Created in COMSOL Multiphysics 6.0

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. This example demonstrates such controlled inflation of balloon based on radial stretch.

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.

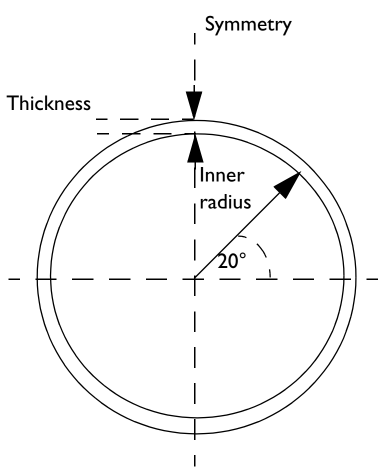

Figure 1: Model geometry. The initial inner radius is set to 10 cm, and the initial thickness to 1 mm.

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.

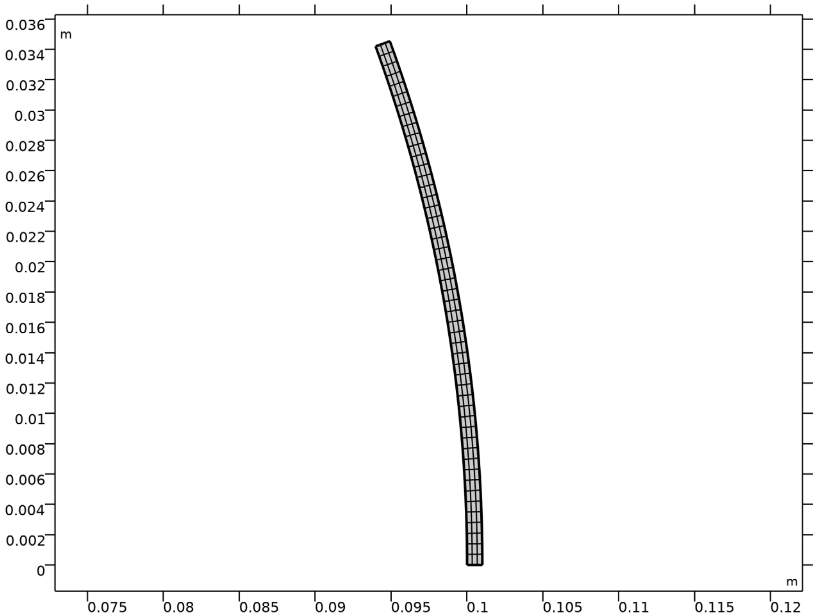

Figure 2: 2D axisymmetric geometry and mesh.

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 σ_θ, current inner radius r and current thickness h

$$p_{\mathrm{i}} = 2\frac{h}{r}\sigma_\theta$$

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 are in 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 α_p and μ_p are Ogden parameters, and λ is the largest principal stretch.

Because $r = R\lambda$ 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

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

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. The local maximum and local minimum for pressure are called as limit points, where the sign of stiffness changes. Some material models, like Mooney-Rivlin and Ogden, can show more than one limit point. A snap-through phenomenon is observed for Ogden and Mooney-Rivlin material models. The neo-Hookean and Varga material models can only reproduce balloon inflations at small strain levels.

The computed inflation pressure and hoop stress as functions of the applied stretch are shown in Figure 3 and Figure 4, respectively. Both figures include the computed results for all different material models, and are in a excellent agreement with the results described in Ref. 1, page 241.

Figure 5 shows the distribution of the von Mises stress for a neo-Hookean material at the final step of the solution..

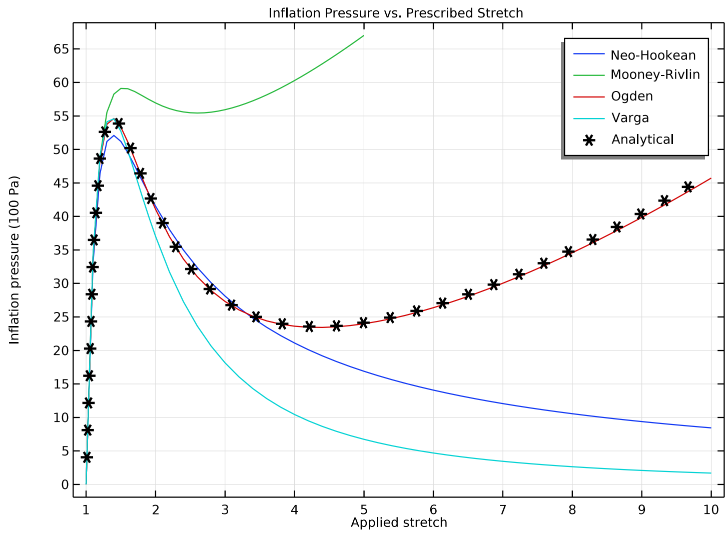

Figure 3: Computed inflation pressure as a function of circumferential stretch for different material models, compared to the analytical expression for the Ogden material model.

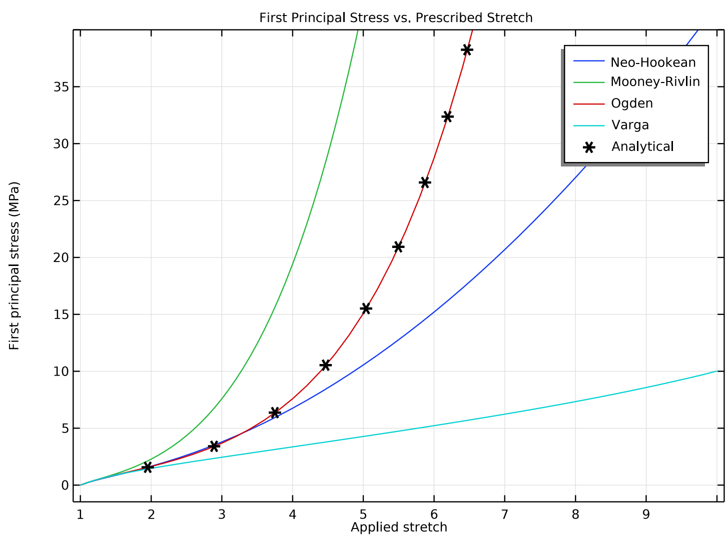

Figure 4: Computed hoop stress as a function of circumferential stretch for different material models, compared to the analytical expression for the Ogden material.

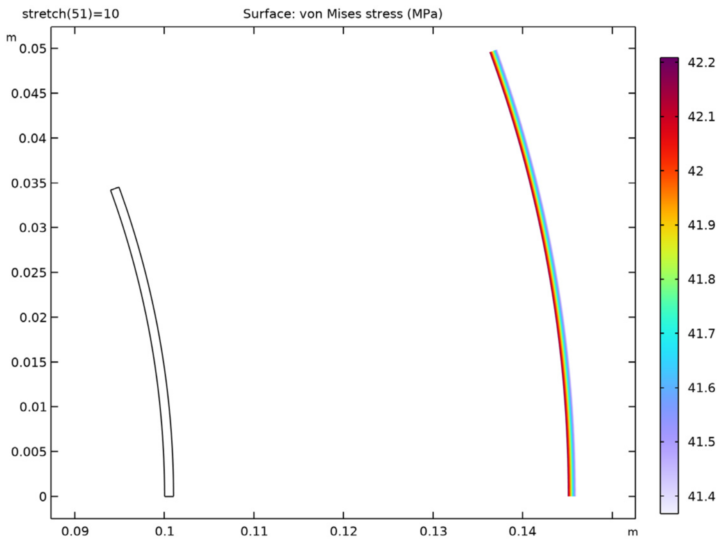

Figure 5: Distribution of von Mises stress on the modeled 2D cross section for the neo-Hookean material at maximum inflation.

Notes About the COMSOL Implementation

Different hyperelastic material models are constructed by specifying different elastic strain energy density 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 with quadratic volumetric strain energy formulation uses the isochoric invariant $I_1(\overline{C_{\mathrm{el}}})$ and the initial bulk modulus κ

$$W_{\mathrm{s}} = \frac{1}{2}\mu(\overline{I_1} - 3) + \frac{1}{2}\kappa(J_{\mathrm{el}} - 1)^2$$

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 with quadratic volumetric strain energy formulation has an elastic strain energy density written in terms of the two isochoric invariant of the elastic right Cauchy-Green deformation tensors $I_1(\overline{C_{\mathrm{el}}})$ and $I_2(\overline{C_{\mathrm{el}}})$, 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 with quadratic volumetric strain energy formulation 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 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 **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Begin by defining model parameters.

Parameters 1

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

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
Ri	10[cm]	0.1 m	Inner radius
H	1[mm]	0.001 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.

DEFINITIONS

Variables 1

1 In 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**
u_appl	(stretch-1)*Ri	m	Applied displacement

Use the applied stretch and the inner radius of the balloon to compute the applied displacement.

GEOMETRY 1

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

Circle 1 (c1)

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

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

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

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**.

SOLID MECHANICS (SOLID)

Add the four hyperelastic material models to be studied.

Neo-Hookean

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Material Models>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. From the **Compressibility** list, choose **Nearly incompressible material, quadratic volumetric strain energy**.

5 In the κ text field, type `kappa`.

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

Mooney-Rivlin

1 In 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 κ text field, type `kappa`.

Ogden

1 In 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 Click **Add** twice.

6 In the **Ogden parameters** table, enter the following settings:

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

7 In the κ text field, type `kappa`.

Varga

1 In 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 From the **Compressibility** list, choose **Nearly incompressible material**.

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

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

To enforce a symmetry constraint, add a **Roller** node.

Roller 1

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

2 Select Boundaries 1 and 2 only.

Control the inflation of the balloon by the pressure.

Boundary Load 1

1 In 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 a nonlocal integration coupling to evaluate the displacement at point 3.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal 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. From the **Frame** list, choose **Material (R, PHI, Z)**.

6 Clear the **Compute integral in revolved geometry** check box.

Variables 1

1 In the **Model Builder** window, 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
ub	intop1(u)	m	Radial displacement, inner boundary

SOLID MECHANICS (SOLID)

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

3 Click **OK** enable global equations and other advanced modeling features to the Solid Mechanics interface.

Global Equations 1

1 In 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) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
p_f	ub-u_appl	0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `pressure` in the text field.

6 Click **Filter**.

7 In the tree, select **General>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, type `displacement` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Displacement (m)**.

14 Click **OK**.

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

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 `3`.

Distribution 2

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

STUDY 1

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

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 tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Mooney-Rivlin**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Ogden**, and **Component 1 (Comp1)> Solid Mechanics (Solid)>Varga**.

5 Right-click and choose **Disable**.

Use an Auxiliary sweep to ramp up the applied stretch from 1.1 to 10.

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

2 Click **Add**.

3 In the table, enter the following settings:

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

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

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

Modify the default solver to improve convergence.

Solution 1 (sol1)

1 In the **Study** toolbar, click **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 **Solution 1 (sol1)** 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 **Neo-Hookean>Solver Configurations> Solution 1 (sol1)>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 **Neo-Hookean>Solver Configurations> Solution 1 (sol1)>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 **Neo-Hookean>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** click **Fully Coupled 1**.

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

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

14 In 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 In 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 **General Studies>Stationary**.

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

MOONEY-RIVLIN

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.

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 tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Neo-Hookean**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Ogden**, and **Component 1 (Comp1)> Solid Mechanics (Solid)>Varga**.

5 Right-click and choose **Disable**.

Use an Auxiliary sweep to ramp up the applied stretch from 1.1 to 5.

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

2 Click **Add**.

3 In the table, enter the following settings:

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

Solution 2 (sol2)

1 In 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 **Mooney-Rivlin>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 **Mooney-Rivlin>Solver Configurations>Solution 2 (sol2)>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 **Mooney-Rivlin>Solver Configurations>Solution 2 (sol2)>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 In 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 **General Studies>Stationary**.

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

OGDEN

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

2 In the **Settings** window for **Study**, type `Ogden` in the **Label** text field.

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

Step 1: Stationary

1 In the **Model Builder** window, under **Ogden** 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 tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Neo-Hookean**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Mooney-Rivlin**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>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 In 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 In the **Study** toolbar, click **Compute**.

Finally, add a fourth study for the Varga material model.

ADD STUDY

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

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

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

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

STUDY 4

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

2 In the **Settings** window for **Study**, 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 **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 tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Neo-Hookean**, **Component 1 (Comp1)>Solid Mechanics (Solid)>Mooney-Rivlin**, and **Component 1 (Comp1)>Solid Mechanics (Solid)>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)

1 In 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 **Varga>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 **Varga>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 **Varga>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 In 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 In the **Stress (solid)** toolbar, click **Plot**.

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

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

Inflation Pressure

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

2 In the **Settings** window for **1D 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 **Inflation Pressure** and choose **Point Graph**.

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

3 From the **Dataset** 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:

Legends
`Neo-Hookean`

10 In the **Inflation Pressure** toolbar, click **Plot**.

Point Graph 2

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

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

3 From the **Dataset** list, choose **Mooney-Rivlin/Solution 2 (sol2)**.

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

Legends
`Mooney-Rivlin`

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

Point Graph 3

1 In the **Model Builder** window, under **Results>Inflation Pressure** right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Odgen/Solution 3 (sol3)**.

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

Legends
`Ogden`

5 In 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 **Dataset** list, choose **Varga/Solution 4 (sol4)**.

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

Legends
`Varga`

5 In 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. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

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 In the **Inflation Pressure** toolbar, click **Plot**.

To reproduce Figure 4, proceed as follows.

First Principal Stress

1 In the **Model Builder** window, right-click **Inflation Pressure** and choose **Duplicate**.

2 In the **Settings** window for **1D 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 vs. 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 **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, 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, 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, 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, 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[Pa]*(stretch^(1.3)-stretch^(-2*1.3)))+(0.012e5[Pa]*(stretch^(5)-stretch^(-2*5)))-(0.1e5[Pa]*(stretch^(-2)-stretch^(2*2))))`.

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

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