Inflation of a Spherical Rubber Balloon

Introduction

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

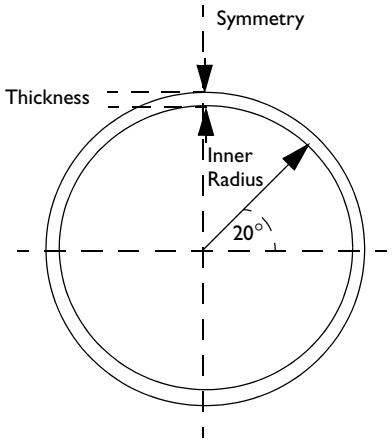


Figure 1: Model geometry. The initial inner radius is set to 10 cm, and the initial thickness to 1 mm.

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.

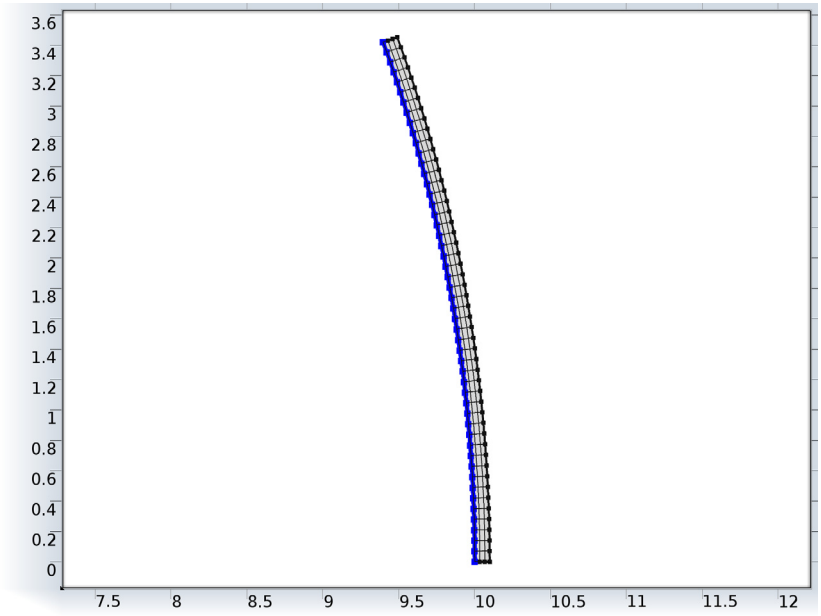


Figure 2: 2D axisymmetric geometry and mesh.

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 σ_θ, current inner radius r and current thickness h

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

For spherical balloons, the hoop stress σ_θ is equal to the largest principal stresses σ_1 and σ_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) $\lambda_3 = 1/\lambda^2 = h/H$, where h and H are the current and initial balloon thicknesses, respectively.

 ©2013 COMSOL

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 α_p and μ_p are Ogden parameters, and λ 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.

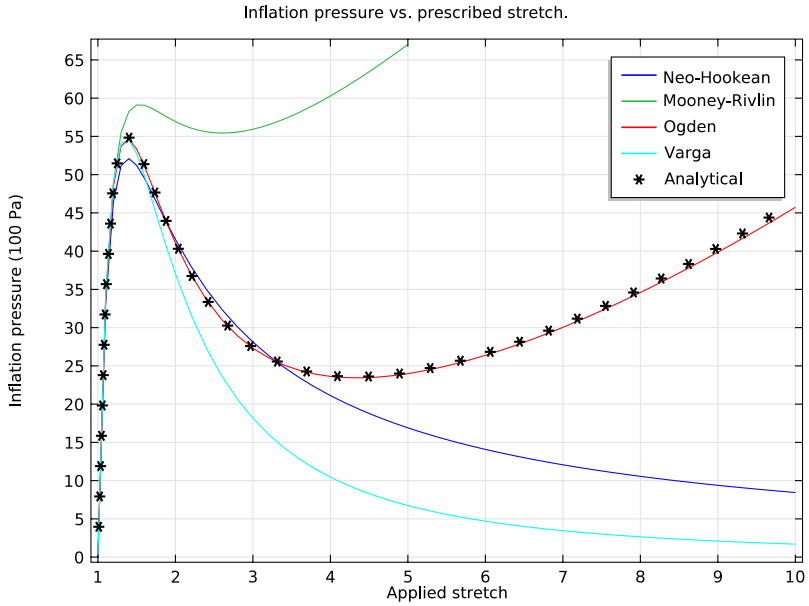


Figure 3: Computed inflation pressure as a function of circumferential stretch for different material models, compared to the analytical expression for Ogden material.

©2013 COMSOL

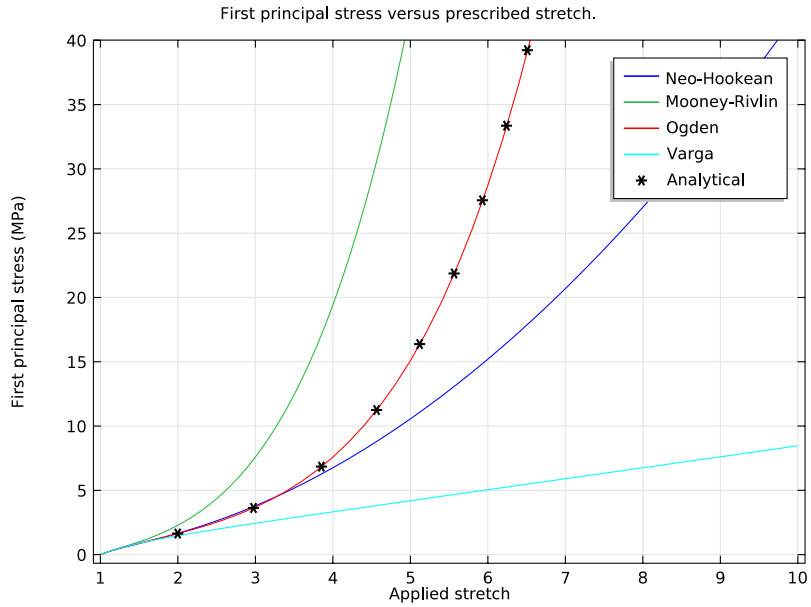


Figure 4: Computed hoop stress as a function of circumferential stretch for different material models, compared to the analytical expression for Ogden material.

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

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 $I_1(\overline{C}_{\mathrm{el}})$ and the initial bulk modulus κ

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

In this example, $\mu = 422.5$ kPa and $\kappa = 10^5\mu$. The Lamé parameter μ 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 $I_1(\overline{C}_{el})$ and $I_2(\overline{C}_{el})$, and the elastic volume ratio J_{el}

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

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 κ

$$W_s = \sum_{p=1}^{N} \frac{\mu_p}{\alpha_p}(\bar{\lambda}_{el1}^{\alpha_p} + \bar{\lambda}_{el2}^{\alpha_p} + \bar{\lambda}_{el3}^{\alpha_p} - 3) + \frac{1}{2}\kappa(J_{el} - 1)^2$$

with $N = 3$, and the Ogden parameters as written in Table 1

TABLE 1: OGDEN PARAMETERS

p	α_p	μ_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

$$W_s = 2\mu(\lambda_{el1} + \lambda_{el2} + \lambda_{el3} - 3) + \frac{1}{2}\kappa(J_{el} - 1)^2$$

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

 ©2013 COMSOL

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*, Wiley, 2000.

2. H. Azarnoush et al. 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. 59, pp. 697-705, 2012.

Model Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/balloon_inflation`

Modeling Instructions

MODEL WIZARD

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

2 Click the **2D axisymmetric** button.

3 Click **Next**.

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

5 Click **Next**.

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

7 Click **Finish**.

GLOBAL DEFINITIONS

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

Begin by defining model parameters.

Parameters

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

2 In the table, enter the following settings:

Name	Expression	Description
Ri	10[m]	Inner radius
H	0.1[m]	Thickness
mu	4.225e5[Pa]	Shear modulus
kappa	1e5*mu	Bulk modulus
stretch	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 Right-click **Global Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Description
u_appl	(stretch-1)*Ri	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

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

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

3 In the **Radius** edit field, type Ri+H.

4 In the **Sector angle** edit 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 the **Build All** button.

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

Delete Entities 1

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

 ©2013 COMSOL

2 In the **Delete Entities** settings window, 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 the **Build All** button.

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

SOLID MECHANICS

Add the four hyperelastic material models.

Hyperelastic Material 1

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

2 Select Domain 1 only.

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

4 Select the **Nearly incompressible material** check box.

5 In the κ edit field, type `kappa`.

6 From the μ list, choose **User defined**. In the associated edit field, type `mu`.

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

8 Go to the **Rename Hyperelastic Material** dialog box and type `Neo-Hookean` in the **New name** edit field.

9 Click **OK**.

Hyperelastic Material 2

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

2 Select Domain 1 only.

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

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

5 From the C_{10} list, choose **User defined**. In the associated edit field, type `0.4375*mu`.

6 From the C_{01} list, choose **User defined**. In the associated edit field, type `0.0625*mu`.

7 In the κ edit field, type `kappa`.

8 Right-click **Model 1>Solid Mechanics>Hyperelastic Material 2** and choose **Rename**.

9 Go to the **Rename Hyperelastic Material** dialog box and type `Mooney-Rivlin` in the **New name** edit field.

10 Click **OK**.

Hyperelastic Material 3

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

2 Select Domain 1 only.

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

4 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:

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

8 In the κ edit field, type `kappa`.

9 Right-click **Model 1>Solid Mechanics>Hyperelastic Material 3** and choose **Rename**.

10 Go to the **Rename Hyperelastic Material** dialog box and type `Ogden` in the **New name** edit field.

11 Click **OK**.

Hyperelastic Material 4

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

2 Select Domain 1 only.

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

4 From the **Material model** list, choose **User defined**.

5 Select the **Nearly incompressible material** check box.

6 In the W_{siso} edit field, type `2*mu*(solid.stchelp1+solid.stchelp2+solid.stchelp3-3)`.

7 In the W_{svol} edit field, type `0.5*kappa*(solid.Jel-1)^2`.

8 Right-click **Model 1>Solid Mechanics>Hyperelastic Material 4** and choose **Rename**.

9 Go to the **Rename Hyperelastic Material** dialog box and type `Varga` in the **New name** edit field.

10 Click **OK**.

Apply symmetry conditions.

 ©2013 COMSOL

Symmetry 1

1 Right-click **Solid Mechanics** and choose **Symmetry**.

2 Select Boundaries 1 and 2 only.

Control the inflation of the balloon by the pressure.

Boundary Load 1

1 Right-click **Solid Mechanics** and choose **Boundary Load**.

2 Select Boundary 3 only.

3 In the **Boundary Load** settings window, locate the **Force** section.

4 From the **Load type** list, choose **Pressure**.

5 In the *p* edit 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

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

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

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

4 Select Point 3 only.

Variables 2

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

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

3 In the table, enter the following settings:

Name	Expression	Description
ub	intop1(u)	Radial displacement, inner boundary

SOLID MECHANICS

1 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.

Global Equations 1

1 In the **Model Builder** window, under **Model 1** right-click **Solid Mechanics** and choose **Global>Global Equations**.

2 In the **Global Equations** settings window, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t)	Initial value (u_0)	Initial value (u_t0)
p_f	ub-u_appl	0	0

MESH 1

Mapped 1

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

Distribution 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

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

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

Distribution 2

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

2 Select Boundary 3 only.

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

4 In the **Number of elements** edit field, type 50.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

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

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

STUDY 1

Step 1: Stationary

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

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

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

 ©2013 COMSOL

4 In the **Physics and variables selection** tree, select **Model 1>Solid Mechanics>Mooney-Rivlin**, **Model 1>Solid Mechanics>Ogden**, and **Model 1>Solid Mechanics>Varga**.

5 Click **Disable**.

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

6 Click to expand the **Study Extensions** section. Select the **Continuation** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Continuation parameter	Parameter value list
stretch	`range(1, 0.1, 2) range(2.2, 0.2, 10)`

Modify the default solver to improve convergence.

Solver 1

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

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

2 In the **Model Builder** window, expand the **Solver 1** node, then click **Dependent Variables 1**.

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

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

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** node, then click **Direct**.

6 In the **Direct** settings window, locate the **General** section.

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

8 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** click **Parametric 1**.

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

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

11 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** click **Fully Coupled 1**.

12 In the **Fully Coupled** settings window, click to expand the **Method and Termination** section.

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

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

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

ROOT

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

MODEL WIZARD

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

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

3 Click **Finish**.

STUDY 2

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

2 In the **Study** settings window, locate the **Study Settings** section.

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

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Stationary** settings window, 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 **Model 1>Solid Mechanics>Neo-Hookean**, **Model 1>Solid Mechanics>Ogden**, and **Model 1>Solid Mechanics>Varga**.

5 Click **Disable**.

6 Locate the **Study Extensions** section. Select the **Continuation** check box.

7 Click **Add**.

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

8 In the table, enter the following settings:

Continuation parameter	Parameter value list
stretch	`range(1, 0.1, 5)`

 ©2013 COMSOL

Solver 2

1 In the **Model Builder** window, right-click **Study 2** and choose **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solver 2** node, then click **Dependent Variables 1**.

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

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

5 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solver 2>Stationary Solver 1** node, then click **Direct**.

6 In the **Direct** settings window, locate the **General** section.

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

8 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2>Stationary Solver 1** click **Parametric 1**.

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

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

11 In the **Model Builder** window, under **Study 2>Solver Configurations>Solver 2>Stationary Solver 1** click **Fully Coupled 1**.

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

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

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

ROOT

Continue with a third study for the Ogden material model.

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

MODEL WIZARD

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

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

3 Click **Finish**.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

2 In the **Study** settings window, locate the **Study Settings** section.

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

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3** click **Step 1: Stationary**.

2 In the **Stationary** settings window, 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 **Model 1>Solid Mechanics>Neo-Hookean**, **Model 1>Solid Mechanics>Mooney-Rivlin**, and **Model 1>Solid Mechanics>Varga**.

5 Click **Disable**.

6 Locate the **Study Extensions** section. Select the **Continuation** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Continuation parameter	Parameter value list
stretch	`range(1, 0.1, 2) range(2.2, 0.2, 10)`

Solver 3

1 In the **Model Builder** window, right-click **Study 3** and choose **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solver 3** node, then click **Dependent Variables 1**.

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

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

5 In the **Model Builder** window, expand the **Study 3>Solver Configurations>Solver 3>Stationary Solver 1** node, then click **Direct**.

6 In the **Direct** settings window, locate the **General** section.

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

8 In the **Model Builder** window, under **Study 3>Solver Configurations>Solver 3>Stationary Solver 1** click **Parametric 1**.

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

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

11 In the **Model Builder** window, under **Study 3>Solver Configurations>Solver 3>Stationary Solver 1** click **Fully Coupled 1**.

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

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

 ©2013 COMSOL

14 In the **Model Builder** window, right-click **Study 3** and choose **Compute**.

ROOT

Finally, add a study for a Varga material.

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

MODEL WIZARD

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

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

3 Click **Finish**.

STUDY 4

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

2 In the **Study** settings window, locate the **Study Settings** section.

3 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 **Stationary** settings window, 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 **Model 1>Solid Mechanics>Neo-Hookean**, **Model 1>Solid Mechanics>Mooney-Rivlin**, and **Model 1>Solid Mechanics>Ogden**.

5 Click **Disable**.

6 Locate the **Study Extensions** section. Select the **Continuation** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Continuation parameter	Parameter value list
stretch	`range(1, 0.1, 2) range(2.2, 0.2, 10)`

Solver 4

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

2 In the **Model Builder** window, expand the **Solver 4** node, then click **Dependent Variables 1**.

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

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

5 In the **Model Builder** window, expand the **Study 4>Solver Configurations>Solver 4>Stationary Solver 1** node, then click **Direct**.

6 In the **Direct** settings window, locate the **General** section.

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

8 In the **Model Builder** window, under **Study 4>Solver Configurations>Solver 4>Stationary Solver 1** click **Parametric 1**.

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

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

11 In the **Model Builder** window, under **Study 4>Solver Configurations>Solver 4>Stationary Solver 1** click **Fully Coupled 1**.

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

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

14 In the **Model Builder** window, right-click **Study 4** and choose **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 the one below.

RESULTS

Stress (solid)

1 In the **Model Builder** window, expand the **Results>Stress (solid)>Surface 1** node, then click **Deformation**.

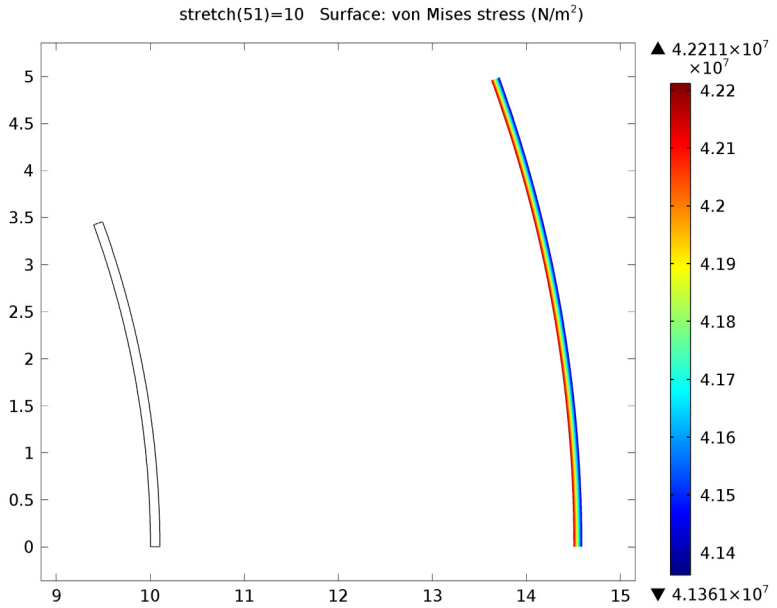
2 In the **Deformation** settings window, locate the **Scale** section.

3 In the **Scale factor** edit field, type `0.05`.

4 Click the **Plot** button.

 ©2013 COMSOL

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



Stress, 3D (solid)

The second default plot shows the von Mises stress in a 3D revolved plot.

To reproduce Figure 3 proceed as follows.

1D Plot Group 3

1 Right-click **Results** and choose **1D Plot Group**.

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

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

4 In the associated edit field, type `Applied stretch`.

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

6 In the associated edit field, type `Inflation pressure (100 Pa)`.

7 In the **1D Plot Group** settings window, click to expand the **Title** section.

8 From the **Title type** list, choose **Manual**.

9 In the **Title** text area, type `Inflation pressure vs. prescribed stretch.`.

10 Right-click **Results>1D Plot Group 3** and choose **Point Graph**.

11 In the **Point Graph** settings window, locate the **Data** section.

12 From the **Data set** list, choose **Solution 1**.

13 Select Point 3 only.

14 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Solid Mechanics>State variable p_f (p_f)**.

15 Locate the **y-Axis Data** section. In the **Expression** edit field, type `p_f/100`.

16 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Definitions>Applied stretch (stretch)**.

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

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

19 In the table, enter the following settings:

Legends
Neo-Hookean

20 Click the **Plot** button.

1 Right-click **Results>1D Plot Group 3>Point Graph 1** and choose **Duplicate**.

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

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

4 In the **Point Graph** settings window, locate the **Legends** section.

5 In the table, enter the following settings:

Legends
Mooney-Rivlin

6 Click the **Plot** button.

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

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

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

Legends
Ogden

5 Click the **Plot** button.

6 Right-click **Point Graph 1** and choose **Duplicate**.

7 In the **Point Graph** settings window, locate the **Data** section.

 ©2013 COMSOL

8 From the **Data set** list, choose **Solution 4**.

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

Legends
Varga

10 Click the **Plot** button.

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 In the **Expression** edit 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. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

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

7 In the **Number** edit field, type `40`.

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

Legends
Analytical

9 Click the **Plot** button.

To reproduce Figure 4 proceed as follows.

1D Plot Group 4

1 In the **Model Builder** window, right-click **1D Plot Group 3** and choose **Duplicate**.

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

3 In the **Title** text area, type `First principal stress versus prescribed stretch.`.

4 Locate the **Plot Settings** section. In the **y-axis label** edit field, type `First principal stress (MPa)`.

5 Click to expand the **Axis** section. Select the **Manual axis limits** check box.

6 In the **y maximum** edit field, type `40`.

7 In the **Model Builder** window, expand the **1D Plot Group 4** node, then click **Point Graph 1**.

8 In the **Point Graph** settings window, locate the **y-Axis Data** section.

9 In the **Expression** edit field, type `solid.sp1`.

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

11 In the **Model Builder** window, under **Results>1D Plot Group 4** click **Point Graph 2**.

12 In the **Point Graph** settings window, locate the **y-Axis Data** section.

13 In the **Expression** edit field, type `solid.sp1`.

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

15 In the **Model Builder** window, under **Results>1D Plot Group 4** click **Point Graph 3**.

16 In the **Point Graph** settings window, locate the **y-Axis Data** section.

17 In the **Expression** edit field, type `solid.sp1`.

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

19 In the **Model Builder** window, under **Results>1D Plot Group 4** click **Point Graph 4**.

20 In the **Point Graph** settings window, locate the **y-Axis Data** section.

21 In the **Expression** edit field, type `solid.sp1`.

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

23 In the **Model Builder** window, under **Results>1D Plot Group 4** click **Point Graph 5**.

24 In the **Point Graph** settings window, locate the **y-Axis Data** section.

25 In the **Expression** edit field, type
`((6.3e5[Pa]*(stretch^(1.3)-stretch^(-2*1.3)))+(0.012e5[Pa]*(stret ch^(5)-stretch^(-2*5)))-(0.1e5[Pa]*(stretch^(-2)-stretch^(2*2))))`.

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

27 Click the **Plot** button.

 ©2013 COMSOL