa*(\text{solid.Je1-1})^2$	Pa	Volumetric strain energy density

## SOLID MECHANICS (SOLID)

### Fixed Constraint 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
- 2 Select Boundary 5 only.

### Symmetry 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

- 2 Select Boundaries 1 and 3 only.

#### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 Specify the  $\mathbf{F}_A$  vector as

0	x
-1 [MPa]	y
0	z

### **TWO-PARAMETER POLYNOMIAL HYPERELASTIC MATERIAL MODEL**

#### *Hyperelastic Material 1*

- 1 On the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Polynomial, Two Parameters in the **Label** text field.
- 3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.
- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **User defined**.
- 5 Select the **Nearly incompressible material** check box.
- 6 In the  $W_{\text{siso}}$  text field, type  $W_{\text{siso\_MR2}}$ .
- 7 In the  $W_{\text{svol}}$  text field, type  $W_{\text{svol1}}$ .

### **MESH 1**

#### *Mapped 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.
- 2 Select Boundary 5 only.

#### *Distribution 1*

- 1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.
- 2 Select Edges 6 and 12 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution properties** list, choose **Predefined distribution type**.

- 5 In the **Number of elements** text field, type 4.
- 6 In the **Element ratio** text field, type 5.
- 7 Select the **Reverse direction** check box.

#### *Distribution 2*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Edges 7 and 8 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution properties** list, choose **Predefined distribution type**.
- 5 In the **Number of elements** text field, type 6.
- 6 In the **Element ratio** text field, type 5.

#### *Distribution 1*

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.
- 2 Right-click **Swept 1** and choose **Distribution**.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 From the **Distribution properties** list, choose **Predefined distribution type**.
- 5 In the **Number of elements** text field, type 15.
- 6 In the **Element ratio** text field, type 5.
- 7 Click **Build All**.

## **MOONEY-RIVLIN HYPERELASTIC MATERIAL MODEL**

### *Hyperelastic Material 2*

- 1 On the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Mooney-Rivlin in the **Label** text field.
- 3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.
- 4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney-Rivlin, two parameters**.
- 5 From the  $C_{10}$  list, choose **User defined**. In the associated text field, type C10.
- 6 From the  $C_{01}$  list, choose **User defined**. In the associated text field, type C01.
- 7 In the  $\kappa$  text field, type kappa.

### *Polynomial, Two Parameters 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Polynomial, Two Parameters** and choose **Duplicate**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Polynomial, five parameters in the **Label** text field.
- 3 Locate the **Hyperelastic Material** section. In the  $W_{\text{iso}}$  text field, type  $W_{\text{iso\_MR5}}$ .  
The Mooney-Rivlin, five parameters is also available as predefined hyperelastic material. Solve the Polynomial Two Parameters model first.

### **STUDY 1**

- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Settings** window for **Study**, type Study: Polynomial, Two Parameters in the **Label** text field.

### **STUDY: POLYNOMIAL, TWO PARAMETERS**

#### *Step 1: Stationary*

- 1 In the **Model Builder** window, expand the **Study: Polynomial, Two Parameters** node, then 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>Mooney-Rivlin** and **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Polynomial, five parameters**.
- 5 Click **Disable**.
- 6 On the **Home** toolbar, click **Compute**.

### **RESULTS**

#### *Stress (solid)*

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **3D Plot Group**, type Stress, Polynomial Two Parameters in the **Label** text field.

#### *Surface 1*

- 1 In the **Model Builder** window, expand the **Results>Stress, Polynomial Two Parameters** node, then click **Surface 1**.

- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.
- 4 On the **Stress, Polynomial Two Parameters** toolbar, click **Plot**.
- 5 Click the **Go to Default View** button on the **Graphics** toolbar.

#### *Volume Maximum 1*

- 1 On the **Results** toolbar, click **More Derived Values** and choose **Maximum>Volume Maximum**.
- 2 In the **Settings** window for **Volume Maximum**, locate the **Selection** section.
- 3 From the **Selection** list, choose **All domains**.
- 4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>Displacement field (material and geometry frames)>v - Displacement field, Y component**.
- 5 Click to expand the **Advanced** section. From the **Find maximum of list**, choose **Absolute value**.
- 6 Click **Evaluate**.

Now solve the Mooney-Rivlin model.

#### **ADD STUDY**

- 1 On 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 **Preset Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.

#### **STUDY 2**

##### *Step 1: Stationary*

- 1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.
- 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>Polynomial, five parameters**.
- 5 Click **Disable**.
- 6 In the **Settings** window for **Study**, type Study: Mooney-Rivlin in the **Label** text field.
- 7 On the **Home** toolbar, click **Compute**.

## RESULTS

### *Stress (solid)*

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **3D Plot Group**, type **Stress, Mooney-Rivlin** in the **Label** text field.

### *Surface 1*

- 1 In the **Model Builder** window, expand the **Results>Stress, Mooney-Rivlin** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.
- 4 Click the **Go to Default View** button on the **Graphics** toolbar.

### *Volume Maximum 1*

- 1 In the **Model Builder** window, under **Results>Derived Values** click **Volume Maximum 1**.
- 2 In the **Settings** window for **Volume Maximum**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study: Mooney-Rivlin/Solution 2 (sol2)**.
- 4 Click **Evaluate**.

Then solve the Polynomial Five Parameters model.

## ADD STUDY

- 1 On 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 **Preset Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.
- 5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

## STUDY 3

- 1 In the **Settings** window for **Study**, type **Study: Polynomial, Five Parameters** in the **Label** text field.
- 2 On the **Home** toolbar, click **Compute**.

## RESULTS

### *Stress (solid)*

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

- 2 In the **Settings** window for **3D Plot Group**, type **Stress, Polynomial Five Parameters** in the **Label** text field.

#### *Surface 1*

- 1 In the **Model Builder** window, expand the **Results>Stress, Polynomial Five Parameters** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **MPa**.
- 4 On the **Stress, Polynomial Five Parameters** toolbar, click **Plot**.
- 5 Click the **Go to Default View** button on the **Graphics** toolbar.

#### *Volume Maximum 1*

- 1 In the **Model Builder** window, under **Results>Derived Values** click **Volume Maximum 1**.
- 2 In the **Settings** window for **Volume Maximum**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study: Polynomial, Five Parameters/Solution 3 (sol3)**.
- 4 Click **Evaluate**.

To compare Polynomial Two Parameters and Mooney-Rivlin results, reproduce [Figure 2](#).

#### *ID Plot Group 4*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Second Piola-Kirchhoff Stress, Y component** in the **Label** text field.
- 3 On the **Second Piola-Kirchhoff Stress, Y component** toolbar, click **Line Graph**.

#### *Line Graph 1*

- 1 In the **Model Builder** window, under **Results>Second Piola-Kirchhoff Stress, Y component** click **Line Graph 1**.
- 2 Select Edge 2 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>Solid Mechanics>Stress>Second Piola-Kirchhoff stress (material and geometry frames)>solid.SYY - Second Piola-Kirchhoff stress, YY component**.
- 4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.
- 5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 6 Click to expand the **Legends** section. Select the **Show legends** check box.

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

8 In the table, enter the following settings:

---

<b>Legends</b>
Polynomial, two parameters

---

*Line Graph 2*

1 Right-click **Results>Second Piola-Kirchhoff Stress, Y component>Line Graph 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study: Mooney-Rivlin/Solution 2 (sol2)**.

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type 10.

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

---

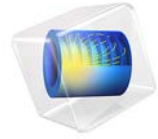
<b>Legends</b>
Mooney-Rivlin

---

6 Click to expand the **Title** section. From the **Title type** list, choose **None**.

7 On the **Second Piola-Kirchhoff Stress, Y component** toolbar, click **Plot**.





# Temperature-Dependent Plasticity in Pressure Vessel

## Introduction

---

This example demonstrates how to use temperature dependent materials within the Nonlinear Structural Materials Module. Material data such as Young's modulus, yield stress and strain hardening have strong temperature dependencies.

A large container holds pressurized hot water. Several pipes are attached to the pressure vessel. Those pipes can rapidly transfer cold water in case of an emergency cooling. The pressure vessel is made of carbon steel with an internal cladding of stainless steel. In case of a fast temperature transient, the differences in thermal expansion properties between the materials causes high stresses.

## Model Definition

---

### GEOMETRY

The pressure vessel has the shape of a closed cylinder. Four pipes are attached at two levels along its height. At each level, the pipes are equidistantly spaced around the container.

The pipes are welded into the vessel and the welding can be considered as a chamfer between those two parts. The structure, together with its key dimensions, is presented in [Figure 1](#).

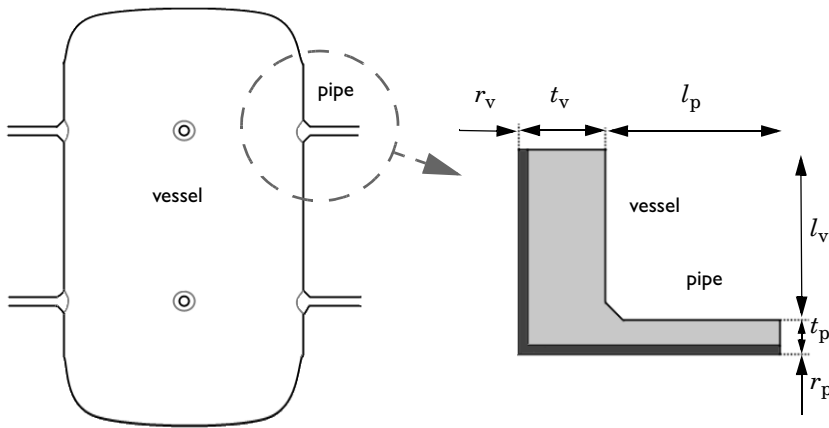


Figure 1: Pressure vessel and the dimensions for the vessel-pipe connection.

The structure has the following dimensions:

- Inner vessel radius,  $r_v = 1000$  mm,
- Inner pipe radius,  $r_p = 60$  mm,

- Vessel thickness,  $t_v = 100$  mm,
- Pipe thickness,  $t_p = 40$  mm.

The vessel length,  $l_v$ , and the pipe length,  $l_p$ , are large compared to the thickness of both parts. For modeling purposes, they need to be large enough so that local effects at the vessel-pipe connection can be disregarded. The chamfer extends 20 mm from the corner at the connection between the pipe and the vessel.

The dual material consists of a thin 10 mm layer of stainless steel (dark gray in [Figure 1](#)) that faces the water, and carbon steel (light gray in [Figure 1](#)) that faces the outside air.

In order to save computational time, only the connection between one pipe and the vessel is modeled, as shown on the left image of [Figure 1](#).

### MATERIAL MODEL

The thermoelastic material data of stainless steel is given in [Table 1](#). [Table 2](#) shows the yield stress as a function of plastic strain at temperatures of 20, 100, 200, and 300 °C.

TABLE 1: THERMOELASTIC MATERIAL DATA OF STAINLESS STEEL.

Temperature (°C)	20	100	200	300
E (GPa)	194	189	179	175
$\alpha$ (1/°C)	$16 \cdot 10^{-6}$	$16.5 \cdot 10^{-6}$	$17 \cdot 10^{-6}$	$17.5 \cdot 10^{-6}$
$c_p$ (J/(kg·K))	482	498	515	524
k (W/(m·K))	13.9	14.9	17.0	18.0

TABLE 2: TEMPERATURE DEPENDENT YIELD STRESS OF STAINLESS STEEL

Temperature (°C)	20	100	200	300
$\sigma_{ys}(0.0)$ (MPa)	228	190	156	140
$\sigma_{ys}(0.0004)$ (MPa)	232	195	160	144
$\sigma_{ys}(0.001)$ (MPa)	238	201	166	148
$\sigma_{ys}(0.002)$ (MPa)	246	210	173	155
$\sigma_{ys}(0.004)$ (MPa)	250	215	177	158
$\sigma_{ys}(0.001)$ (MPa)	263	230	189	169

The yield stress of the carbon steel is two or three times higher than that of stainless steel. It is therefore considered as elastic. Its material properties are shown in [Table 3](#).

TABLE 3: THERMOELASTIC MATERIAL DATA OF CARBON STEEL

Temperature (°C)	20	100	200	300
E (GPa)	208	202	196	189
$\alpha$ (1/°C)	$10.910^{-6}$	$12.410^{-6}$	$13.810^{-6}$	$14.910^{-6}$
$c_p$ (J/(kg·K))	489	519	546	569
k (W/(m·K))	51.2	48.3	45.5	42.7

The heat transfer coefficient between steel and air is  $10 \text{ W}/(\text{m}^2 \cdot \text{K})$ , and between steel and water is  $100 \text{ W}/(\text{m}^2 \cdot \text{K})$ .

### BOUNDARY CONDITIONS

A pressure of 70 bar acts on the vessel and pipe's inner walls. The temperature on the inside of the pressure vessel is initially at  $280 \text{ }^\circ\text{C}$ , while the outside air remains at  $50 \text{ }^\circ\text{C}$ . Suddenly and instantaneously, cold water at  $20 \text{ }^\circ\text{C}$  is pumped through the pipe into the vessel, where the hot water needs 30 minutes to cool down to  $20 \text{ }^\circ\text{C}$ . The cooling speed is constant.

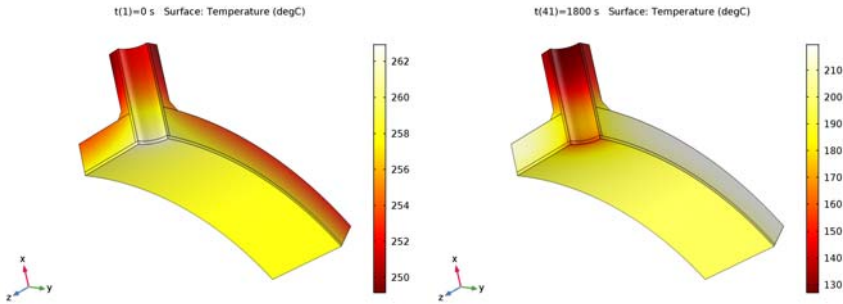
### MODEL ASSUMPTIONS

Due to symmetries only a  $45^\circ$  sector of the vessel is modeled. The influence of the hot water pressure at the end of the vessel is approximated with an axial stress of  $33.3 \text{ MPa}$ , which is 4.76 times the inner pressure. The parameters  $l_v$  and  $l_p$  are both set to  $200 \text{ mm}$ .

### *Results and Discussion*

Three studies are performed in this analysis. In an initial step, the mechanical and thermal stationary state is computed. This serves as initial conditions for a transient step which solves the heat transfer problem only, where the cold water flows through the pipe, cooling

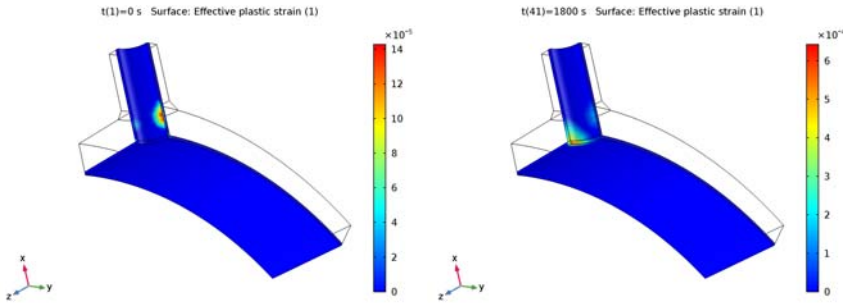
the initially hot water in the vessel. A comparison of the temperature profiles, before and after the event, are shown in [Figure 2](#).



*Figure 2: Temperature profiles before and after the cooling.*

After 30 min, the water inside the vessel has cooled down to 20 °C, but the container is still locally more than 100 °C hotter. There are large gradients in the thermal strains.

The last step solves the elastoplastic deformation with temperature-dependent material parameters. The development of plastic strains in the stainless steel layer is demonstrated in [Figure 3](#)



*Figure 3: Plasticity before and after the cooling.*

From [Figure 3](#), it can be seen that a plastic zone develops as the vessel cools down. Initially, when the vessel is at steady state, some plastic strain is generated by stresses caused by differences in the thermal expansion of the two steels. In a real structure, such stresses would have been relaxed after the first service cycle. In the transient study, when the vessel cools down and the yield limit increases, the pipe deforms plastically in other locations.

As the temperature decreases, the yield stress increases, so hot parts are more sensitive to high stresses. Figure 4 shows the effective stress after 30 min of cooling.

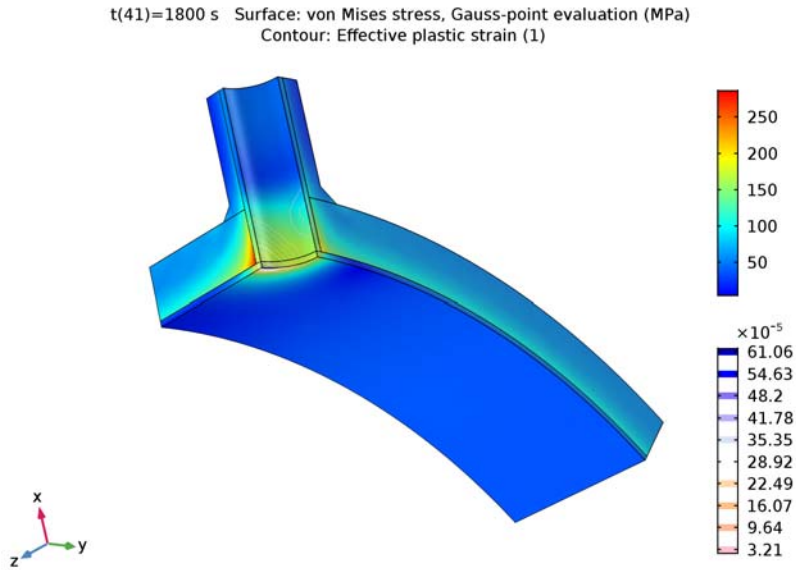


Figure 4: Effective stress after cooling down the hot water.

### Notes About the COMSOL Implementation

COMSOL can handle material data depending on several parameters. Use Interpolation functions, which you can select from the Definitions or Materials nodes. You can type in the data in a table or define your function in a text file. Use the symbol % in a text file to include comments or headers.

In this example, the Young's modulus  $E(T)$ , the coefficient of thermal expansion  $\alpha(T)$ , thermal conductivity  $k(T)$ , and the heat capacity at constant pressure  $C_p(T)$  are defined in the Materials node from interpolated data. The initial yield stress  $\sigma_{y0}(T)$  and the nonlinear hardening function  $\sigma_h(T, \epsilon_{pe})$  are defined by data imported from a text file.

Interpolation functions can handle any number of arguments. For convenience, specify units (Pa, m, s, and so on) for the function and arguments. When applicable, COMSOL Multiphysics automatically scales any input into the correct unit. For more details see [Operators, Functions, and Constants](#) in the *COMSOL Multiphysics Reference Manual*.

In this example there are two cuts which are “almost symmetry cuts” in the sense that they should stay plane, but are still allowed to move in the normal direction. One is the cut in the pipe, and the other the cut through the pressure vessel wall. In order to accomplish this, the displacement in the normal direction is prescribed for the whole boundary, but not to a constant value. It is prescribed to be the displacement in one point on the boundary. Thus it is constant, but yet determined by the solution.

Heat transfer in solids is a time-dependent phenomenon, and since plasticity is a path dependent process, it is important to capture the evolution of temperature profiles accurately. You need to limit the time step in the BDF solver settings to account for the rate at which thermal loads change in time. Use a different study step to compute the elastoplastic deformation after computing the heat transfer. Since this is a one-way coupled problem, this segregated approach reduces computational time.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Plasticity/temperature\_dependent\_plasticity

---

### *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 **3D**.
- 2** In the **Select Physics** tree, select **Structural Mechanics>Thermal Stress**.
- 3** Click **Add**.
- 4** Click **Study**.
- 5** In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
- 6** Click **Done**.

#### **GLOBAL DEFINITIONS**

##### *Parameters*

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

- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
internalPressure	70[bar]	7E6 Pa	Internal pressure
t	0[s]	0 s	Time variable; used for stationary analysis

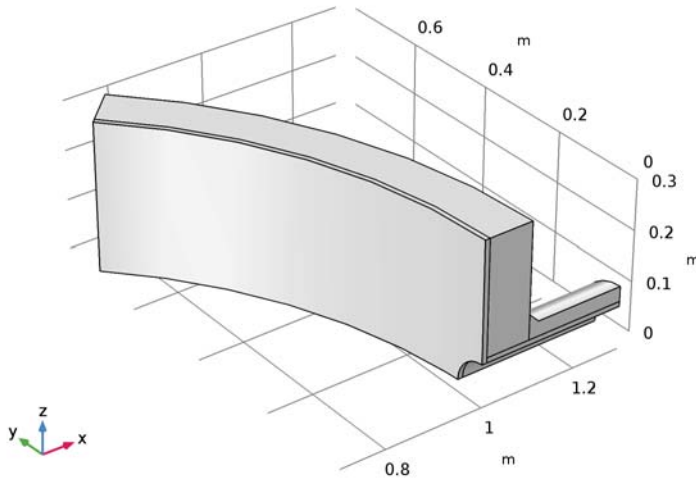
The thermal shock caused by the cold water is time dependent. Therefore the variable for time,  $t$ , needs to be set to zero so that the heat boundary conditions can be evaluated also in the static analysis.

### GEOMETRY I

- 1 On the **Geometry** toolbar, click **Insert Sequence**.
- 2 Browse to the model's Application Libraries folder and double-click the file `temperature_dependent_plasticity_geom_sequence.mph`.

*Work Plane 1 (wp1)*

- 1 On the **Geometry** toolbar, click **Build All**.
- 2 Click the **Go to Default View** button on the **Graphics** toolbar.



Full geometry instructions can be found in [Appendix — Geometry Modeling Instructions](#).

## SOLID MECHANICS (SOLID)

### *Linear Elastic Material 1*

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

### *Plasticity 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Plasticity**.
- 2 Select Domains 1 and 3 only.
- 3 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.
- 4 Find the **Isotropic hardening model** subsection. From the list, choose **User defined**.

## 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**, type **Stainless Steel** in the **Label** text field.
- 3 Select Domains 1 and 3 only.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Poisson's ratio	nu	0.3	1	Basic
Density	rho	8000	kg/m <sup>3</sup>	Basic

### *Stainless Steel (mat1)*

Add Young's modulus as function of temperature for the stainless steel.

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Stainless Steel (mat1)** node.

### *Interpolation 1 (int1)*

- 1 Right-click **Basic (def)** and choose **Functions>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type **fE**.

4 In the table, enter the following settings:

t	f(t)
20	194
100	189
200	179
300	175

5 Locate the **Units** section. In the **Arguments** text field, type degC.

6 In the **Function** text field, type GPa.

Add thermal expansion as function of temperature for the stainless steel.

*Interpolation 2 (int2)*

1 Right-click **Component 1 (comp1)**>**Materials**>**Stainless Steel (mat1)**>**Basic (def)**>**Interpolation 1 (int1)** and choose **Functions**>**Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type fA.

4 In the table, enter the following settings:

t	f(t)
20	16e-6
100	16.5e-6
200	17e-6
300	17.5e-6

5 Locate the **Units** section. In the **Arguments** text field, type degC.

6 In the **Function** text field, type 1/K.

Add thermal conductivity as function of temperature for the stainless steel.

*Interpolation 3 (int3)*

1 Right-click **Component 1 (comp1)**>**Materials**>**Stainless Steel (mat1)**>**Basic (def)**>**Interpolation 2 (int2)** and choose **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type fK.

4 In the table, enter the following settings:

t	f(t)
20	13.9
100	15.5
200	16.8
300	17.8

5 Locate the **Units** section. In the **Arguments** text field, type degC.

6 In the **Function** text field, type  $W / (m \cdot \text{degC})$ .

Add heat capacity as function of temperature for the stainless steel.

*Interpolation 4 (int4)*

1 Right-click **Component 1 (comp1)**>**Materials**>**Stainless Steel (mat1)**>**Basic (def)**>**Interpolation 3 (int3)** and choose **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type fCp.

4 In the table, enter the following settings:

t	f(t)
20	482
100	504
200	521
300	530

5 Locate the **Units** section. In the **Arguments** text field, type degC.

6 In the **Function** text field, type  $J / (kg \cdot \text{degC})$ .

*Stainless Steel (mat1)*

Add the temperature as a model input for the Basic properties.

1 In the **Model Builder** window, under **Component 1 (comp1)**>**Materials**>**Stainless Steel (mat1)** click **Basic (def)**.

2 In the **Settings** window for **Property Group**, locate the **Output Properties and Model Inputs** section.

3 Find the **Quantities** subsection. In the tree, select **Model Inputs**>**Temperature**.

4 Click **Add**.

5 In the **Model Builder** window, click **Stainless Steel (mat1)**.

- 6 In the **Settings** window for **Material**, click to expand the **Material properties** section.
- 7 Locate the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Elastoplastic Material>Elastoplastic Material Model**.
- 8 Click **Add to Material**.  
Load the table containing yield stress as function of plastic strain and temperature.

*Interpolation 1 (int1)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials>Stainless Steel (mat1)** right-click **Elastoplastic material model (ElastoplasticModel)** and choose **Functions>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 From the **Data source** list, choose **File**.
- 4 Click **Browse**.
- 5 Browse to the model's Application Libraries folder and double-click the file `temperature_dependent_plasticity_function.txt`.
- 6 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
sY	1

- 7 Locate the **Units** section. In the **Arguments** text field, type `degC, 1`.
- 8 In the **Function** text field, type `Pa`.
- 9 Locate the **Definition** section. Click **Import**.

*Stainless Steel (mat1)*

Add the temperature and the effective plastic strain as model inputs for the Elastoplastic material model properties.

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials>Stainless Steel (mat1)** click **Elastoplastic material model (ElastoplasticModel)**.
- 2 In the **Settings** window for **Property Group**, locate the **Output Properties and Model Inputs** section.
- 3 Find the **Quantities** subsection. In the tree, select **Model Inputs>Temperature**.
- 4 Click **Add**.
- 5 In the tree, select **Model Inputs>Effective Plastic Strain**.
- 6 Click **Add**.
- 7 In the **Model Builder** window, click **Stainless Steel (mat1)**.

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

9 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Initial yield stress	sigmags	$sY(T, 0)$	Pa	Elastoplastic material model
Hardening function	sigmagh	$sY(T, \epsilon_p) - sY(T, 0)$	Pa	Elastoplastic material model
Young's modulus	E	$fE(T)$	Pa	Basic
Thermal conductivity	k	$fK(T)$	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	$fCp(T)$	J/(kg·K)	Basic
Coefficient of thermal expansion	alpha	$fA(T)$	1/K	Basic

The hardening function is the stress increase from the initial yield stress. As the full stress-strain curve is given, subtract the stress at zero strain.

#### *Material 2 (mat2)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Carbon Steel in the **Label** text field.
- 3 Select Domains 2 and 4 only.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Poisson's ratio	nu	0.3	1	Basic
Density	rho	8000	kg/m <sup>3</sup>	Basic

#### *Carbon Steel (mat2)*

Add Young's modulus as function of temperature for the carbon steel.

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Carbon Steel (mat2)** node.

#### *Interpolation 1 (int1)*

- 1 Right-click **Basic (def)** and choose **Functions>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type  $fE$ .

4 In the table, enter the following settings:

<b>t</b>	<b>f(t)</b>
20	208
100	202
200	196
300	189

5 Locate the **Units** section. In the **Arguments** text field, type  $\text{degC}$ .

6 In the **Function** text field, type  $\text{GPa}$ .

Add thermal expansion as function of temperature for the carbon steel.

*Interpolation 2 (int2)*

1 Right-click **Component 1 (comp1)**>**Materials**>**Carbon Steel (mat2)**>**Basic (def)**>**Interpolation 1 (int1)** and choose **Functions**>**Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type  $fA$ .

4 In the table, enter the following settings:

<b>t</b>	<b>f(t)</b>
20	$10.9\text{e-}6$
100	$12.4\text{e-}6$
200	$13.8\text{e-}6$
300	$14.9\text{e-}6$

5 Locate the **Units** section. In the **Arguments** text field, type  $\text{degC}$ .

6 In the **Function** text field, type  $1/\text{K}$ .

Add thermal conductivity as function of temperature for the carbon steel.

*Interpolation 3 (int3)*

1 Right-click **Component 1 (comp1)**>**Materials**>**Carbon Steel (mat2)**>**Basic (def)**>**Interpolation 2 (int2)** and choose **Functions**>**Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type  $fK$ .

4 In the table, enter the following settings:

t	f(t)
20	51.2
100	48.3
200	45.5
300	42.7

5 Locate the **Units** section. In the **Arguments** text field, type degC.

6 In the **Function** text field, type  $W / (m \cdot \text{degC})$ .

Add heat capacity as function of temperature for the carbon steel.

*Interpolation 4 (int4)*

1 Right-click **Component 1 (comp1)>Materials>Carbon Steel (mat2)>Basic (def)>Interpolation 3 (int3)** and choose **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type  $fCp$ .

4 In the table, enter the following settings:

t	f(t)
20	489
100	519
200	546
300	569

5 Locate the **Units** section. In the **Arguments** text field, type degC.

6 In the **Function** text field, type  $J / (kg \cdot \text{degC})$ .

*Carbon Steel (mat2)*

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Carbon Steel (mat2)**.

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
Young's modulus	E	$fE(T)$	Pa	Basic
Thermal conductivity	k	$fK(T)$	$W/(m \cdot K)$	Basic

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Cp	fCp(T)	J/(kg·K)	Basic
Coefficient of thermal expansion	alpha	fA(T)	1/K	Basic

## DEFINITIONS

Add the time history for the temperature of the water in the pipe.

### Step 1 (step1)

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Step**.
- 2 In the **Settings** window for **Step**, type pipeWaterTemp in the **Function name** text field.
- 3 Locate the **Parameters** section. In the **Location** text field, type 1.
- 4 In the **From** text field, type 280.
- 5 In the **To** text field, type 20.

Add the time history for the temperature of the water in the pressure vessel.

### Interpolation 1 (int1)

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type vesselWaterTemp.
- 4 In the table, enter the following settings:

t	f(t)
0	280
1800	20

- 5 Locate the **Units** section. In the **Arguments** text field, type s.
- 6 In the **Function** text field, type degC.

Add an operator for capturing the x-displacement in a point at the pipe end.

### Integration 1 (intop1)

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Point**.
- 4 Select Point 23 only.

Add an operator for capturing the z-displacement in a point at the cut in the pressure vessel.

*Integration 2 (intop2)*

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Point**.
- 4 Select Point 14 only.

Create variables containing the displacements where the model is cut.

*Variables 1*

- 1 On 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:

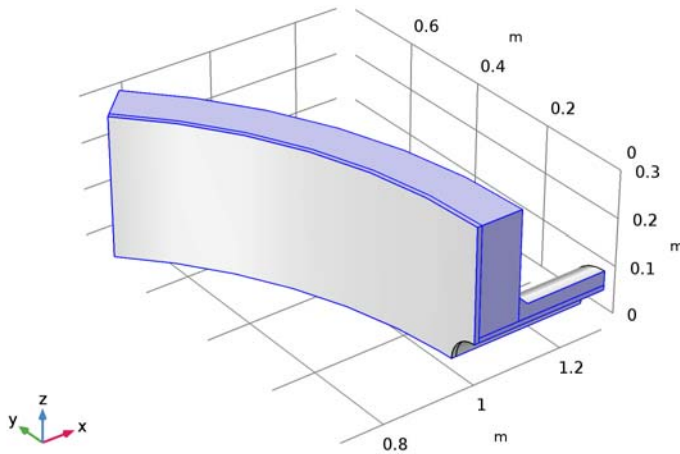
Name	Expression	Unit
edgeX	intop1(u)	m
edgeZ	intop2(w)	m

Create an explicit selection to use in the symmetry boundary conditions.

*Explicit 1*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Symmetry Boundaries in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

- 4 Select Boundaries 1, 3–5, 7, 8, 11, 13, 16, 17, 19, and 22 only.



## HEAT TRANSFER IN SOLIDS (HT)

### *Initial Values 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Heat Transfer in Solids (ht)** click **Initial Values 1**.
- 2 In the **Settings** window for **Initial Values**, type 280[degC] in the  $T$  text field.
- 3 In the **Model Builder** window, click **Heat Transfer in Solids (ht)**.

### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry Boundaries**.

### *Heat Flux 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
- 2 Select Boundaries 9, 20, and 23 only.
- 3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
- 4 Click the **Convective heat flux** button.
- 5 In the  $h$  text field, type 10.
- 6 In the  $T_{\text{ext}}$  text field, type 50[degC].

### *Heat Flux 2*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
- 2 Select Boundaries 10 and 15 only.
- 3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
- 4 Click the **Convective heat flux** button.
- 5 In the  $h$  text field, type 100.
- 6 In the  $T_{\text{ext}}$  text field, type `pipeWaterTemp(t[1/s])[degC]`.

### *Heat Flux 3*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
- 4 Click the **Convective heat flux** button.
- 5 In the  $h$  text field, type 100.
- 6 In the  $T_{\text{ext}}$  text field, type `vesselWaterTemp(t)`.

## **SOLID MECHANICS (SOLID)**

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

### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry Boundaries**.

You will overwrite the extra boundaries of the explicit selection with a Prescribed Displacement condition below.

### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundaries 2, 10, and 15 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 From the **Load type** list, choose **Pressure**.
- 5 In the  $p$  text field, type `internalPressure`.

### *Boundary Load 2*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundaries 4 and 8 only.

**3** In the **Settings** window for **Boundary Load**, locate the **Force** section.

**4** Specify the  $\mathbf{F}_A$  vector as

0	x
0	y
$4.76 \cdot \text{internalPressure}$	z

Make the cuts stay plane under deformation by prescribing the normal displacement over the entire boundary to be equal to the value in a single point in the boundary.

#### *Prescribed Displacement 1*

**1** On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

**2** Select Boundaries 24 and 25 only.

**3** In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

**4** Select the **Prescribed in x direction** check box.

**5** In the  $u_{0,x}$  text field, type edgeX.

#### *Prescribed Displacement 2*

**1** On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

**2** Select Boundaries 4 and 8 only.

**3** In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

**4** Select the **Prescribed in z direction** check box.

**5** In the  $u_{0,z}$  text field type edgeZ

### **STUDY 1**

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

**2** In the **Settings** window for **Study**, type Study 1: Initialization in the **Label** text field.

In an initialization study, the mechanical and thermal stationary state is computed. This serves as initial conditions for a transient step.

**3** On the **Home** toolbar, click **Compute**.

### **ADD STUDY**

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

In a second study, the transient temperature distribution is computed.

- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Time Dependent**.
- 4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Solid Mechanics (solid)** interface.
- 5 Click **Add Study** in the window toolbar.

## STUDY 2

### *Step 1: Time Dependent*

- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 In the **Times** text field, type `range(0,0.2,2) range(60,60,1800)`.  
Analyze half an hour, storing the results once every minute. First steps are refined in order to improve the convergence.
- 3 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 4 From the **Method** list, choose **Solution**.
- 5 From the **Study** list, choose **Study 1: Initialization, Stationary**.
- 6 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
- 7 From the **Method** list, choose **Solution**.
- 8 From the **Study** list, choose **Study 1: Initialization, Stationary**.
- 9 In the **Model Builder** window, click **Study 2**.
- 10 In the **Settings** window for **Study**, type Study 2: Heat Transfer in the **Label** text field.
- 11 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

### *Solution 2 (sol2)*

On the **Study** toolbar, click **Show Default Solver**.

## STUDY 2: HEAT TRANSFER

### *Solution 2 (sol2)*

- 1 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

- 2 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

Since cold water is suddenly injected in the vessel, it is important to capture the development accurately, so you want to enforce intermediate steps.

- 3 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Intermediate**.
- 4 On the **Study** toolbar, click **Compute**.

#### **ADD STUDY**

- 1 Go to the **Add Study** window.
- 2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.
- 3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Heat Transfer in Solids (ht)** interface.
- 4 Click **Add Study** in the window toolbar.

In a third study, the elasto-plastic problem is computed using a stationary continuation study step.

#### **STUDY 3**

##### *Step 1: Stationary*

- 1 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.
- 2 In the **Settings** window for **Stationary**, click to expand the **Values of dependent variables** section.
- 3 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 4 From the **Method** list, choose **Solution**.
- 5 From the **Study** list, choose **Study 1: Initialization, Stationary**.
- 6 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
- 7 From the **Method** list, choose **Solution**.
- 8 From the **Study** list, choose **Study 2: Heat Transfer, Time Dependent**.
- 9 From the **Time (s)** list, choose **All**.
- 10 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t (Time variable; used for stationary analysis)	range (0,0.2,2) range (60,60,1800)	s

13 In the **Settings** window for **Study**, type Study 3: Plasticity in the **Label** text field.

14 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

*Solution 3 (sol3)*

On the **Study** toolbar, click **Show Default Solver**.

### STUDY 3: PLASTICITY

*Solution 3 (sol3)*

1 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

2 In the **Model Builder** window, expand the **Study 3: Plasticity>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node, then click **Fully Coupled 1**.

3 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

Use a double dogleg solver in order to improve the convergence.

4 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Double dogleg**.

5 On the **Study** toolbar, click **Compute**.

### RESULTS

*Surface 1*

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

*Stress (solid)*

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

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3: Plasticity/Solution 3 (sol3)**.

### *Deformation*

- 1 In the **Model Builder** window, under **Results>Stress (solid)>Surface 1** click **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 Select the **Scale factor** check box.
- 4 In the associated text field, type 0.
- 5 On the **Stress (solid)** toolbar, click **Plot**.

### *Stress (solid) 1*

- 1 In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Duplicate**.
- 2 In the **Settings** window for **3D Plot Group**, type Plastic Strain in the **Label** text field.

### *Plastic strain*

- 1 In the **Model Builder** window, expand the **Results>Plastic Strain** node.
- 2 Right-click **Plastic strain** and choose **Disable**.

### *Surface 1*

- 1 In the **Model Builder** window, under **Results>Plastic Strain** click **Surface 1**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Effective plastic strain**.
- 3 On the **Plastic Strain** toolbar, click **Plot**.

### *Plastic Strain*

- 1 In the **Model Builder** window, under **Results** click **Plastic Strain**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Parameter value (t (s))** list, choose **0**.
- 4 On the **Plastic Strain** toolbar, click **Plot**.
- 5 From the **Parameter value (t (s))** list, choose **1200**.
- 6 On the **Plastic Strain** toolbar, click **Plot**.

### *Temperature (ht)*

- 1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Data set** list, choose **Study 3: Plasticity/Solution 3 (sol3)**.

### *Surface*

- 1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.
- 4 On the **Temperature (ht)** toolbar, click **Plot**.

#### *Temperature (ht)*

- 1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Parameter value (t (s))** list, choose **0**.
- 4 On the **Temperature (ht)** toolbar, click **Plot**.

### *Appendix — Geometry 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 **3D**.
- 2 In the **Select Physics** tree, select **Structural Mechanics>Thermal Stress**.
- 3 Click **Add**.
- 4 Click **Study**.
- 5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
- 6 Click **Done**.

#### **GEOMETRY I**

##### *Work Plane 1 (wpl)*

- 1 On the **Geometry** toolbar, click **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
- 3 From the **Plane** list, choose **xz-plane**.
- 4 Locate the **Unite Objects** section. Clear the **Unite objects** check box.
- 5 Click **Show Work Plane**.

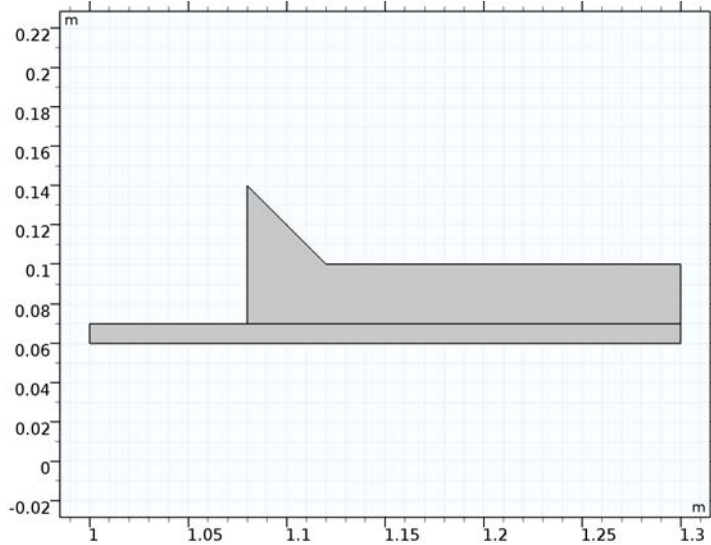
### *Bézier Polygon 1 (b1)*

- 1** On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.
- 2** In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
- 3** Find the **Added segments** subsection. Click **Add Linear**.
- 4** Find the **Control points** subsection. In row **1**, set **xw** to 1.3 and **yw** to 0.07.
- 5** In row **2**, set **xw** to 1.08 and **yw** to 0.07.
- 6** Find the **Added segments** subsection. Click **Add Linear**.
- 7** Find the **Control points** subsection. In row **2**, set **yw** to 0.14.
- 8** Find the **Added segments** subsection. Click **Add Linear**.
- 9** Find the **Control points** subsection. In row **2**, set **xw** to 1.12 and **yw** to 0.1.
- 10** Find the **Added segments** subsection. Click **Add Linear**.
- 11** Find the **Control points** subsection. In row **2**, set **xw** to 1.3 and **yw** to 0.1.
- 12** Find the **Added segments** subsection. Click **Add Linear**.
- 13** Find the **Control points** subsection. Click **Close Curve**.
- 14** Right-click **Bézier Polygon 1 (b1)** and choose **Build Selected**.

### *Rectangle 1 (r1)*

- 1** On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.
- 2** In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3** In the **Width** text field, type 0.3.
- 4** In the **Height** text field, type 0.01.
- 5** Locate the **Position** section. In the **xw** text field, type 1.
- 6** In the **yw** text field, type 0.06.

7 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.



*Work Plane 1 (wp1)*

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

*Revolve 1 (rev1)*

- 1 On the **Geometry** toolbar, click **Revolve**.
- 2 In the **Settings** window for **Revolve**, locate the **Revolution Axis** section.
- 3 Find the **Direction of revolution axis** subsection. In the **xw** text field, type 1.
- 4 In the **yw** text field, type 0.
- 5 Locate the **Revolution Angles** section. Click the **Angles** button.
- 6 In the **End angle** text field, type -90.
- 7 Right-click **Revolve 1 (rev1)** and choose **Build Selected**.

*Work Plane 2 (wp2)*

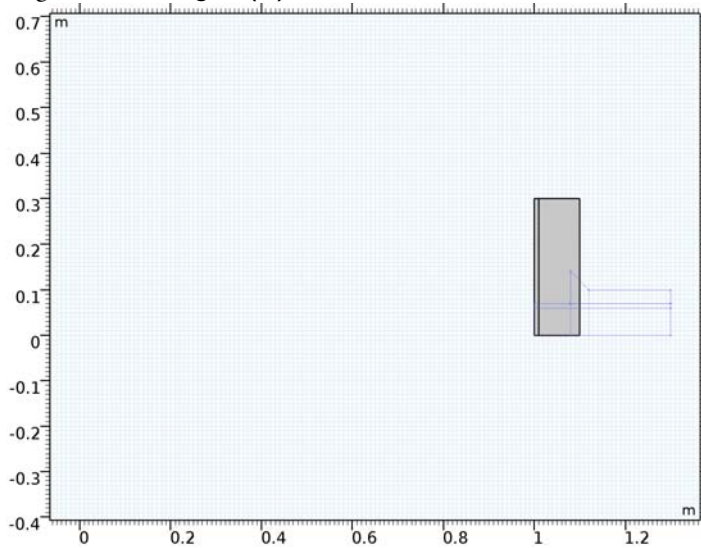
- 1 On the **Geometry** toolbar, click **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
- 3 From the **Plane** list, choose **xz-plane**.
- 4 Locate the **Unite Objects** section. Clear the **Unite objects** check box.
- 5 Click **Show Work Plane**.

### Rectangle 1 (r1)

- 1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 0.09.
- 4 In the **Height** text field, type 0.3.
- 5 Locate the **Position** section. In the **xw** text field, type 1.01.
- 6 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

### Rectangle 2 (r2)

- 1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 0.01.
- 4 In the **Height** text field, type 0.3.
- 5 Locate the **Position** section. In the **xw** text field, type 1.
- 6 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.



### Work Plane 2 (wp2)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 2 (wp2)**.

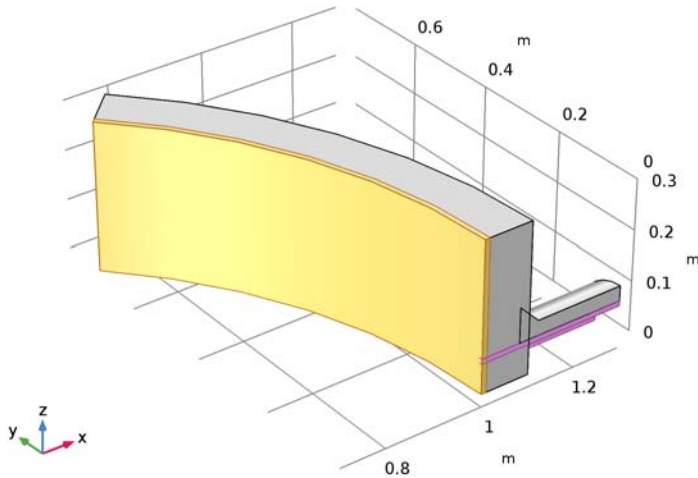
### Revolve 2 (rev2)

- 1 On the **Geometry** toolbar, click **Revolve**.

- 2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.
- 3 Click the **Angles** button.
- 4 In the **End angle** text field, type 45.
- 5 Right-click **Revolve 2 (rev2)** and choose **Build Selected**.
- 6 Click the **Zoom Extents** button on the **Graphics** toolbar.

*Difference 1 (dif1)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 In the **Settings** window for **Difference**, locate the **Difference** section.
- 3 Select the **Keep input objects** check box.
- 4 Clear the **Keep interior boundaries** check box.
- 5 Select the object **rev1(2)** only.
- 6 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 7 Select the object **rev2(2)** only.

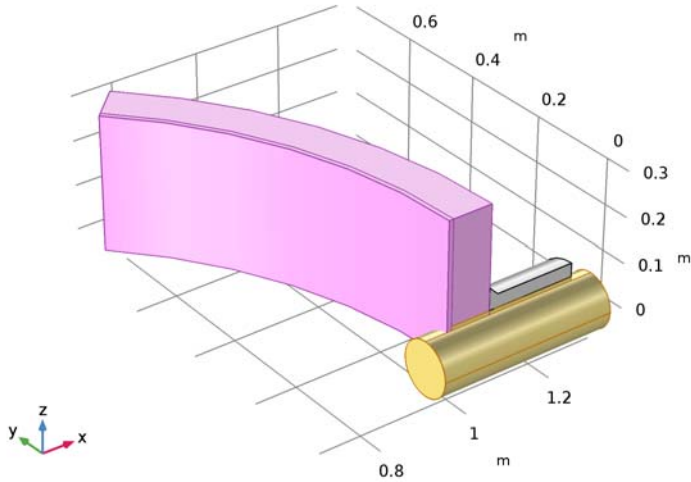


- 8 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

*Difference 2 (dif2)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 In the **Settings** window for **Difference**, locate the **Difference** section.
- 3 Select the **Keep input objects** check box.

- 4 Select the object **rev1(1)** only.
- 5 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 6 Select the object **rev2(1)** only.

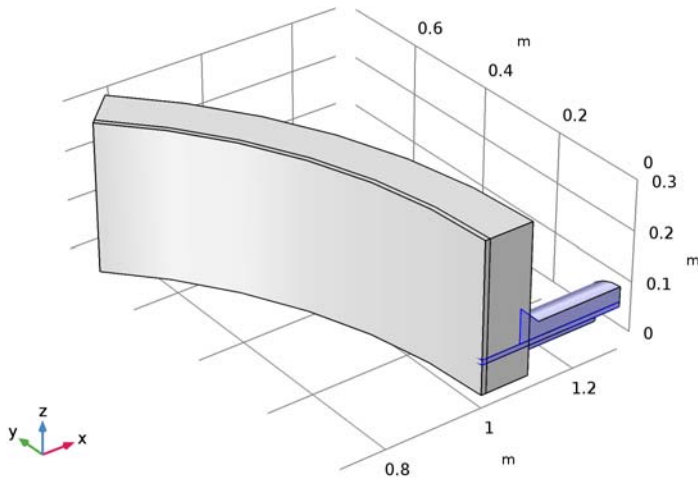


- 7 Right-click **Difference 2 (dif2)** and choose **Build Selected**.

*Delete Entities 1 (del1)*

- 1 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 **Object**.

- 4 Select the objects **rev1(1)** and **rev1(2)** only.



- 5 Right-click **Component 1 (comp1)>Geometry 1>Delete Entities 1 (del1)** and choose **Build Selected**.

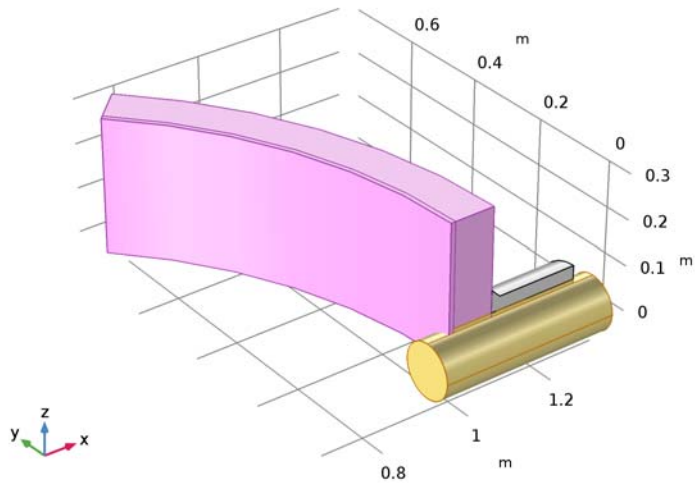
#### *Cylinder 1 (cyl1)*

- 1 On the **Geometry** toolbar, click **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 0.06.
- 4 In the **Height** text field, type 0.4.
- 5 Locate the **Position** section. In the **x** text field, type 0.95.
- 6 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.
- 7 In the **x** text field, type 1.
- 8 In the **z** text field, type 0.
- 9 Right-click **Cylinder 1 (cyl1)** and choose **Build Selected**.

#### *Difference 3 (dif3)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 In the **Settings** window for **Difference**, locate the **Difference** section.
- 3 Select the **Keep input objects** check box.
- 4 Select the objects **rev2(1)** and **rev2(2)** only.
- 5 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

6 Select the objects **cyll** and **dif1** only.



7 Right-click **Difference 3 (dif3)** and choose **Build Selected**.

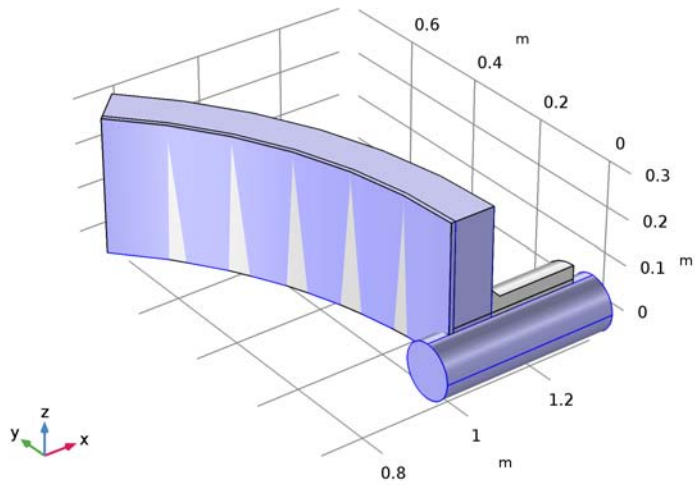
*Delete Entities 2 (del2)*

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

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

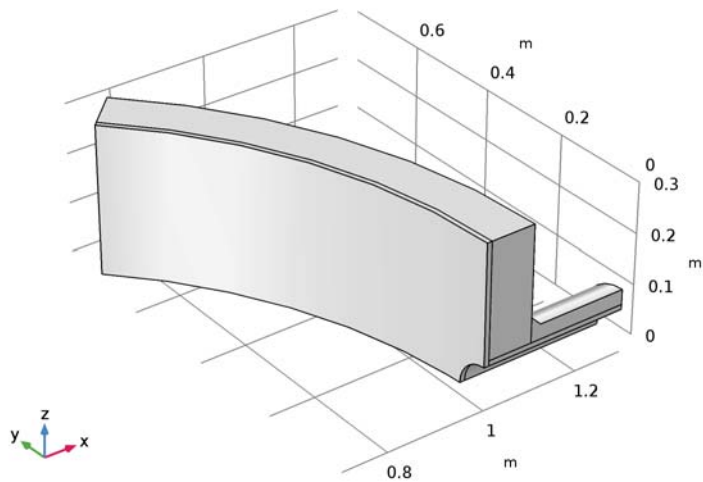
3 From the **Geometric entity level** list, choose **Object**.

4 Select the objects **rev2(1)**, **rev2(2)**, and **cyl1** only.

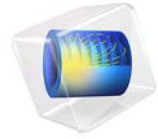


5 Right-click **Component 1 (comp1)**>**Geometry 1**>**Delete Entities 2 (del2)** and choose **Build Selected**.

6 On the **Geometry** toolbar, click **Build All**.







# Thermally Induced Creep

## Introduction

---

This example computes the stress history over a very long time for a material that exhibits creep behavior. The model is taken from *NAFEMS Selected Benchmarks For Material Non-Linearity, Volume 2* (Ref. 1). The displacement and stress levels are compared with the values given in the reference.

## Model Definition

---

The geometry is a hollow sphere with inner radius 200 mm and outer radius 500 mm. The problem has rotational symmetry where the solution depends only on the radial coordinate. You could therefore select any section having radial cuts as the computational domain. To follow the original example the sphere is modeled with a 2D axisymmetric  $10^\circ$  sector with symmetry constraint conditions applied on edges of the sector; see Figure 1.

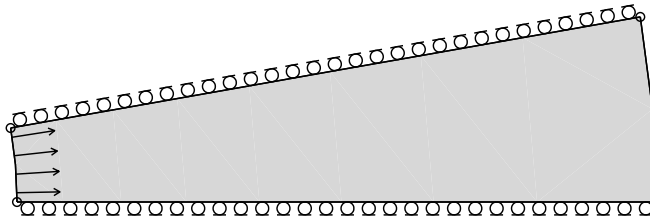


Figure 1: The model geometry, using a  $10^\circ$  sector of the original geometry.

### MATERIAL PROPERTIES

- Isotropic with  $E = 10$  GPa,  $\nu = 0.25$ .
- Creep data according to:

$$\dot{\epsilon}_c = A_1 \sigma_e^{n_1} f_2(T) \quad (1)$$

with  $A_1 = 3.0 \cdot 10^{-6} \text{ h}^{-1}$  that accounts for the stress normalization of the effective stress,  $\sigma_e$ , in MPa,  $n_1 = 5.5$ , and  $f_2(T) = e^{-12500/T}$ , where  $T$  defines the temperature in K.

### LOADS

- An internal pressure of 30 MPa.
- A temperature field with the distribution  $T = 333(1 + 100/(\sqrt{R^2 + Z^2}))$  where  $R$  and  $Z$  are material coordinates in mm.

## Results and Discussion

The evolution of displacement with time appears in Figure 2. The upper curve represents the inner radius, and the lower curve represents the outer radius.

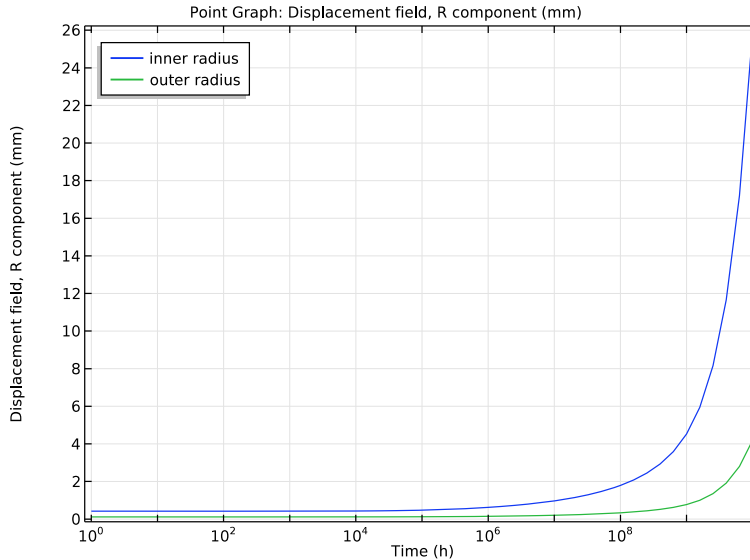


Figure 2: Radial displacement at inner radius and outer radius with time.

In the following table you can compare the values at time  $10^{10}$  h with the reference values:

RADIUS	COMSOL MULTIPHYSICS	REFERENCE (Ref. 1)
200 mm	26.0 mm	26.1 mm
500 mm	4.20 mm	4.22 mm

Initially the mechanical and thermal load have greater influence on the inner boundary of the sphere and results in larger creep strains. This with time causes relaxation that

propagates from the inner radius towards the outer radius. This phenomena is visible in Figure 3 where the von Mises effective stress is shown at  $10^8$  h.

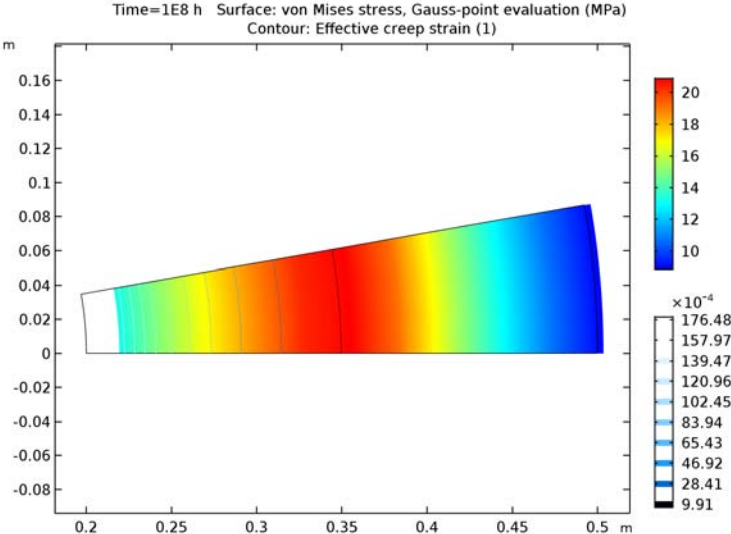


Figure 3: von Mises effective stress at  $t = 10^8$  h.

The next graph shows the variation of the von Mises effective stress with time at the inner, middle, and outer radii. Notice that significant changes in the stress state occur already at the time  $10^4$  h, that is, after one millionth of the total analysis time. In the final state the

stresses are completely redistributed. The mechanical load is then larger on the outer exterior than the inner exterior.

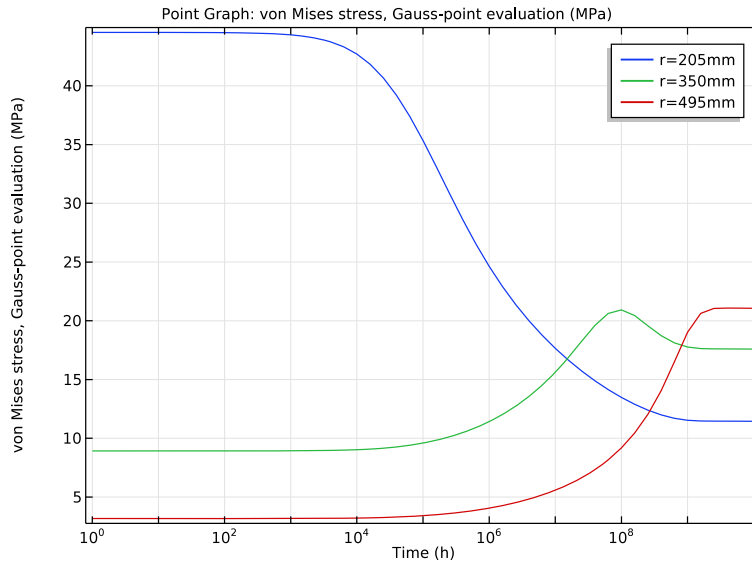


Figure 4: Time history of the von Mises effective stress at  $r = 205$  mm,  $350$  mm, and  $495$  mm.

The following table shows the final values of the von Mises effective stress at  $t = 10^{10}$  h and the reference values for comparison:

RADIUS	COMSOL MULTIPHYSICS	REFERENCE (Ref. 1)
205 mm	11.4 MPa	11.5 MPa
350 mm	17.6 MPa	17.6 MPa
495 mm	21.1 MPa	21.1 MPa

### Notes About the COMSOL Implementation

An interesting feature of creep problems is the extreme variation in the time scales over which different phenomena occur. Figure 4 shows that the a significant stress change starts after about 1000 h. It is therefore wise to solve for time steps before and after any significant change in the response. In order to capture this onset of the stress change a strict time stepping is used which forces the solver to provide a solution for all specified time steps. Alternative ways is to do it is to either provide an analytical solution for the

inner pressure as initial conditions or to first solve a stationary problem with the inner pressure followed by a time study.

The creep law defined in [Equation 1](#) follows Norton law which is accessible in the Nonlinear Structural Materials Module. In COMSOL Multiphysics it is defined as

$$\dot{\epsilon}_c = A \left( \frac{\sigma_e}{\sigma_r} \right)^n e^{-\frac{Q}{RT}}$$

In order to normalize the effective stress in MPa set the reference stress  $\sigma_r = 1$  MPa. In the exponential temperature function  $R = 8.314$  J/(mol·K) and therefore the creep activation energy  $Q = 1.039 \cdot 10^5$  J/mol.

### *Reference*

---

1. D. Linkens, *Selected Benchmarks For Material Non-Linearity, vol 2*, NAFEMS, 1993.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/Creep/  
thermally\_induced\_creep

---

### *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 Axisymmetric**.
- 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 **Preset Studies>Time Dependent**.
- 6** Click **Done**.

## DEFINITIONS

### *Variables 1*

- 1 On the **Home** toolbar, click **Variables** and choose **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
T	$333[\text{K}] * (1 + 0.1[\text{m}] / \sqrt{R * R + Z * Z})$	K	Prescribed temperature field

## GEOMETRY 1

### *Circle 1 (c1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 0.5.
- 4 In the **Sector angle** text field, type 10.
- 5 Click **Build All Objects**.
- 6 Click the **Zoom Extents** button on the **Graphics** toolbar.

### *Circle 2 (c2)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type 0.2.
- 4 In the **Sector angle** text field, type 10.
- 5 Click **Build All Objects**.

### *Difference 1 (dif1)*

- 1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **c1** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 5 Select the object **c2** only.
- 6 Click **Build All Objects**.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

## SOLID MECHANICS (SOLID)

### *Linear Elastic Material 1*

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

### *Creep 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Creep**.
- 2 In the **Settings** window for **Creep**, locate the **Model Input** section.
- 3 In the  $T$  text field, type  $T$ .
- 4 Locate the **Creep Data** section. Select the **Include temperature dependency** check box.
- 5 In the  $Q$  text field, type  $1.0393e5[\text{J/mol}]$ .

### *Symmetry 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 1 and 2 only.

### *Boundary Load 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Coordinate System Selection** section.
- 4 From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.
- 5 Locate the **Force** section. Specify the  $\mathbf{F}_A$  vector as

0	$t $
$-30[\text{MPa}]$	$n$

- 6 In the **Model Builder** window, click **Solid Mechanics (solid)**.
- 7 In the **Settings** window for **Solid Mechanics**, locate the **Structural Transient Behavior** section.
- 8 From the list, choose **Quasi-static**.

## 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
Young's modulus	E	10 [GPa]	Pa	Basic
Poisson's ratio	nu	0.25		Basic
Density	rho	1000	kg/m <sup>3</sup>	Basic
Creep rate coefficient	A_nor	3e-6 [1/h]	1/s	Norton
Reference stress	sigRef_nor	1 [MPa]	N/m <sup>2</sup>	Norton
Stress exponent	n_nor	5.5		Norton

## MESH I

### Size

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Coarse**.
- 4 Click **Build All**.

## STUDY I

### Step 1: Time Dependent

- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 From the **Time unit** list, choose **h**.
- 3 In the **Times** text field, type 0.
- 4 Click **Range**.
- 5 In the **Range** dialog box, type 0 in the **Start** text field.
- 6 In the **Stop** text field, type 10.
- 7 In the **Step** text field, type 0.2.
- 8 From the **Function to apply to all values** list, choose **exp10(x) – Exponential function (base 10)**.
- 9 Click **Add**.
- 10 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 11 From the **Tolerance** list, choose **User controlled**.

**12** In the **Relative tolerance** text field, type  $1e-4$ .

#### *Solution 1 (sol1)*

- 1** On the **Study** toolbar, click **Show Default Solver**.
- 2** In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
- 3** In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.
- 4** Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.
- 5** Select the **Initial step** check box.
- 6** In the associated text field, type  $1$  [min].  
Setting the initial step ensures a good calculation of creep strain at  $t = 0$ .
- 7** Click **Compute**.

## **RESULTS**

#### *Stress (solid)*

Select the solution at  $10^8$  hours to reproduce [Figure 3](#).

- 1** In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2** In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 3** From the **Time (h)** list, choose **1E8**.

#### *Surface 1*

- 1** In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
- 2** In the **Settings** window for **Surface**, locate the **Expression** section.
- 3** From the **Unit** list, choose **MPa**.
- 4** On the **Stress (solid)** toolbar, click **Plot**.
- 5** Click the **Zoom Extents** button on the **Graphics** toolbar.

#### *Surface 1*

- 1** In the **Model Builder** window, expand the **Results>Stress, 3D (solid)** node, then click **Surface 1**.
- 2** In the **Settings** window for **Surface**, locate the **Expression** section.
- 3** From the **Unit** list, choose **MPa**.

Follow the commands below to generate [Figure 2](#).

### *ID Plot Group 3*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Radial Displacement in the **Label** text field.
- 3 Locate the **Axis** section. Select the **x-axis log scale** check box.

### *Point Graph 1*

- 1 Right-click **Radial Displacement** and choose **Point Graph**.
- 2 Select Points 2 and 4 only.
- 3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>Displacement field (material and geometry frames)>u - Displacement field, R component**.
- 4 Locate the **y-Axis Data** section. From the **Unit** list, choose **mm**.
- 5 Click to expand the **Legends** section. Select the **Show legends** check box.
- 6 From the **Legends** list, choose **Manual**.
- 7 In the table, enter the following settings:

<b>Legends</b>
inner radius
outer radius

### *Radial Displacement*

- 1 In the **Model Builder** window, under **Results** click **Radial Displacement**.
- 2 In the **Settings** window for **ID Plot Group**, click to expand the **Legend** section.
- 3 From the **Position** list, choose **Upper left**.
- 4 On the **Radial Displacement** toolbar, click **Plot**.

### *Data Sets*

The commands below generate [Figure 4](#).

### *Cut Point 2D 1*

- 1 On the **Results** toolbar, click **Cut Point 2D**.
- 2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.
- 3 In the **R** text field, type 205[mm] 350[mm] 495[mm].
- 4 In the **Z** text field, type 0.

#### *ID Plot Group 4*

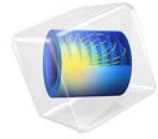
- 1 On the **Results** toolbar, click **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Von Mises Stress in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D I**.
- 4 Locate the **Axis** section. Select the **x-axis log scale** check box.

#### *Point Graph 1*

- 1 Right-click **Von Mises Stress** and choose **Point Graph**.
- 2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress (Gauss points)>solid.misesGp - von Mises stress, Gauss-point evaluation**.
- 3 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.
- 4 Click to expand the **Legends** section. Select the **Show legends** check box.
- 5 From the **Legends** list, choose **Manual**.
- 6 In the table, enter the following settings:

<b>Legends</b>
r=205mm
r=350mm
r=495mm

- 7 On the **Von Mises Stress** toolbar, click **Plot**.



# Uniaxial Loading of Shape Memory Alloy

## Introduction

---

Shape memory alloys (SMAs) are used in a wide range of applications due to their ability to recover their initial shape when heated, and to the hysteresis they show during a loading-unloading cycle before recovering their initial state. These two properties, called shape memory effect and pseudoelasticity, make these alloys interesting for many industrial applications, such as the aerospace, transportation, and medical industries.

This example model shows the SMA behavior under uniaxial loading. Three studies are performed: a parametric sweep shows the pseudoelasticity effect at different fixed temperatures, a prescribed displacement sweep shows the pseudoelasticity effect is a partial unloading-partial loading loop, and the shape memory effect is portrayed after increasing the temperature.

### PHASE TRANSFORMATION

Shape memory alloys exist in two crystalline structures. Austenite is present at higher temperature, and Martensite is present at lower temperatures. The transformation from one to other phase is the cause of the specific behavior of SMAs. In the general case the material contains both austenite and martensite phases, due an incomplete transformation. The transformation rate is denoted by the martensite volume fraction  $\xi$ .

According to [Ref. 1](#), the evolution of the transformation is calculated by Kuhn-Tucker conditions on the general thermodynamics force conjugated to the martensite volume factor  $\xi$ :

$$\begin{aligned} \text{if } \xi > 0; \quad & \pi - Y \leq 0; \quad (\pi - Y)\xi \leq 0 \\ \text{if } \xi < 0; \quad & -\pi - Y \leq 0; \quad (-\pi - Y)\xi \leq 0 \end{aligned}$$

where

$$\pi(\sigma, T, \xi) = \sigma:\Lambda + \frac{1}{2}\sigma:\Delta S:\sigma + \rho C_p \left( (T - T_0) - T \ln\left(\frac{T}{T_0}\right) \right) + \rho \Delta s_0 T - \rho \Delta u_0 - \frac{\partial f}{\partial \xi}$$

## Model Definition

A cylinder of nickel-titanium (NiTi) alloy is submitted to axial tension. This is modeled by a rectangle in a 2D axisymmetric geometry. The **Shape Memory Alloy** feature uses Lagoudas SMA model. The material properties for each phase are given by:

TABLE 1: MATERIAL PROPERTIES OF PHASES

MATERIAL PROPERTY	AUSTENITE	MARTENSITE
Young's modulus	55 GPa	46 GPa
Poisson's ratio	0.33	0.33
Heat capacity at constant pressure	400 J/(kg.K)	400 J/(kg.K)
Density	6500 kg/m <sup>3</sup>	6500 kg/m <sup>3</sup>

The phase transformation parameters are:

TABLE 2: PHASE TRANSFORMATION PARAMETERS

PARAMETER	VALUE
Martensite start temperature	245 K
Martensite finish temperature	230 K
Austenite start temperature	270 K
Austenite finish temperature	280 K
Slope of martensite limit curve	7.4 MPa/K
Slope of austenite limit curve	7.4 MPa/K
Maximum transformation strain	0.056

The first study runs a parametric sweep for four different prescribed temperatures. Two of the temperatures (328 K, 308 K) are above the austenite finish temperature, one temperature (276 K) lies between the austenite start and finish temperature, and one temperature (260 K) is below the austenite start temperature.

The boundary load is a loading-unloading cycle up to 850 MPa (Figure 1).

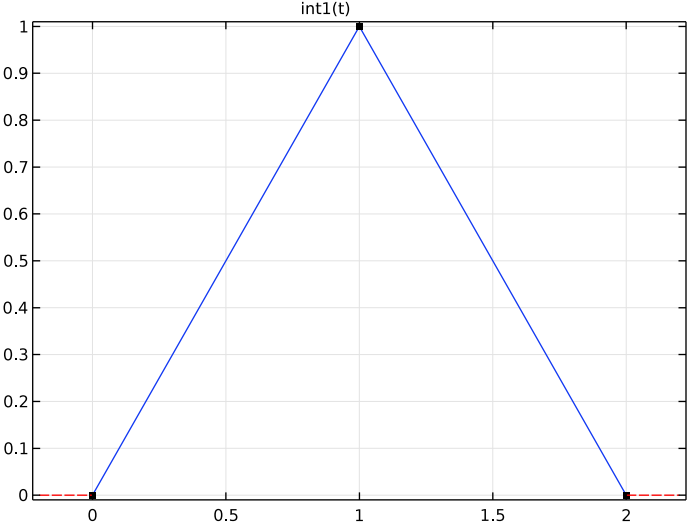


Figure 1: Boundary load for the first and third studies.

The second study runs a parametric sweep on the axial displacement with a constant temperature of 298 K. The prescribed displacement is applied on one face in order to reach an axial strain value of 0.07. After reaching this maximum value, the axial displacement is decreased down to 40% of its value, then increased up to 80% the maximum, to finally

decrease to 0 (Figure 2).

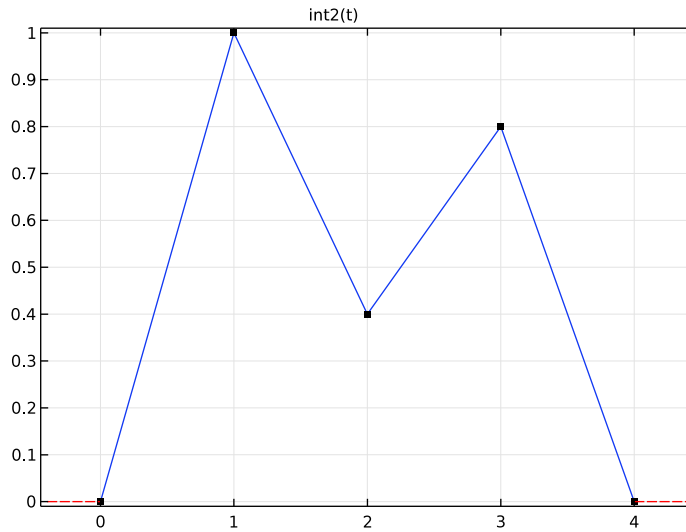


Figure 2: Prescribed displacement for the second study.

The third study runs at a constant temperature of 260 K, which is below the austenite start temperature. After one mechanical cycle (Figure 1), the temperature is uniformly increased to 300 K to achieve the reverse transformation to austenite (Figure 3).

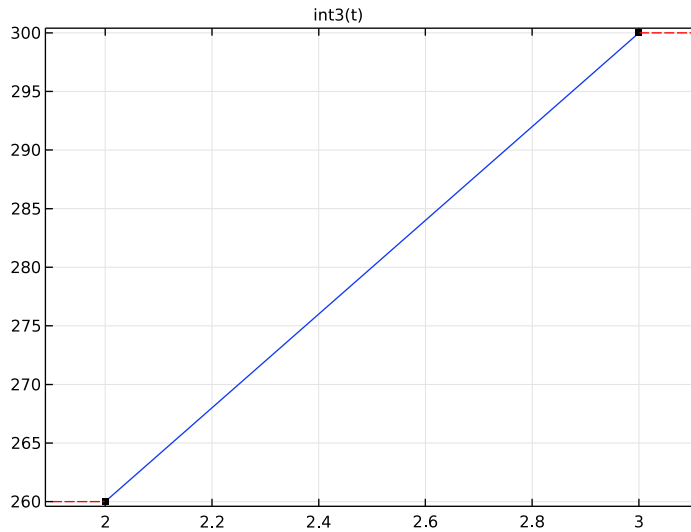


Figure 3: Temperature increase in the third study.

## Results and Discussion

---

### PARAMETRIC STUDY ON TEMPERATURE

In Figure 4, the curves for the high temperatures (328 K and 308 K) show the pseudoelasticity effect: the stress-strain relation is linear up to a temperature-dependent stress limit. Above this limit the martensite transformation begins, resulting in a material with lower stiffness. The transformation is complete when reaching the maximum strain; then the alloy microstructure is 100% martensite, and the tangent stiffness is taken from the martensite Young's modulus. During unloading, the reverse transformation occurs at a lower stress level than the stress limit for the forward transformation.

The axial stress-strain curve for a prescribed temperature of 276 K (between the austenite start and finish temperatures) shows that the forward and reverse transformations occur at lower stress levels. Also, the reverse transformation is not complete when the stress is completely released. This can be seen in Figure 5 where the forward transformation occurs sooner, that is, at lower stress level. The backward transformation starts later, and it is not complete at the end of the sweep.

At a lower temperature, the forward transformation occurs at a lower stress level, and the reverse transformation does not even start, resulting in residual strains.

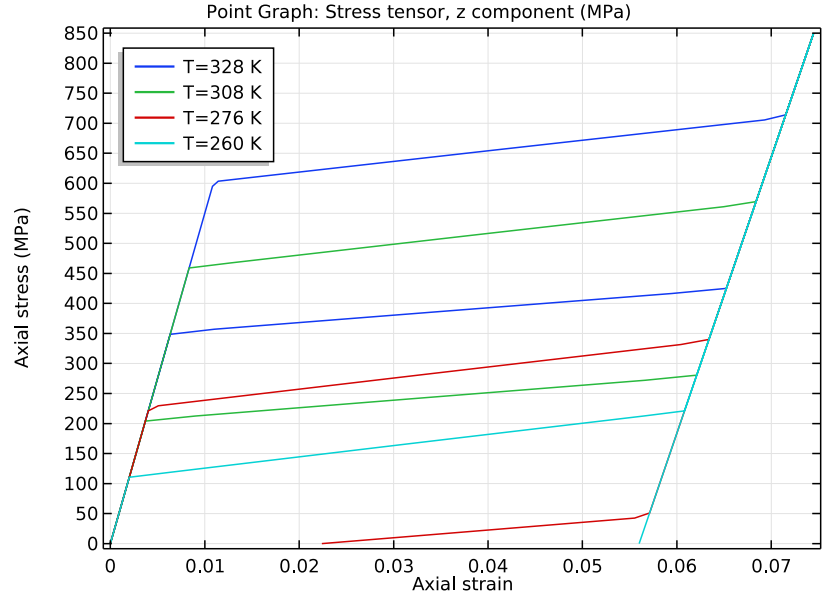


Figure 4: Stress vs strain curve at several temperatures.

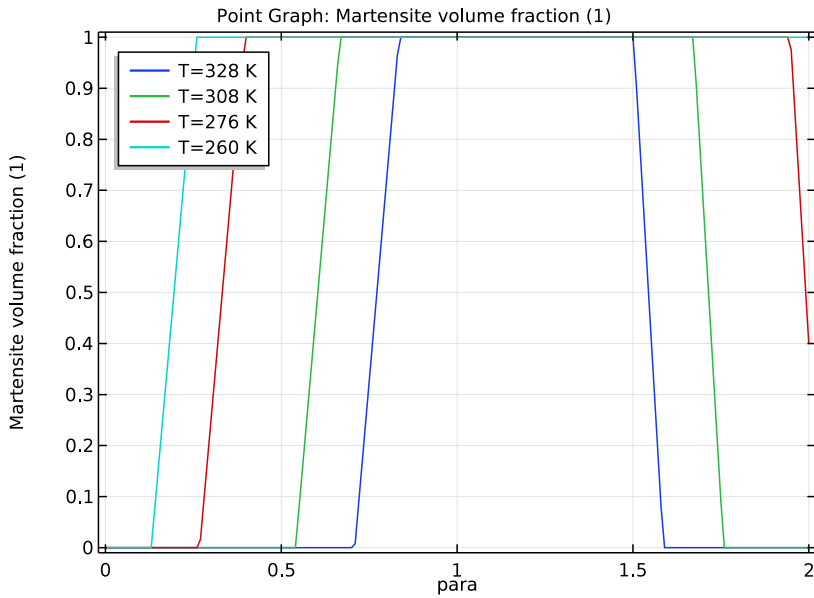


Figure 5: Evolution of martensite volume fraction at several temperatures

#### PARTIAL TRANSFORMATION

The first loading step and the first unloading step are similar to the ones in the previous study. At a strain value of 0.028, the material is loaded again, which leads to a stress increase with high stiffness (Figure 6). This part of the curve is represented by a plateau in Figure 7. When the yield stress limit is reached, the forward transformation continues, following the same path as the first load. When the material is unloaded, the stress decreased, and the material undergoes the reverse transformation to pure austenite. Finally the stress is decreased to zero.

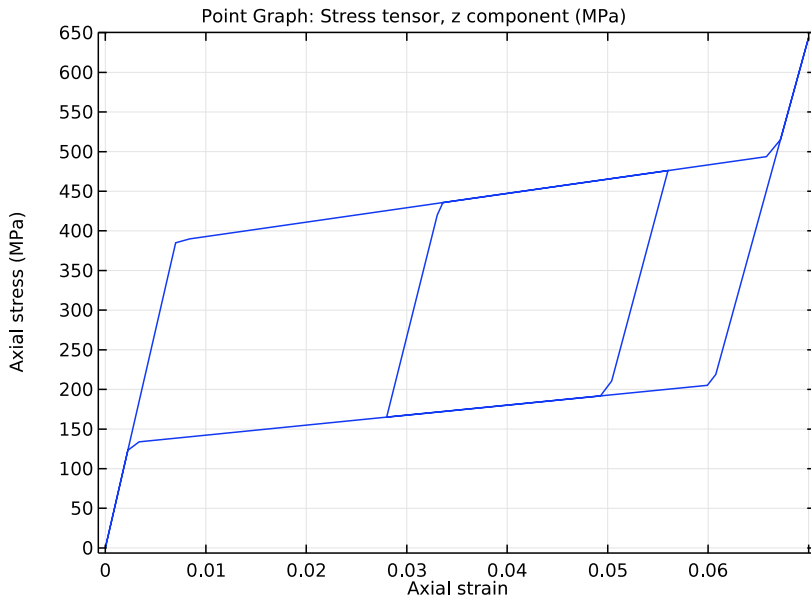
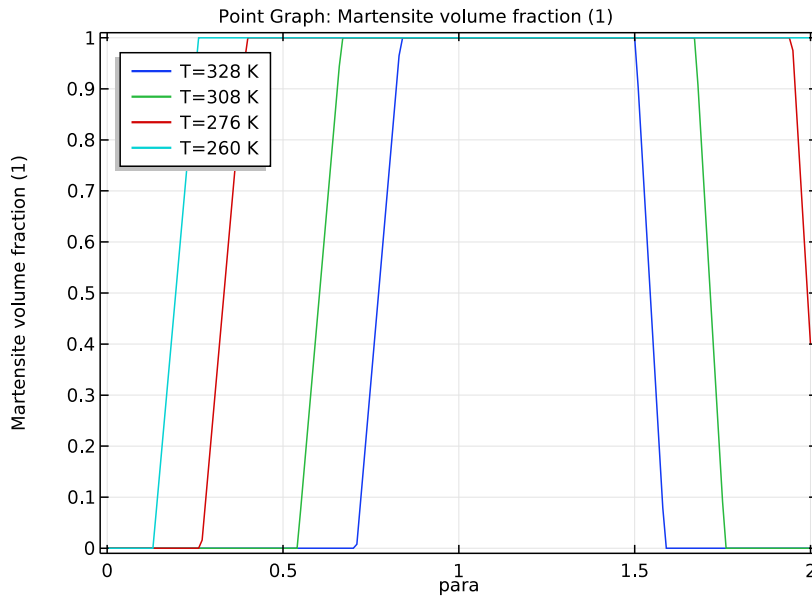


Figure 6: Stress vs strain curve for loading-unloading cycle with an internal loop.



*Figure 7: Evolution of martensite volume fraction during loading-unloading cycle with an internal loop.*

#### **SHAPE MEMORY EFFECT**

The mechanical loading-unloading cycle is the same as in the first study (260 K). At this point the residual strain is 0.056. Increasing the temperature from 260 K to 300 K decreases the residual strain to 0.

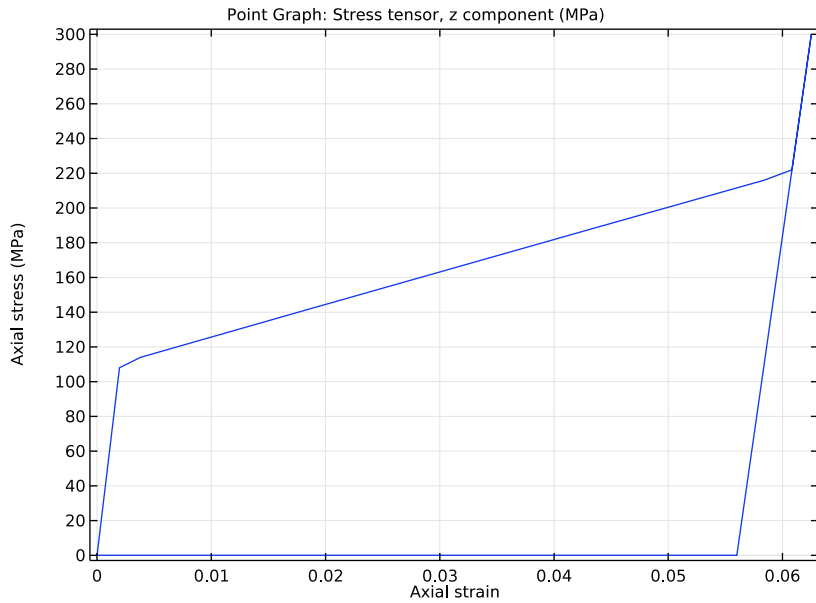


Figure 8: Stress vs strain curve showing the shape memory effect

## References

1. D.C. Lagoudas, ed., *Shape Memory Alloys*, Springer 2008

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Shape\_Memory\_Alloys/uniaxial\_loading\_of\_shape\_memory\_alloy

---

## 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 Axisymmetric**.
- 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 **Preset Studies>Stationary**.
- 6 Click **Done**.

## GLOBAL DEFINITIONS

### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
T	298[K]	298 K	Applied temperature
para	0	0	Continuation parameter

## GEOMETRY 1

- 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 **cm**.

### *Rectangle 1 (r1)*

- 1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 6.
- 4 In the **Height** text field, type 20.

## 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**, type Austenite in the **Label** text field.

### Material 2 (mat2)

- 1 Right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Martensite in the **Label** text field.

## SOLID MECHANICS (SOLID)

### Shape Memory Alloy 1

- 1 On the **Physics** toolbar, click **Domains** and choose **Shape Memory Alloy**.
- 2 In the **Settings** window for **Shape Memory Alloy**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **All domains**.
- 4 Locate the **Model Input** section. In the  $T$  text field, type T.
- 5 Locate the **Shape Memory Alloy** section. In the  $T_0$  text field, type 318.
- 6 Find the **Austenite** subsection. From the list, choose **Austenite (mat1)**.
- 7 Find the **Martensite** subsection. From the list, choose **Martensite (mat2)**.
- 8 Find the **Phase transformation parameters** subsection. In the  $M_s$  text field, type 245.
- 9 In the  $M_f$  text field, type 230.
- 10 In the  $C_M$  text field, type 7.4e6.
- 11 In the  $A_s$  text field, type 270.
- 12 In the  $A_f$  text field, type 280.
- 13 In the  $C_A$  text field, type 7.4e6.
- 14 In the  $\epsilon_{tr, max}$  text field, type 0.056.

Before adding the loads and boundary conditions, enter the material properties for austenite and martensite.

## MATERIALS

### Austenite (mat1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Austenite (mat1)**.
- 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
Poisson's ratio	nu	0.33	1	Basic
Density	rho	6500	kg/m <sup>3</sup>	Basic

Property	Variable	Value	Unit	Property group
Young's modulus	E	55 [GPa]	Pa	Basic
Heat capacity at constant pressure	Cp	400	J/(kg·K)	Basic

#### *Martensite (mat2)*

- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Materials** click **Martensite (mat2)**.
- 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
Young's modulus	E	46 [GPa]	Pa	Basic
Heat capacity at constant pressure	Cp	400	J/(kg·K)	Basic

### **SOLID MECHANICS (SOLID)**

#### *Roller 1*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Roller**.
- 2 Select Boundary 2 only.

Create an interpolation function to apply load cycle.

### **DEFINITIONS**

#### *Interpolation 1 (int1)*

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the table, enter the following settings:

t	f(t)
0	0
1	1
2	0

- 4 Click **Plot**.

## SOLID MECHANICS (SOLID)

### Boundary Load 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 Specify the  $\mathbf{F}_A$  vector as

0	r
850[MPa]*int1(para)	z

## MESH 1

### Size

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Coarser**.
- 4 Click **Build All**.

## STUDY 1

### Parametric Sweep

- 1 On the **Study** toolbar, click **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
- 3 Click **Add**.
- 4 Click to select row number 1 in the table.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
T	328 308 276 260	K

### Step 1: Stationary

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.
- 3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Continuation parameter)	range (0, 0.01, 2)	

#### *Solution 1 (sol1)*

- 1 On the **Study** toolbar, click **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
- 3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.
- 4 In the **Settings** window for **Parametric**, click to collapse the **Continuation** section.
- 5 Click to expand the **Continuation** section. Select the **Tuning of step size** check box.
- 6 In the **Model Builder** window, click **Study 1**.
- 7 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 8 Clear the **Generate default plots** check box.
- 9 On the **Study** toolbar, click **Compute**.

## RESULTS

#### *ID Plot Group 1*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Stress vs Strain (Parametric) in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 1 / Parametric Solutions 1 (sol2)**.

#### *Point Graph 1*

- 1 Right-click **Stress vs Strain (Parametric)** and choose **Point Graph**.
- 2 Select Point 4 only.
- 3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Stress>Stress tensor (spatial frame)>solid.sz - Stress tensor, z component**.
- 4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.
- 5 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Strain>Strain tensor (material and geometry frames)>solid.eZZ - Strain tensor, ZZ component**.

- 6 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Point** check box.
- 7 Select the **Show legends** check box.

#### *Stress vs Strain (Parametric)*

- 1 In the **Model Builder** window, under **Results** click **Stress vs Strain (Parametric)**.
- 2 In the **Settings** window for **ID 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 Axial strain.
- 6 In the **y-axis label** text field, type Axial stress (MPa).
- 7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.
- 8 On the **Stress vs Strain (Parametric)** toolbar, click **Plot**.  
The resulting plot should look like that in [Figure 4](#).

#### *ID Plot Group 2*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Martensite Volume Fraction (Parametric) in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study I/ Parametric Solutions I (sol2)**.

#### *Point Graph 1*

- 1 Right-click **Martensite Volume Fraction (Parametric)** and choose **Point Graph**.
- 2 Select Point 4 only.
- 3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics> Shape memory alloys>solid.xi\_M - Martensite volume fraction**.
- 4 Locate the **Legends** section. Select the **Show legends** check box.
- 5 Find the **Include** subsection. Clear the **Point** check box.

#### *Martensite Volume Fraction (Parametric)*

- 1 In the **Model Builder** window, under **Results** click **Martensite Volume Fraction (Parametric)**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.
- 3 From the **Position** list, choose **Upper left**.

4 On the **Martensite Volume Fraction (Parametric)** toolbar, click **Plot**.

The resulting plot should look like that in [Figure 5](#).

Now perform a study with a partial unloading-loading cycle. First define the load function.

#### DEFINITIONS

##### *Interpolation 2 (int2)*

1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the table, enter the following settings:

t	f(t)
0	0
1	1
2	0.4
3	0.8
4	0

4 Click **Plot**.

#### SOLID MECHANICS (SOLID)

##### *Prescribed Displacement 1*

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

5 In the  $u_{0z}$  text field, type  $20[\text{cm}] * 0.07 * \text{int2}(\text{para})$ .

#### ADD STUDY

1 On 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 **Preset Studies>Stationary**.

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

## STUDY 2

### Step 1: Stationary

- 1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.
- 2 In the **Settings** window for **Stationary**, click to expand the **Study extensions** section.
- 3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 4 Click **Add**.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Continuation parameter)	range (0, 0.02, 4)	

- 6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
- 7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 1**.
- 8 Click **Disable**.
- 9 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 10 Clear the **Generate default plots** check box.
- 11 On the **Home** toolbar, click **Compute**.  
Duplicate the existing plots to reproduce [Figure 6](#) and [Figure 7](#).

## RESULTS

### Stress vs Strain (Parametric) I

- 1 In the **Model Builder** window, under **Results** right-click **Stress vs Strain (Parametric)** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type **Stress vs Strain (Partial Loading-Unloading)** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 7 (sol7)**.
- 4 Locate the **Legend** section. Clear the **Show legends** check box.
- 5 On the **Stress vs Strain (Partial Loading-Unloading)** toolbar, click **Plot**.

### Martensite Volume Fraction (Parametric) I

- 1 In the **Model Builder** window, under **Results** right-click **Martensite Volume Fraction (Parametric)** and choose **Duplicate**.

- 2 In the **Settings** window for **ID Plot Group**, type Martensite Volume Fraction (Partial Loading-Unloading) in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 7 (sol7)**.
- 4 Locate the **Legend** section. Clear the **Show legends** check box.
- 5 On the **Martensite Volume Fraction (Partial Loading-Unloading)** toolbar, click **Plot**.

To show the shape memory effect you need to apply sweep on mechanical loading, then on temperature.

## DEFINITIONS

### *Interpolation 3 (int3)*

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the table, enter the following settings:

t	f(t)
2	260
3	300

- 4 Click **Plot**.

## SOLID MECHANICS (SOLID)

### *Shape Memory Alloy 1*

Duplicate the **Shape Memory Alloy 1** node to apply function-defined temperature.

### *Shape Memory Alloy 2*

- 1 In the **Model Builder** window, right-click **Shape Memory Alloy 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Shape Memory Alloy**, locate the **Model Input** section.
- 3 In the  $T$  text field, type `int3(para)`.

### *Boundary Load 2*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 3 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the  $\mathbf{F}_A$  vector as

0	r
300[MPa]*int1(para)	z

### ADD STUDY

- 1 On 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 **Preset Studies>Stationary**.
- 4 Click **Add Study** in the window toolbar.

### STUDY 3

#### *Step 1: Stationary*

- 1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.
- 2 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.
- 3 Select the **Auxiliary sweep** check box.
- 4 Click **Add**.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Continuation parameter)	range(0,0.02,2) range(2.05,0.05,3)	

- 6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
- 7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 1**.
- 8 Click **Disable**.
- 9 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Prescribed Displacement 1**.
- 10 Click **Disable**.
- 11 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 12 Clear the **Generate default plots** check box.
- 13 On the **Home** toolbar, click **Compute**.  
Duplicate the third plot to reproduce [Figure 8](#).

## RESULTS

### *Stress vs Strain (Partial Loading-Unloading) 1*

- 1 In the **Model Builder** window, under **Results** right-click **Stress vs Strain (Partial Loading-Unloading)** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type **Stress vs Strain (Shape Memory Effect)** in the **Label** text field.
- 3 Locate the **Data** section. From the **Data set** list, choose **Study 3/Solution 8 (sol8)**.
- 4 On the **Stress vs Strain (Shape Memory Effect)** toolbar, click **Plot**.

You may disable the following nodes in the studies if you want to run them again.

## STUDY 1

### *Step 1: Stationary*

- 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)>Prescribed Displacement 1**.
- 5 Click **Disable**.
- 6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Shape Memory Alloy 2**.
- 7 Click **Disable**.
- 8 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Boundary Load 2**.
- 9 Click **Disable**.

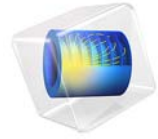
## STUDY 2

### *Step 1: Stationary*

- 1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Shape Memory Alloy 2**.
- 4 Click **Disable**.

- 5** In the **Physics and variables selection** tree, select **Component 1 (compl)**>  
**Solid Mechanics (solid)**>**Boundary Load 2**.
- 6** Click **Disable**.





# Viscoplastic Creep in Solder Joints

## Introduction

---

This example studies viscoplastic creep in solder joints under thermal loading using the Anand viscoplasticity model.

The Anand model is suitable for large, isotropic, viscoplastic deformations in combination with small elastic deformations. The following flow equation takes the stress dependence into account when evaluating strain rate

$$\dot{\epsilon}_p = A \exp\left(-\frac{Q}{RT}\right) \left[ \sinh\left(\xi \frac{\sigma}{s_a}\right) \right]^{\frac{1}{m}}$$

where  $\dot{\epsilon}_p$  is the inelastic strain rate,  $A$  is the pre-exponential factor,  $Q$  is the activation energy,  $m$  is the strain rate sensitivity,  $\xi$  is the multiplier of stress,  $R$  is the Boltzmann constant, and  $T$  is the absolute temperature.

The internal variable  $s_a$  is called deformation resistance, and it obeys the evolution equation

$$\dot{s}_a = \left\{ h_0 \left| 1 - \frac{s_a}{s^*} \right|^a \operatorname{sign}\left(1 - \frac{s_a}{s^*}\right) \right\} \dot{\epsilon}_p$$

where

$$s^* = s_0 \left[ \frac{\dot{\epsilon}_p}{A} \exp\left(\frac{Q}{RT}\right) \right]^n$$

is the saturation value of  $s$ ,  $h_0$  is the hardening constant,  $a$  is the strain rate sensitivity of hardening,  $s_0$  is the coefficient for deformation resistance saturation, and  $n$  is the strain rate sensitivity of deformation resistance.

## Model Definition

The model geometry is shown in Figure 1. It includes two electronic components (chips) mounted on a circuit board by means of several solder ball joints.

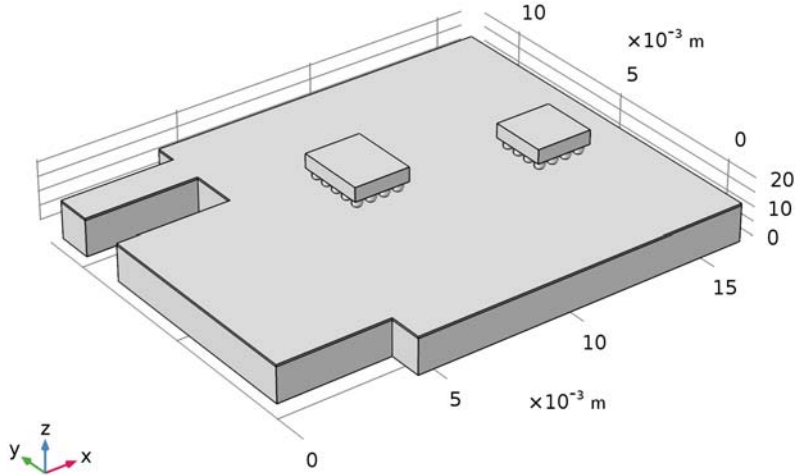


Figure 1: Model geometry.

The solder material is 60Sn40Pb. The circuit board consists of two layers: a thin layer of copper and a thicker layer of FR4 material. The chips are made of silicon. You can find the material and thermal properties for these three materials and for 60Sn40Pb in the material library available in COMSOL Multiphysics.

The nine material parameters needed to apply the Anand model for this solder are available in the literature (Ref. 1). They are summarized in the following table:

TABLE I: MODEL DATA FOR THE VISCOPLASTIC SOLDER JOINTS MODEL

PROPERTY	VALUE	DESCRIPTION
$A$	$1.49 \cdot 10^7$ 1/s	Pre-exponential factor
$Q/R$	10,830 K	Activation energy/Boltzmann constant
$m$	0.303	Strain rate sensitivity of stress
$n$	0.0231	Sensitivity for deformation resistance
$a$	1.34	Strain rate sensitivity of hardening

TABLE 1: MODEL DATA FOR THE VISCOPLASTIC SOLDER JOINTS MODEL

PROPERTY	VALUE	DESCRIPTION
$s_0$	80.42 MPa	Coefficient for deformation resistance saturation
$s_{init}$	56.33 MPa	Initial value of deformation resistance
$\xi$	11	Multiplier of stress
$h_0$	2640.75 MPa	Hardening constant

The structure has initially constant temperature  $T_0 = 20$  °C. The heat generation within the chips causes the thermal loading of the structure. At first both components is switched on and operates during 4 h generating a power of  $5 \cdot 10^7$  W/m<sup>3</sup>. Thereafter the both components are put at stand-by during 2 h where the power decreases to  $1 \cdot 10^7$  W/m<sup>3</sup>.

### Results and Discussion

When you study the results, bear in mind that the mesh used here is too coarse to produce converged and reliable results for the stresses and strains. The model serves only to display the principal features.

The temperature distribution after 4 h of operation is shown in [Figure 2](#). The temperature is at its maximum and the increase is about 50 degrees.

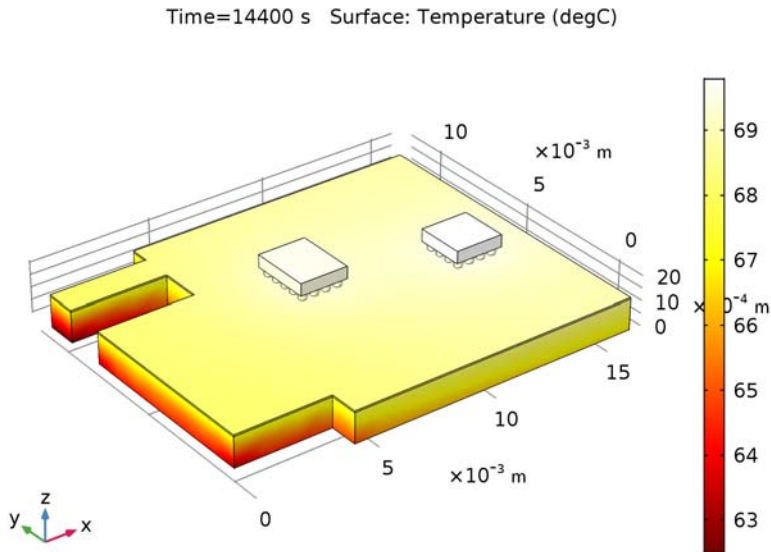


Figure 2: Final temperature distribution.

Figure 3 shows change in the deformation resistance through out the operating time.

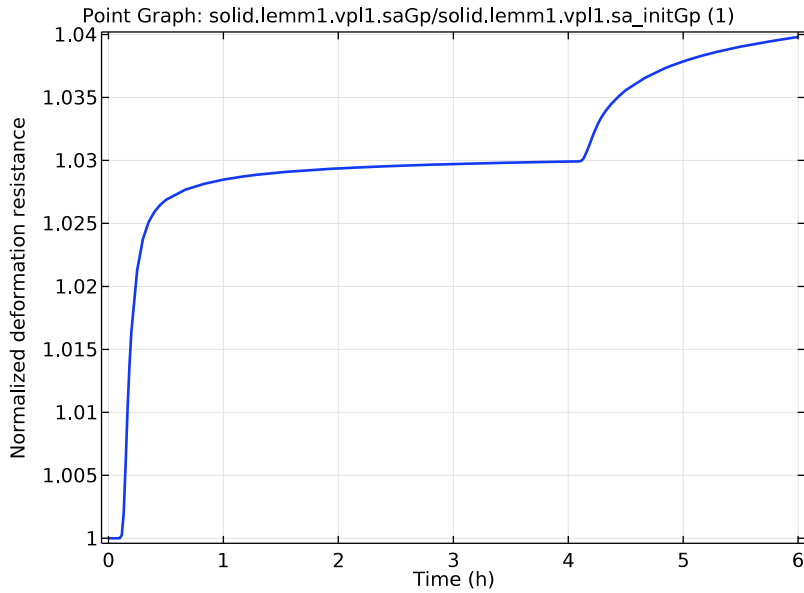


Figure 3: Hardening.

The development of elastic and inelastic strains at a point in a solder joint is shown in Figure 4. An intensive plastic flow appears after about 40 s of the loading and inelastic strains dominates after about 5 min. The smooth transition in the beginning of the load history and after 4 h is partly affected by the time dependent hardening behavior and partly affected by the smooth transition in the power load function where a Heaviside step is replaced with a smooth ramping over a 10 min period.

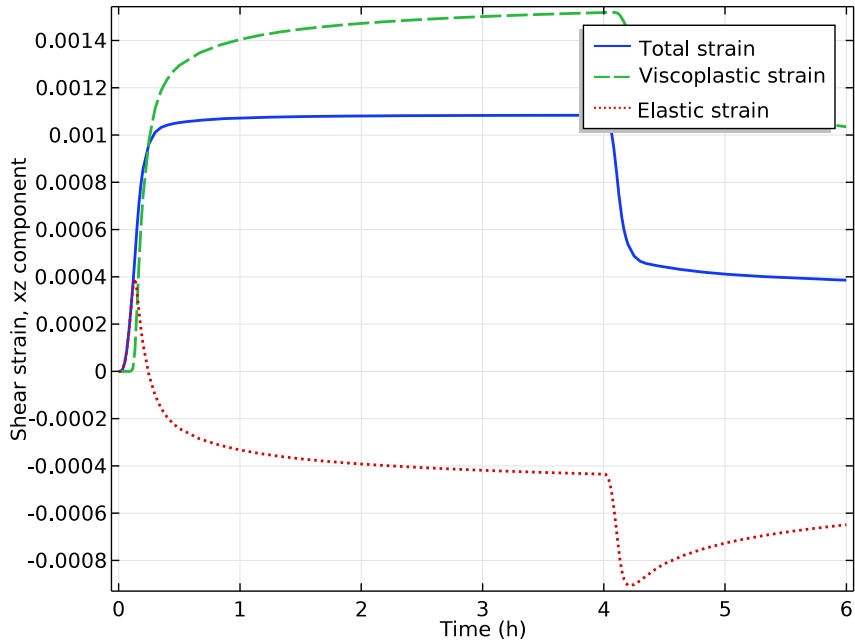
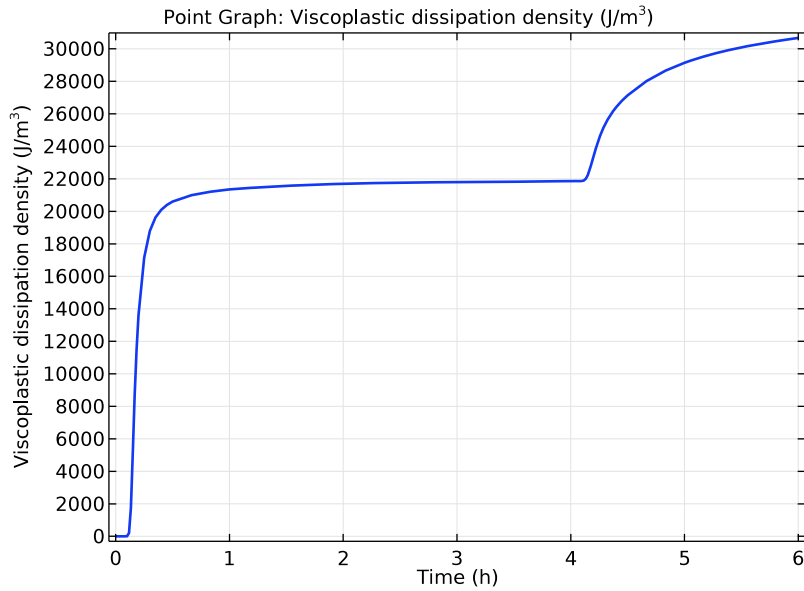


Figure 4: Shear strains in the most critical point.

In a model with creep or viscoplasticity, you can select to compute also the dissipated energy, as is shown in this example. This quantity is used in several fatigue evaluation criteria when dimensioning against thermal fatigue in electronic components. In Figure 5 the dissipated energy as function of time is shown for the same point as the graphs above.



*Figure 5: Creep energy dissipation density.*

### *Notes About the COMSOL Implementation*

---

In order to keep the model size down, the mesh is rather coarse see [Figure 6](#). The results in the solder balls are not accurate enough for making quantitative predictions. In reality, the best approach would probably be to first run a model of this type to find out which solder ball has the largest strains. In a second analysis, you can then analyze a model where an individual solder ball has an improved resolution.

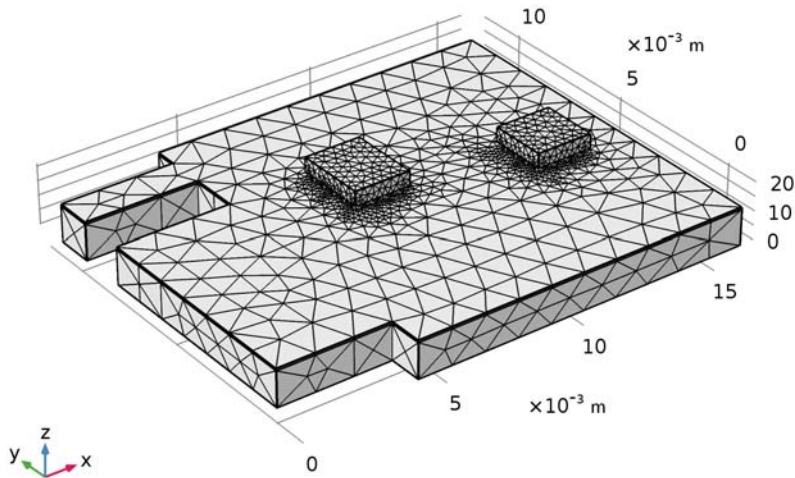


Figure 6: Meshed geometry.

### Reference

1. Z.N. Cheng, G.Z. Wang, L. Chen, J. Wilde, and K. Becker, “Viscoplastic Anand Model for Solder Alloys and its Application,” *Soldering & Surface Mount Technology*, vol. 12, no. 2, pp. 31–36, 2000.

---

**Application Library path:** Nonlinear\_Structural\_Materials\_Module/  
Viscoplasticity/viscoplastic\_solder\_joints

---

### 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 **3D**.
- 2 In the **Select Physics** tree, select **Structural Mechanics>Thermal Stress**.
- 3 Click **Add**.
- 4 Click **Study**.
- 5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.
- 6 Click **Done**.

## GLOBAL DEFINITIONS

### *Parameters*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
T0	20[degC]	293.15 K	Initial temperature

### *Analytic 1 (an1)*

- 1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.
- 2 In the **Settings** window for **Analytic**, type power in the **Function name** text field.
- 3 Locate the **Definition** section. In the **Expression** text field, type  $(f1c2hs(x-5*60,5*60)*5e7) - f1c2hs(x - (4*60*60+5*60),5*60)*4e7$ .
- 4 Locate the **Units** section. In the **Arguments** text field, type s.
- 5 In the **Function** text field, type  $W/m^3$ .

## GEOMETRY 1

### *Import 1 (imp1)*

- 1 On the **Home** toolbar, click **Import**.
- 2 In the **Settings** window for **Import**, locate the **Import** section.
- 3 Click **Browse**.
- 4 Browse to the model's Application Libraries folder and double-click the file `viscoplastic_solder_joints.mphbin`.
- 5 Click **Import**.

*Form Union (fin)*

On the **Home** toolbar, click **Build All**.

## **DEFINITIONS**

*Explicit 1*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type FR4 in the **Label** text field.
- 3 Select Domain 1 only.

*Explicit 2*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Copper in the **Label** text field.
- 3 Select Domain 2 only.

*Explicit 3*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Silicon in the **Label** text field.
- 3 Select Domains 3 and 24 only.

*Explicit 4*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Solder in the **Label** text field.
- 3 Locate the **Input Entities** section. Select the **All domains** check box.
- 4 Select Domains 4–23 and 25–40 only.

You can do this by first copying the text '4-23 and 25-40' and then clicking the **Paste Selection** button next to the **Selection** box or clicking in the box and pressing **Ctrl+V**.

*Solder 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **Solder** and choose **Duplicate**.
- 2 In the **Settings** window for **Explicit**, type Solder\_face in the **Label** text field.
- 3 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

*Explicit 6*

- 1 On the **Definitions** toolbar, click **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Symmetry Boundaries in the **Label** text field.

- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 17, 19, 182, and 183 only.

#### *Complement 1*

- 1 On the **Definitions** toolbar, click **Complement**.
- 2 In the **Settings** window for **Complement**, type Symmetry Complement in the **Label** text field.
- 3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
- 4 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.
- 5 In the **Add** dialog box, select **Symmetry Boundaries** in the **Selections to invert** list.
- 6 Click **OK**.

### **MULTIPHYSICS**

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Thermal Expansion 1 (te1)**.
- 2 In the **Settings** window for **Thermal Expansion**, locate the **Thermal Expansion Properties** section.
- 3 In the  $T_{\text{ref}}$  text field, type T0.

### **SOLID MECHANICS (SOLID)**

#### *Linear Elastic Material 1*

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

#### *Viscoplasticity 1*

- 1 On the **Physics** toolbar, click **Attributes** and choose **Viscoplasticity**.
- 2 In the **Settings** window for **Viscoplasticity**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Solder**.  
Add an equation for integrating the dissipated viscoplastic energy.
- 4 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

#### *Linear Elastic Material 1*

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

- 2 In the **Settings** window for **Linear Elastic Material**, click to expand the **Energy dissipation** section.
- 3 Locate the **Energy Dissipation** section. Select the **Calculate dissipated energy** check box.

#### *Symmetry I*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry Boundaries**.

#### *Prescribed Displacement I*

- 1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.
- 2 Select Point 193 only.
- 3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
- 4 Select the **Prescribed in z direction** check box.

### **HEAT TRANSFER IN SOLIDS (HT)**

#### *Initial Values I*

- 1 In the **Model Builder** window, under **Component 1 (comp 1)>Heat Transfer in Solids (ht)** click **Initial Values I**.
- 2 In the **Settings** window for **Initial Values**, type T0 in the  $T$  text field.
- 3 In the **Model Builder** window, click **Heat Transfer in Solids (ht)**.

#### *Symmetry I*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
- 2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry Boundaries**.

#### *Heat Source I*

- 1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.
- 2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Silicon**.
- 4 Locate the **Heat Source** section. In the  $Q_0$  text field, type power (t).

#### *Heat Flux I*

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.  
Heat flux applies on all exterior boundaries except those with prescribed symmetry.

- 2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Symmetry Complement**.
- 4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.
- 5 In the  $h$  text field, type 10.
- 6 In the  $T_{\text{ext}}$  text field, type T0.

#### **ADD MATERIAL**

- 1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Built-In>FR4 (Circuit Board)**.
- 4 Click **Add to Component** in the window toolbar.

#### **MATERIALS**

*FR4 (Circuit Board) (mat1)*

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 From the **Selection** list, choose **FR4**.

#### **ADD MATERIAL**

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-In>Copper**.
- 3 Click **Add to Component** in the window toolbar.

#### **MATERIALS**

*Copper (mat2)*

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 From the **Selection** list, choose **Copper**.

#### **ADD MATERIAL**

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-In>Silicon**.
- 3 Click **Add to Component** in the window toolbar.

## MATERIALS

*Silicon (mat3)*

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 From the **Selection** list, choose **Silicon**.

## ADD MATERIAL

- 1 Go to the **Add Material** window.
- 2 In the tree, select **Built-In>Solder, 60Sn-40Pb**.
- 3 Click **Add to Component** in the window toolbar.
- 4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

## MATERIALS

*Solder, 60Sn-40Pb (mat4)*

- 1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 2 From the **Selection** list, choose **Solder**.
- 3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Creep rate coefficient	A_ana	1.49e7	1/s	Anand viscoplasticity
Activation energy	Q_ana	90046	J/mol	Anand viscoplasticity
Multiplier of stress	xi_ana	11	1	Anand viscoplasticity
Stress sensitivity	m_ana	0.303	1	Anand viscoplasticity
Deformation resistance saturation coefficient	s0_ana	80.42 [MPa]	N/m <sup>2</sup>	Anand viscoplasticity
Deformation resistance initial value	sa_init	56.33 [MPa]	N/m <sup>2</sup>	Anand viscoplasticity
Hardening constant	h0_ana	2640.75 [MPa]	N/m <sup>2</sup>	Anand viscoplasticity
Hardening sensitivity	a_ana	1.34	1	Anand viscoplasticity
Deformation resistance sensitivity	n_ana	0.0231	1	Anand viscoplasticity

## MESH I

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

- 2 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.
- 3 From the **Sequence type** list, choose **User-controlled mesh**.

*Free Tetrahedral I*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.
- 2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Solder**.

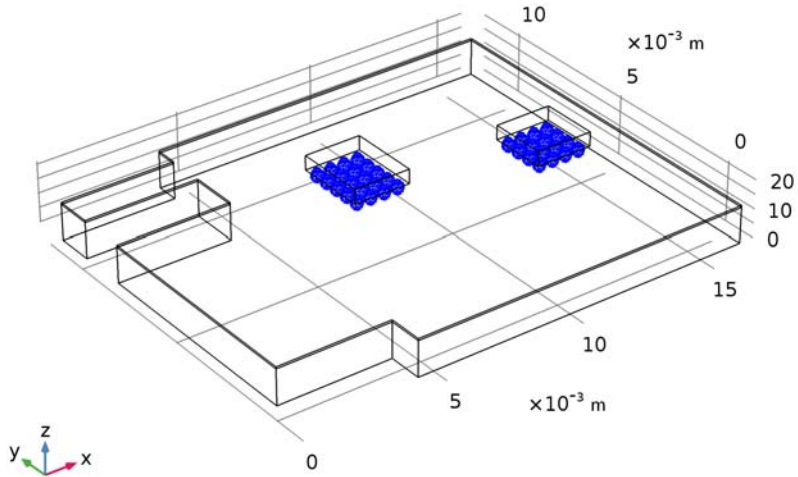
*Size*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Fine**.

*Free Tetrahedral I*

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.
- 2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

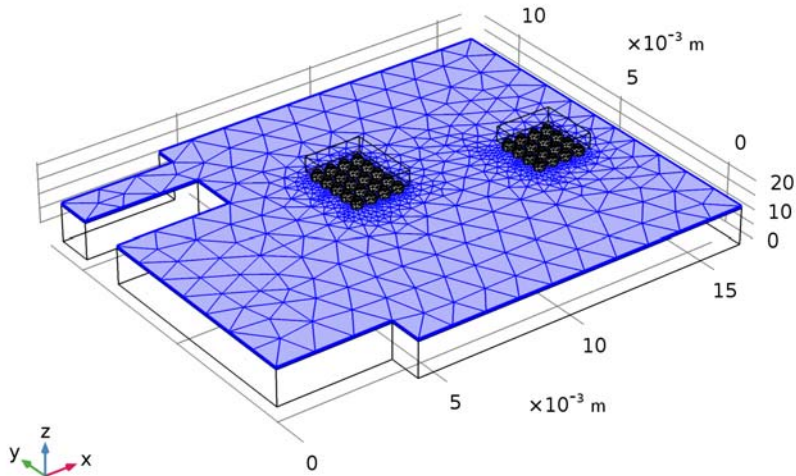
- 3 Click the **Wireframe Rendering** button on the **Graphics** toolbar to see the meshed domains.



#### *Free Triangular I*

- 1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations> Free Triangular**.
- 2 Select Boundary 7 only.

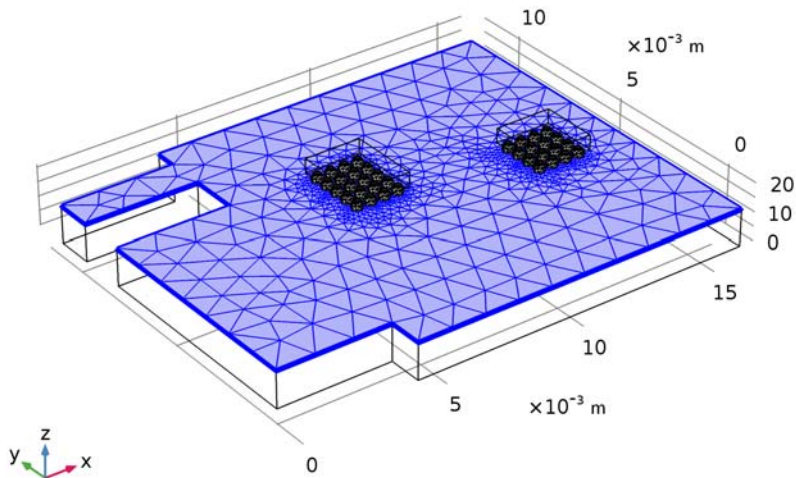
- 3 In the **Settings** window for **Free Triangular**, click **Build All**.



#### *Swept 1*

- 1 Right-click **Mesh 1** and choose **Swept**.
- 2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Copper**.

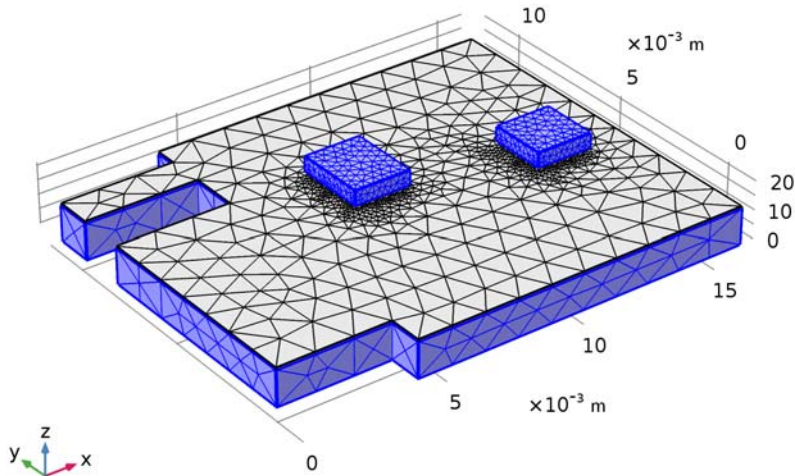
5 Click **Build All**.



*Free Tetrahedral 2*

- 1 Right-click **Mesh 1** and choose **Free Tetrahedral**.
- 2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

- 3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



## STUDY I

### Step 1: Time Dependent

The coupling only applies from Heat Transfer in Solids to Solid Mechanics. Solve Heat Transfer in Solids in a first time-dependent step and then Solid Mechanics in a second time-dependent step.

- 1 In the **Model Builder** window, expand the **Study I** node, then click **Step 1: Time Dependent**.
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 In the **Times** text field, type 0 20 60 range(2\*60, 60, 12\*60) range(15\*60, 3\*60, 30\*60) range(40\*60, 10\*60, 3.5\*60\*60) 3.7\*3600 3.9\*3600 4\*3600 4\*3600+20 4\*3600+60 range(4\*3600+2\*60, 60, 4\*3600+12\*60) range(4\*3600+15\*60, 3\*60, 4\*3600+30\*60) range(4\*3600+40\*60, 10\*60, 4\*3600+1.5\*60\*60) 4\*3600+1.7\*3600 4\*3600+1.9\*3600 6\*3600.
- 4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

### Step 2: Time Dependent 2

- 1 On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 In the **Times** text field, type  $0\ 20\ 60\ \text{range}(2*60,60,12*60)\ \text{range}(15*60,3*60,30*60)\ \text{range}(40*60,10*60,3.5*60*60)\ 3.7*3600\ 3.9*3600\ 4*3600\ 4*3600+20\ 4*3600+60\ \text{range}(4*3600+2*60,60,4*3600+12*60)\ \text{range}(4*3600+15*60,3*60,4*3600+30*60)\ \text{range}(4*3600+40*60,10*60,4*3600+1.5*60*60)\ 4*3600+1.7*3600\ 4*3600+1.9*3600\ 6*3600$ .
- 4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Heat Transfer in Solids (ht)** interface.
- 5 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
- 6 From the **Method** list, choose **Solution**.
- 7 From the **Study** list, choose **Study 1, Time Dependent**.
- 8 From the **Selection** list, choose **All**.

#### *Solution 1 (sol1)*

- 1 On the **Study** toolbar, click **Show Default Solver**.  
In cases where the time derivatives are not important as results, the file size can be significantly reduced by not storing these variables.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.
- 3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Output** section.
- 4 Clear the **Store time derivatives** check box.
- 5 Click to expand the **Time stepping** section. Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.
- 6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** click **Time-Dependent Solver 2**.
- 7 In the **Settings** window for **Time-Dependent Solver**, locate the **Output** section.
- 8 Clear the **Store time derivatives** check box.
- 9 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.  
The viscoplastic energy dissipation is not part of problem to be solved, but rather a result quantity to be computed. Set the solver to segregated and place the variable on its own segregated step. Changing this is not necessary, but it will reduce the memory requirements somewhat.

- 10 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 2** and choose **Segregated**.
- 11 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 2>Segregated 1** node, then click **Segregated Step**.
- 12 In the **Settings** window for **Segregated Step**, locate the **General** section.
- 13 In the **Variables** list, select **Viscoplastic dissipation density (comp1.solid.Wvp)**.
- 14 Under **Variables**, click **Delete**.
- 15 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 2** right-click **Segregated 1** and choose **Segregated Step**.
- 16 In the **Settings** window for **Segregated Step**, locate the **General** section.
- 17 Under **Variables**, click **Add**.
- 18 In the **Add** dialog box, select **Viscoplastic dissipation density (comp1.solid.Wvp)** in the **Variables** list.
- 19 Click **OK**.
- 20 On the **Study** toolbar, click **Compute**.

## RESULTS

### *Temperature (ht)*

- 1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Time (s)** list, choose **14400**.

### *Surface*

- 1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.
- 4 On the **Temperature (ht)** toolbar, click **Plot**.

Display the deformation resistance history.

### *ID Plot Group 4*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Deformation Resistance History in the **Label** text field.

### *Point Graph 1*

- 1 Right-click **Deformation Resistance History** and choose **Point Graph**.
- 2 Select Point 36 only.
- 3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type `solid.1emm1.vp11.saGp/  
solid.1emm1.vp11.sa_initGp`.
- 5 Locate the **x-Axis Data** section. From the **Unit** list, choose **h**.
- 6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type 2.

### *Deformation Resistance History*

- 1 In the **Model Builder** window, under **Results** click **Deformation Resistance History**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Plot Settings** section.
- 3 Select the **x-axis label** check box.
- 4 In the associated text field, type Time (h).
- 5 Select the **y-axis label** check box.
- 6 In the associated text field, type Normalized deformation resistance.
- 7 On the **Deformation Resistance History** toolbar, click **Plot**.

Display the strain history.

### *ID Plot Group 5*

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Strain History in the **Label** text field.

### *Point Graph 1*

- 1 Right-click **Strain History** and choose **Point Graph**.
- 2 Select Point 36 only.
- 3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Strain>Strain tensor, local coordinate system>solid.el13 - Strain tensor, local coordinate system, 13 component**.
- 4 Locate the **x-Axis Data** section. From the **Unit** list, choose **h**.
- 5 Locate the **Coloring and Style** section. In the **Width** text field, type 2.
- 6 Click to expand the **Legends** section. Select the **Show legends** check box.
- 7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

---

**Legends**

---

Total strain

---

*Point Graph 2*

- 1 Right-click **Results>Strain History>Point Graph 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Strain (Gauss points)>Viscoplastic strain tensor, local coordinate system>solid.evpGp13 - Viscoplastic strain tensor, local coordinate system, 13 component**.
- 3 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

---

Viscoplastic strain

---

- 4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

*Point Graph 3*

- 1 Right-click **Results>Strain History>Point Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type `solid.e113-solid.evpGp13`.
- 4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
- 5 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

---

Elastic strain

---

*Strain History*

- 1 In the **Model Builder** window, under **Results** click **Strain History**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Plot Settings** section.
- 3 Select the **x-axis label** check box.
- 4 In the associated text field, type `Time (h)`.
- 5 Select the **y-axis label** check box.
- 6 In the associated text field, type `Shear strain, xz component`.
- 7 Click to expand the **Title** section. From the **Title type** list, choose **None**.

**8** On the **Strain History** toolbar, click **Plot**.

Display the dissipation history.

#### *Deformation Resistance History I*

- 1** In the **Model Builder** window, under **Results** right-click **Deformation Resistance History** and choose **Duplicate**.
- 2** In the **Settings** window for **ID Plot Group**, type **Dissipation History** in the **Label** text field.

#### *Point Graph I*

- 1** In the **Model Builder** window, expand the **Results>Dissipation History** node, then click **Point Graph I**.
- 2** In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics>Energy and power>solid.WvpGp - Viscoplastic dissipation density**.

#### *Dissipation History*

- 1** In the **Model Builder** window, under **Results** click **Dissipation History**.
- 2** In the **Settings** window for **ID Plot Group**, locate the **Plot Settings** section.
- 3** Clear the **y-axis label** check box.
- 4** On the **Dissipation History** toolbar, click **Plot**.  
Finally, display the temperature history.

#### *Dissipation History I*

- 1** In the **Model Builder** window, right-click **Dissipation History** and choose **Duplicate**.
- 2** In the **Settings** window for **ID Plot Group**, type **Temperature History** in the **Label** text field.

#### *Point Graph I*

- 1** In the **Model Builder** window, expand the **Results>Temperature History** node, then click **Point Graph I**.
- 2** In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Heat Transfer in Solids>Temperature>T - Temperature**.
- 3** Locate the **y-Axis Data** section. From the **Unit** list, choose **degC**.
- 4** On the **Temperature History** toolbar, click **Plot**.



