



Inflation of a Spherical Rubber Balloon — Shell and Membrane Version

Introduction

The purpose of this example is to illustrate how the Shell and Membrane interfaces can be used to model deformation in thin hyperelastic structures. The model is identical to the Application Library example, [Inflation of a Spherical Rubber Balloon](#), however, in this example, the Shell and Membrane interfaces are used instead of the Solid Mechanics interface.

When using the Membrane interface, the main difference is that it does not compute the variation across the thickness as the Solid Mechanics interface does, since the interface is based on the plane stress assumption for thin structures.

In contrast to the Membrane interface, the hyperelastic material in the Shell interface computes variations across the thickness direction by using the layered shell technology. Both the Membrane interface and the Shell interface share the same geometric dimension, but the Shell interface is a bit more computationally expensive compared to the Membrane interface due to the constitutive equation across the thickness and rotational degrees of freedom. However, since either the Membrane or the Shell interface are defined on a geometric dimension lower than a Solid Mechanics interface, the modeling with these interfaces is computationally more efficient.

All relevant details about the geometry and material parameters can be found in the [Inflation of a Spherical Rubber Balloon](#) example.

Results and Discussions

The results obtained with the Membrane and Shell interface are almost equivalent to the results from obtained in a Solid Mechanics interface.

[Figure 1](#) shows the distribution of von Mises stress for a neo-Hookean material in the Shell interface at maximum inflation. The stress varies from 41.5 MPa to 42.2 MPa across the thickness, which agrees with the results obtained with the Solid Mechanics interface, where the stress varies from 41.4 MPa to 42.2 MPa across the thickness.

[Figure 2](#) shows the distribution of von Mises stress for a neo-Hookean material in the Membrane interface at maximum inflation. A uniform stress through thickness of 41.83 MPa agrees well with the results obtained with the Solid Mechanics interface, where the stress varies from 41.4 MPa to 42.2 MPa across the thickness.

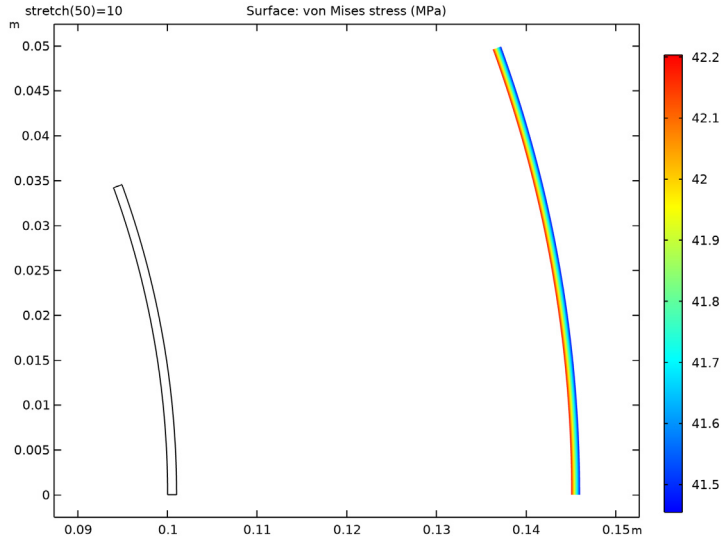


Figure 1: Distribution of von Mises stress for the neo-Hookean material in the shell interface at maximum inflation.

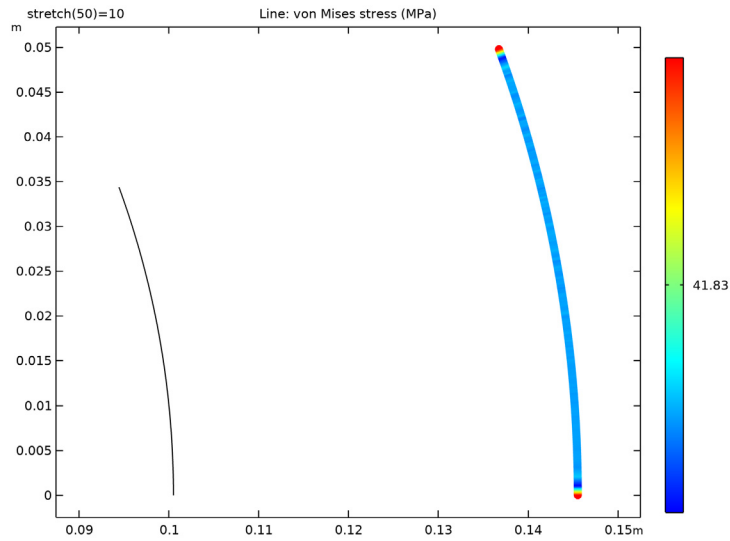


Figure 2: Distribution von Mises stress for the neo-Hookean material in the membrane interface at maximum inflation

The variation of inflation pressure versus applied stretch for different hyperelastic material models is shown in Figure 3. The inflation pressure in the Shell interface matches exactly the results shown in the Membrane interface. Also, the plot is identical to the results obtained with the Solid Mechanics interface for all material models.

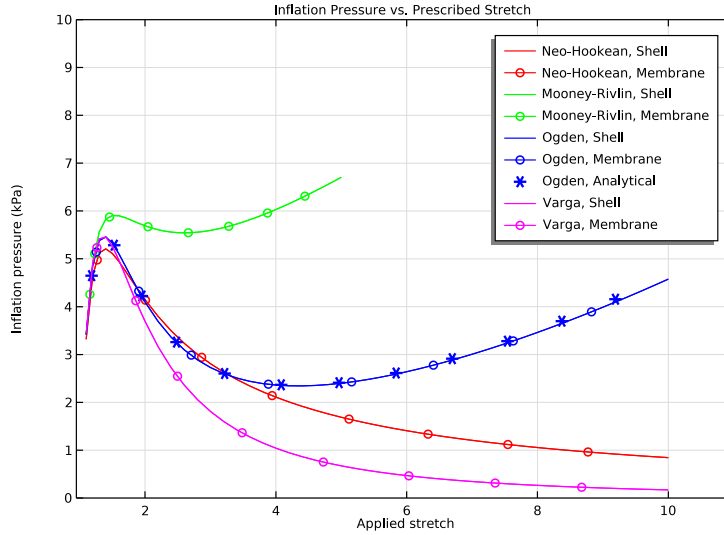


Figure 3: Computed inflation pressure as a function of circumferential stretch for different material models, compared to the analytical expression for the Ogden material.

The variation of hoop stress versus applied stretch for different hyperelastic material models is shown in Figure 4. The hoop stress in the Shell interface matches exactly the results from the Membrane interface. The plot is identical to the results obtained with the Solid Mechanics interface for all material models.

Figure 5 shows a comparison of the trough thickness deformation in the Shell, Membrane and Solid Mechanics interfaces. The results from Shell, Membrane and Solid Mechanics versions match, so the thinning of the balloon can be accurately captured using either the Shell or Membrane interfaces, thus saving computational cost.

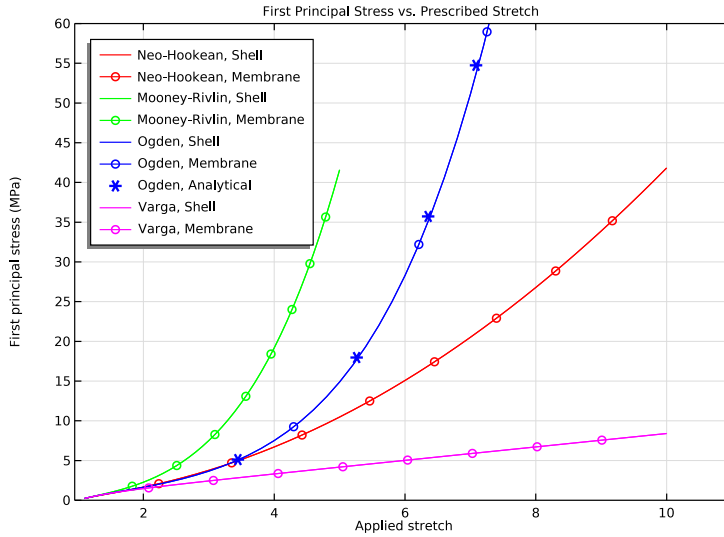


Figure 4: Computed hoop stress as a function of circumferential stretch for different material models, compared to the analytical expression for the Ogden material

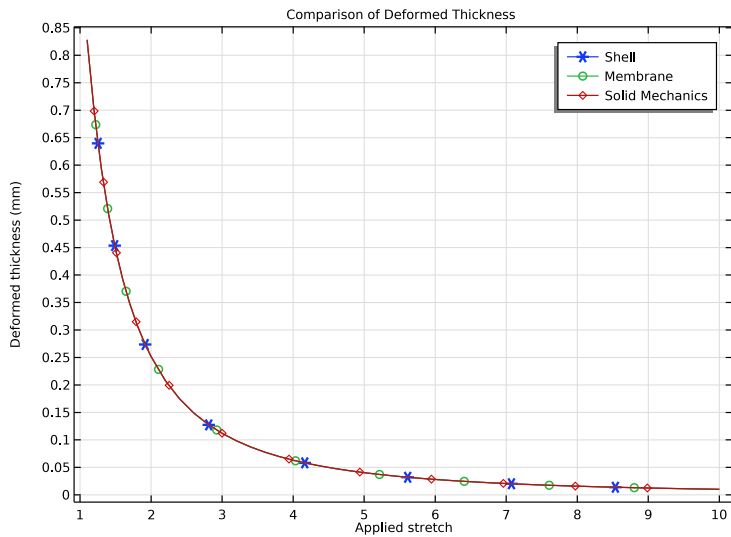


Figure 5: Comparison of the trough thickness deformation in the Shell, Membrane and Solid Mechanics interfaces.

Notes About the COMSOL Implementation

The absence of bending stiffness in the Membrane interface requires a pre-stretching step before solving the inflation step. A separate study is created to compute this step, and the results from this study are used as initial values for the inflation step.

Although the Shell interface does not need a pre-stretching step, this step is computed anyways for easier comparison with the results from the Membrane interface.

Application Library path: Nonlinear_Structural_Materials_Module/
Hyperelasticity/balloon_inflation_shell_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>Shell (shell)**.
- 3 Click **Add**.
- 4 In the **Select Physics** tree, select **Structural Mechanics>Membrane (mbrn)**.
- 5 Click **Add**.
- 6 Click  **Study**.
- 7 In the **Select Study** tree, select **General Studies>Stationary**.
- 8 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 nearly incompressible.

Create an interpolation function of deformed thickness versus stretch. The imported data was computed with the Solid Mechanics interface.

Interpolation 1 (int1)

1 In 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_shell_membrane_interpolation.txt`.

5 Locate the **Units** section. In the **Arguments** text field, type 1.

6 In the **Function** text field, type mm.

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 **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 In the **Settings** window for **Delete Entities**, click  **Build Selected**.

Add a **Single Layer Material** before adding a **Layered Hyperelastic Material** node in the shell interface.

MATERIALS

Hyperelastic Material

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Layers>Single Layer Material**.
- 2 In the **Settings** window for **Material**, type Hyperelastic Material in the **Label** text field.
- 3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1000	kg/m ³	Basic
Thickness	lth	H	m	Shell

SHELL (SHELL)

Neo-Hookean

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Shell (shell)** and choose **Material Models>Layered Hyperelastic Material**.
- 2 In the **Settings** window for **Layered Hyperelastic Material**, type Neo-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. From the **Compressibility** list, choose **Nearly incompressible material, quadratic volumetric strain energy**.

- 5 From the μ list, choose **User defined**. In the associated text field, type μ .
- 6 In the κ text field, type κ .

Mooney-Rivlin

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Layered Hyperelastic Material**.
- 2 In the **Settings** window for **Layered 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 κ text field, type κ .

Ogden

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Layered Hyperelastic Material**.
- 2 In the **Settings** window for **Layered 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 Click **Add** twice.
- 6 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

- 7 In the κ text field, type κ .

Varga

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Layered Hyperelastic Material**.
- 2 In the **Settings** window for **Layered 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 κ text field, type kappa.

To enforce a symmetry constraint, use **Prescribed Displacement** nodes. Add a rotated coordinate system to enforce the symmetry constraint at the top.

Prescribed Displacement/Rotation 1

- 1 In the **Physics** toolbar, click  **Points** and choose **Prescribed Displacement/Rotation**.
- 2 Select Point 2 only.
- 3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.
- 4 Select the **Prescribed in z direction** check box.
- 5 Locate the **Prescribed Rotation** section. From the **By** list, choose **Rotation**.

DEFINITIONS (COMPI)

Rotated System 2 (sys2)

- 1 In the **Definitions** toolbar, click  **Coordinate Systems** and choose **Rotated System**.
- 2 In the **Settings** window for **Rotated System**, locate the **Rotation** section.
- 3 In the **Rotation about out-of-plane axis** text field, type $20[\text{deg}]$.

SHELL (SHELL)

Prescribed Displacement/Rotation 2

- 1 In the **Physics** toolbar, click  **Points** and choose **Prescribed Displacement/Rotation**.
- 2 Select Point 1 only.
- 3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Coordinate System Selection** section.
- 4 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.
- 5 Locate the **Prescribed Displacement** section. Select the **Prescribed in x3 direction** check box.
- 6 Locate the **Prescribed Rotation** section. From the **By** list, choose **Rotation**.

Prescribe the displacement in the normal direction for the prestretch analysis.

Prescribed Displacement/Rotation 3

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Prescribed Displacement/Rotation**.

- 2 In the **Settings** window for **Prescribed Displacement/Rotation**, 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 1 (sys1)**.
- 5 Locate the **Prescribed Displacement** section. Select the **Prescribed in n direction** check box.
- 6 In the u_{0n} text field, type -1 [mm].
- 7 Locate the **Prescribed Rotation** section. From the **By** list, choose **Rotation**.

Control the inflation of the balloon by the pressure.

Face Load 1

- 1 In 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 pf_s .

Define the pressure pf_s using a **Global Equation** feature. First, define a nonlocal integration coupling to evaluate the displacement at point 2.

DEFINITIONS (COMPI)

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 2 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
us	intop1(u)	m	Radial displacement, shell

- 4 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.
- 6 Click **OK** to enable a global equations and other advanced modeling features to the Shell and Membrane interfaces.

SHELL (SHELL)

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	$f(u, ut, utt, t)$ (I)	Initial value (u_0) (I)	Initial value (u_t0) (I/s)	Description
pf_s	us-u_app1	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 length in the text field.
- 12 Click  **Filter**.
- 13 In the tree, select **General>Length (m)**.
- 14 Click **OK**.

MEMBRANE (MBRN)

Thickness and Offset I

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

Add the four hyperelastic material models.

Neo-Hookean

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Hyperelastic Material**.
- 2 In the **Settings** window for **Hyperelastic Material**, type Neo-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. From the **Compressibility** list, choose **Nearly incompressible material, quadratic volumetric strain energy**.
- 5 From the μ list, choose **User defined**. In the associated text field, type μ .
- 6 In the κ text field, type kappa.

Mooney-Rivlin

- 1 In 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 κ text field, type kappa.

Ogden

- 1 In 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 Click **Add** twice.

6 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

7 In the κ text field, type kappa.

Varga

- 1 In 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 $2 \cdot \mu$.
- 6 From the c_2 list, choose **User defined**. In the κ text field, type kappa.

Prescribed Displacement 1

- 1 In the **Physics** toolbar, click  **Points** and choose **Prescribed Displacement**.
- 2 Select Point 2 only.
- 3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
- 4 Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

- 1 In the **Physics** toolbar, click  **Points** and choose **Prescribed Displacement**.
- 2 Select Point 1 only.
- 3 In the **Settings** window for **Prescribed Displacement**, locate the **Coordinate System Selection** section.
- 4 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.
- 5 Locate the **Prescribed Displacement** section. Select the **Prescribed in x3 direction** check box.

Prescribed Displacement 3

- 1 In 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 1 (sys1)**.
- 5 Locate the **Prescribed Displacement** section. Select the **Prescribed in n direction** check box.
- 6 In the u_{0n} text field, type -1 [mm].

Control the inflation of the balloon by the pressure.

Face Load 1

- 1 In 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 pf_m.

Define the pressure pf_m using a **Global Equation** feature.

DEFINITIONS (COMP1)

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
um	intop1(u2)	m	Radial displacement, membrane

MEMBRANE (MBRN)

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
pf_m	um-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 length in the text field.
- 12 Click  **Filter**.
- 13 In the tree, select **General>Length (m)**.
- 14 Click **OK**.

MESH I

Edge I

- 1 In the **Mesh** toolbar, click  **Edge**.
- 2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.

Distribution I

- 1 Right-click **Edge I** 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: PRESTRETCH

The first study solves for the prestretch analysis.

- 1 In the **Model Builder** window, click **Study I**.
- 2 In the **Settings** window for **Study**, type Study: Prestretch 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 **Study: Prestretch** 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)>Shell (shell), Controls spatial frame**.
- 5 Click  **Control Frame Deformation**.
- 6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Face Load 1** and **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Global Equations 1**.
- 7 Click  **Disable**.
- 8 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**.
- 9 Click  **Disable**.

Modify the default solver to improve convergence. Use manual scaling to help the nonlinear solver in the first steps.

Solution 1 (sol1)

- 1 In 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: Prestretch>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Displacement of shell normals (comp1.ar)**.
- 4 In the **Settings** window for **Field**, locate the **Scaling** section.
- 5 In the **Scale** text field, type $1e-9$.
- 6 In the **Study** toolbar, click  **Compute**.

Add a study for the neo-Hookean 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.
- 5 In the **Study** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY: NEO-HOOKEAN

- 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.

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

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 model configuration for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Controls spatial frame**.
- 5 Click  **Control Frame Deformation**.
- 6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Mooney-Rivlin**, **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Ogden**, **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Varga**, and **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Prescribed Displacement/Rotation 3**.
- 7 Click  **Disable**.
- 8 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 3**.
- 9 Click  **Disable**.
- 10 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 11 From the **Method** list, choose **Solution**.
- 12 From the **Study** list, choose **Study: Prestretch, Stationary**.
Use an Auxiliary sweep to ramp up the applied stretch from 1.1 to 10.
- 13 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 14 Click  **Add**.
- 15 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 and use a constant predictor to improve convergence.

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 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: Neo-Hookean>Solver Configurations>Solution 2 (sol2)>Dependent Variables I** node, then click **Displacement of shell normals (comp1.ar)**.
- 6 In the **Settings** window for **Field**, locate the **Scaling** section.
- 7 In the **Scale** text field, type $1e-9$.
- 8 In the **Model Builder** window, click **Displacement field (comp1.u)**.
- 9 In the **Settings** window for **Field**, locate the **Scaling** section.
- 10 From the **Method** list, choose **Manual**.
- 11 In the **Scale** text field, type $1e-3$.
- 12 In the **Model Builder** window, expand the **Study: Neo-Hookean>Solver Configurations>Solution 2 (sol2)>Stationary Solver I** node, then click **Parametric I**.
- 13 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
- 14 From the **Predictor** list, choose **Constant**.
- 15 In the **Model Builder** window, click **Fully Coupled I**.
- 16 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
- 17 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 18 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.
- 19 In the **Study** toolbar, click  **Compute**.

Add a 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.

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

STUDY: MOONEY-RIVLIN

- 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.

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 model configuration for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Controls spatial frame**.
- 5 Click  **Control Frame Deformation**.
- 6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Neo-Hookean, Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Ogden, Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Varga, and Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Prescribed Displacement/Rotation 3**.
- 7 Click  **Disable**.
- 8 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Neo-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 3**.
- 9 Click  **Disable**.
- 10 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 11 From the **Method** list, choose **Solution**.
- 12 From the **Study** list, choose **Study: Prestretch, Stationary**.
Use an Auxiliary sweep to ramp up the applied stretch from 1.1 to 5.
- 13 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 14 Click  **Add**.

15 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 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 **Study: Mooney-Rivlin>Solver Configurations>Solution 3 (sol3)>Dependent Variables 1** node, then click **Displacement of shell normals (comp1.ar)**.
- 6 In the **Settings** window for **Field**, locate the **Scaling** section.
- 7 In the **Scale** text field, type $1e-9$.
- 8 In the **Model Builder** window, click **Displacement field (comp1.u)**.
- 9 In the **Settings** window for **Field**, locate the **Scaling** section.
- 10 From the **Method** list, choose **Manual**.
- 11 In the **Scale** text field, type $1e-3$.
- 12 In the **Model Builder** window, expand the **Study: Mooney-Rivlin>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node, then click **Parametric 1**.
- 13 In the **Settings** window for **Parametric**, locate the **Continuation** section.
- 14 From the **Predictor** list, choose **Constant**.
- 15 In the **Model Builder** window, click **Fully Coupled 1**.
- 16 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
- 17 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 18 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.
- 19 In the **Study** toolbar, click  **Compute**.

Add a study for the Ogden 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.
- 5 In the **Study** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY: OGDEN

- 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.

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 model configuration for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Controls spatial frame**.
- 5 Click  **Control Frame Deformation**.
- 6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Neo-Hookean, Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Mooney-Rivlin, Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Varga, and Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Prescribed Displacement/Rotation 3**.
- 7 Click  **Disable**.
- 8 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Neo-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 3**.
- 9 Click  **Disable**.
- 10 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 11 From the **Method** list, choose **Solution**.
- 12 From the **Study** list, choose **Study: Prestretch, Stationary**.
Use an Auxiliary sweep to ramp up the applied stretch from 1.1 to 10.
- 13 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

14 Click  **Add**.

15 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 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 4 (sol4)** 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: Ogden>Solver Configurations>Solution 4 (sol4)>Dependent Variables I** node, then click **Displacement of shell normals (comp1.ar)**.
- 6 In the **Settings** window for **Field**, locate the **Scaling** section.
- 7 In the **Scale** text field, type $1e-9$.
- 8 In the **Model Builder** window, click **Displacement field (comp1.u)**.
- 9 In the **Settings** window for **Field**, locate the **Scaling** section.
- 10 From the **Method** list, choose **Manual**.
- 11 In the **Scale** text field, type $1e-3$.
- 12 In the **Model Builder** window, expand the **Study: Ogden>Solver Configurations>Solution 4 (sol4)>Stationary Solver I** node, then click **Parametric I**.
- 13 In the **Settings** window for **Parametric**, locate the **Continuation** section.
- 14 From the **Predictor** list, choose **Constant**.
- 15 In the **Model Builder** window, click **Fully Coupled I**.
- 16 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
- 17 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 18 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.
- 19 In the **Study** toolbar, click  **Compute**.

Add a study for the Varga 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.
- 5 In the **Study** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY: VARGA

- 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.

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 model configuration for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Controls spatial frame**.
- 5 Click  **Control Frame Deformation**.
- 6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Neo-Hookean, Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Mooney-Rivlin, Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Ogden, and Component 1 (comp1)>Shell (shell), Spatial frame control disabled>Prescribed Displacement/Rotation 3**.
- 7 Click  **Disable**.
- 8 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Membrane (mbrn), Controls spatial frame>Neo-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 3**.
- 9 Click  **Disable**.
- 10 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 11 From the **Method** list, choose **Solution**.

12 From the **Study** list, choose **Study: Prestretch, Stationary**.

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

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

14 Click  **Add**.

15 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 In 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)>Dependent Variables I** node, then click **Displacement of shell normals (comp1.ar)**.

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

7 In the **Scale** text field, type $1e-9$.

8 In the **Model Builder** window, click **Displacement field (comp1.u)**.

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

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

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

12 In the **Model Builder** window, expand the **Study: Varga>Solver Configurations>Solution 5 (sol5)>Stationary Solver I** node, then click **Parametric I**.

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

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

15 In the **Model Builder** window, click **Fully Coupled I**.

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

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

18 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

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

RESULTS

Add a **Layered Material** dataset for showing the shell results.

Neo-Hookean

- 1 In the **Model Builder** window, expand the **Results** node.
- 2 Right-click **Results>Datasets** and choose **More Datasets>Layered Material**.
- 3 In the **Settings** window for **Layered Material**, type Neo-Hookean in the **Label** text field.
- 4 Locate the **Data** section. From the **Dataset** list, choose **Study: Neo-Hookean/Solution 2 (sol2)**.

Stress (shell)

- 1 In the **Results** toolbar, click  **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type Stress (shell) in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Neo-Hookean**.

Surface 1

- 1 Right-click **Stress (shell)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type shell.mises.
- 4 From the **Unit** list, choose **MPa**.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Deformation 1

- 1 Right-click **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 0.05.
- 5 In the **Stress (shell)** toolbar, click  **Plot**.

Stress (mbrn)

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type Stress (mbrn) in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study: Neo-Hookean/Solution 2 (sol2)**.

Line 1

- 1 Right-click **Stress (mbrn)** and choose **Line**.

- 2 In the **Settings** window for **Line**, locate the **Expression** section.
- 3 In the **Expression** text field, type `mbrn.mises`.
- 4 From the **Unit** list, choose **MPa**.
- 5 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.
- 6 In the **Tube radius expression** text field, type `3`.
- 7 Select the **Radius scale factor** check box.
- 8 In the associated text field, type `1.5E-4`.
- 9 From the **Color table** list, choose **RainbowLight**.

Deformation I

- 1 Right-click **Line 1** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Expression** section.
- 3 In the **R component** text field, type `u2`.
- 4 In the **Z component** text field, type `w2`.
- 5 Locate the **Scale** section. Select the **Scale factor** check box.
- 6 In the associated text field, type `0.05`.
- 7 In the **Stress (mbrn)** toolbar, click  **Plot**.

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

Inflation Pressure

- 1 In 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 (kPa)`.
- 7 Locate the **Axis** section. Select the **Manual axis limits** check box.
- 8 In the **x minimum** text field, type `0.95`.
- 9 In the **x maximum** text field, type `11`.
- 10 In the **y minimum** text field, type `0`.
- 11 In the **y maximum** text field, type `10`.

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 **Study: Neo-Hookean/Solution 2 (sol2)**.
- 4 Select Point 2 only.
- 5 Locate the **y-Axis Data** section. In the **Expression** text field, type pf_s .
- 6 From the **Unit** list, choose **kPa**.
- 7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 8 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Global definitions>Parameters>stretch - Applied stretch**.
- 9 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Red**.
- 10 Click to expand the **Legends** section. Select the **Show legends** check box.
- 11 From the **Legends** list, choose **Manual**.
- 12 In the table, enter the following settings:

Legends
Neo-Hookean, Shell

Point Graph 2

- 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 pf_m .
- 4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
- 5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Neo-Hookean, Membrane

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 **Study: Mooney-Rivlin/Solution 3 (sol3)**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

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

Legends
Mooney-Rivlin, Shell

Point Graph 4

- 1 In the **Model Builder** window, under **Results>Inflation Pressure** right-click **Point Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study: Mooney-Rivlin/Solution 3 (sol3)**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.
- 5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Mooney-Rivlin, Membrane

Point Graph 5

- 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 **Study: Ogden/Solution 4 (sol4)**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Ogden, Shell

Point Graph 6

- 1 In the **Model Builder** window, under **Results>Inflation Pressure** right-click **Point Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study: Ogden/Solution 4 (sol4)**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Ogden, Membrane

Point Graph 7

- 1 Right-click **Point Graph 6** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
- 3 In the **Expression** text field, type $2*(H/Ri)*((6.3e5[Pa]*(stretch^{(1.3-3)} - stretch^{(-2*1.3-3)})) + (0.012e5[Pa]*(stretch^{(5-3)} - stretch^{(-2*5-3)})) - (0.1e5[Pa]*(stretch^{(-2-3)} - stretch^{(2*2-3)})))$.
- 4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 5 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.
- 6 In the **Number** text field, type 12.
- 7 Locate the **Legends** section. In the table, enter the following settings:

Legends
Ogden, Analytical

Point Graph 8

- 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 **Study: Varga/Solution 5 (sol5)**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Magenta**.
- 5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Varga, Shell

Point Graph 9

- 1 In the **Model Builder** window, under **Results>Inflation Pressure** right-click **Point Graph 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study: Varga/Solution 5 (sol5)**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Magenta**.
- 5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Varga, Membrane

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

Inflation Pressure

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 **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 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. In the **y maximum** text field, type 60.
- 6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

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 `mat1_xdim.atxd1(shell.d/2,mean(shell.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 `mbrn.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 `mat1_xdim.atxd1(shell.d/2,mean(shell.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 `mbrn.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 `mat1_xdim.atxd1(shell.d/2,mean(shell.sp1))`.

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

Point Graph 6

1 In the **Model Builder** window, click **Point Graph 6**.

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 7

1 In the **Model Builder** window, click **Point Graph 7**.

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**.

Point Graph 8

1 In the **Model Builder** window, click **Point Graph 8**.

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

3 In the **Expression** text field, type `mat1_xdim.atxd1(shell.d/2,mean(shell.sp1))`.

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

Point Graph 9

1 In the **Model Builder** window, click **Point Graph 9**.

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**.

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

Finally, to reproduce [Figure 5](#), proceed as follows.

Deformed Thickness

- 1 In 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 **Dataset** list, choose **Study: Neo-Hookean/ Solution 2 (sol2)**.
- 4 Locate the **Title** section. From the **Title type** list, choose **Manual**.
- 5 In the **Title** text area, type Comparison of Deformed Thickness.
- 6 Locate the **Plot Settings** section. Select the **y-axis label** check box.
- 7 In the associated text field, type Deformed thickness (mm).

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**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type `mat1_xdim.atxd1(shell.d/2,mean(shell.ddef))`.
- 5 From the **Unit** list, choose **mm**.
- 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 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 9 Locate the **Legends** section. Select the **Show legends** check box.
- 10 From the **Legends** list, choose **Manual**.
- 11 In the table, enter the following settings:

Legends
Shell

Point Graph 2

- 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 `mbrn.ddef`.
- 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:

Legends
Membrane

Global 1

1 In the **Model Builder** window, 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)	mm	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. Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

6 In the **Number** text field, type 12.

7 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
Solid Mechanics

9 In the **Deformed Thickness** toolbar, click  **Plot**.