Model created in COMSOL Multiphysics 6.4

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, here, 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 in 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 the rotational degrees of freedom. However, since both the Membrane and the Shell interface are defined on one geometric dimension lower than the corresponding Solid Mechanics interface, modeling with these interfaces is computationally more efficient.

All relevant details about the geometry and the 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 using the Solid Mechanics interface.

Figure 1 shows the distribution of the hoop 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 is in good agreement 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 hoop stress for a neo-Hookean material in the Membrane interface at maximum inflation. A uniform stress through thickness of 41.8 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.

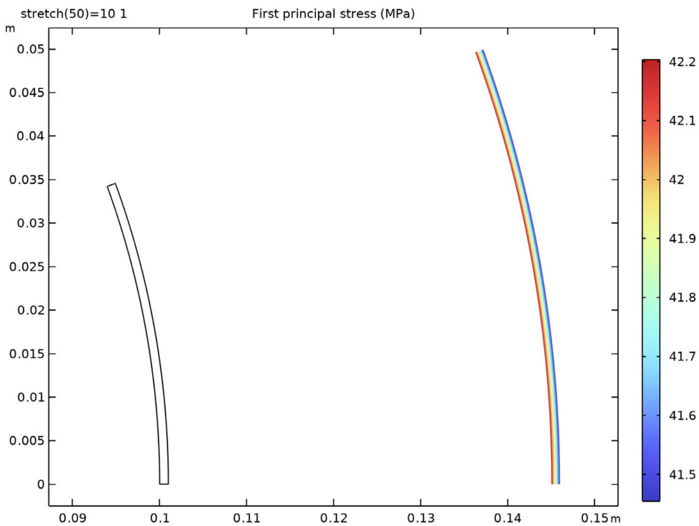

Figure 1: Distribution of hoop stress for the neo-Hookean material in the shell interface at maximum inflation.

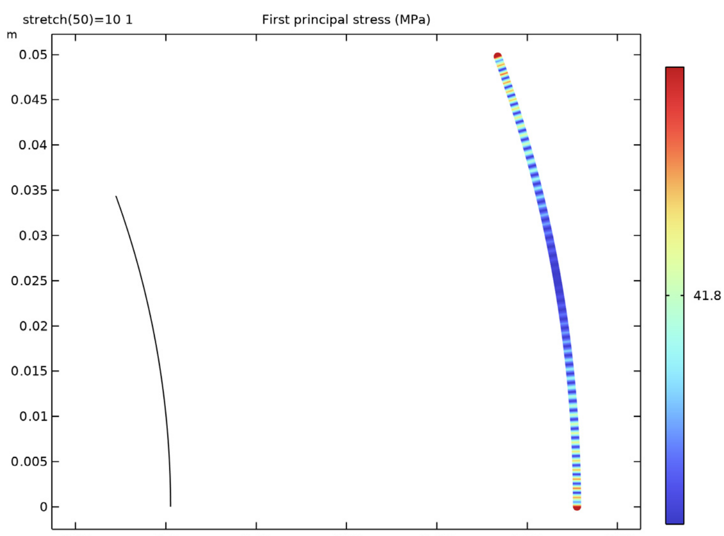

Figure 2: Distribution of hoop stress for the neo-Hookean material in the membrane interface at maximum inflation.

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

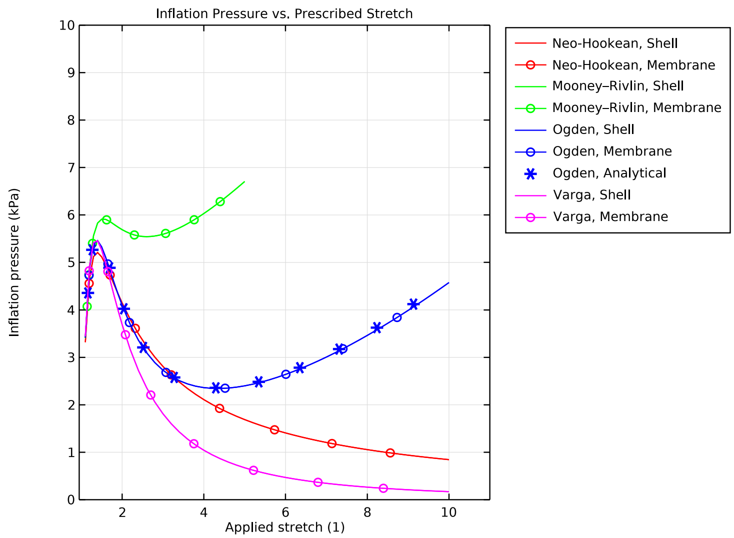

Figure 3: Computed inflation pressure as a function of circumferential stretch for different material models, compared to the analytical expression for the Ogden material.

The variation in hoop stress versus applied stretch for different hyperelastic material models is shown in Figure 4.

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

.

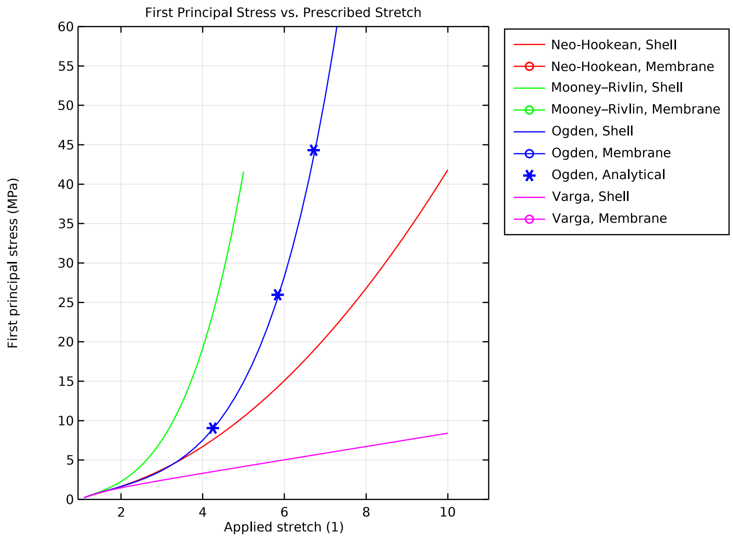

Figure 4: Computed hoop stress as a function of circumferential stretch for different material models, compared to the analytical expression for the Ogden material.

.

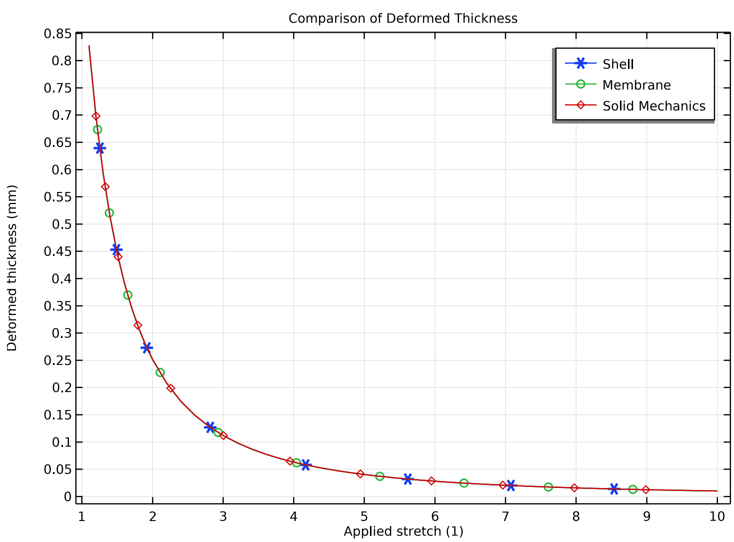

Figure 5: Comparison of the through-thickness deformation in the Shell, Membrane, and Solid Mechanics interfaces.

Notes About the COMSOL Implementation

The absence of bending stiffness in a membrane requires a prestretching 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 prestretching 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]	1	Applied stretch
C10	0.4375*mu	1.8484E5 Pa	Mooney-Rivlin parameter C10
C01	0.0625*mu	26406 Pa	Mooney-Rivlin parameter C01
mu1	6.3e5[Pa]	6.3E5 Pa	Ogden parameter mu1
mu2	0.012e5[Pa]	1200 Pa	Ogden parameter mu2
mu3	-0.1e5[Pa]	-10000 Pa	Ogden parameter mu3
alpha1	1.3	1.3	Ogden parameter alpha1
alpha2	5	5	Ogden parameter alpha2
alpha3	-2	-2	Ogden parameter alpha3

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 f(x) **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 **Argument** table, enter the following settings:

Argument	Unit
t	1

6 In the **Function** table, enter the following settings:

Function	Unit
int1	mm

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
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 `Ri+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 **Hyperelastic Material, Layered** 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	1	Young's modulus and Poisson's ratio
Thickness	lth	H	m	Shell

SHELL (SHELL)

Neo-Hookean

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

2 In the **Settings** window for **Hyperelastic Material, Layered**, 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**.

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

6 In the κ text field, type `kappa`.

Mooney–Rivlin

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

2 In the **Settings** window for **Hyperelastic Material, Layered**, 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 `C10`.

6 From the C_{01} list, choose **User defined**. In the associated text field, type `C01`.

7 In the κ text field, type `kappa`.

Ogden

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

2 In the **Settings** window for **Hyperelastic Material, Layered**, 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)
1	mu1	alpha1
2	mu2	alpha2
3	mu3	alpha3

7 In the κ text field, type `kappa`.

Varga

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

2 In the **Settings** window for **Hyperelastic Material, Layered**, 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 From the **Displacement in z direction** list, choose **Prescribed**.

5 Locate the **Prescribed Rotation** section. From the **By** list, choose **Rotation**.

DEFINITIONS (COMP1)

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[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. From the **Displacement in x3 direction** list, choose **Prescribed**.
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. From the **Displacement in n direction** list, choose **Prescribed**.
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**. First, define a nonlocal integration coupling to evaluate the displacement at point 2.

DEFINITIONS (COMP1)

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** checkbox.

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, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

6 Click **OK** to enable a global equations and other advanced modeling tools.

SHELL (SHELL)

Global Equations 1 (ODE1)

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 (ut_0) (1/s)	Description
pf_s	us-u_appl	0	0	

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

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

6 In the tree, select **General** > **Pressure (Pa)**.

7 Click **OK**.

8 In the **Settings** window for **Global Equations**, locate the **Units** section.

9 Click **Select Source Term Quantity**.

10 In the **Physical Quantity** dialog, type `length` in the text field.

11 In the tree, select **General** > **Length (m)**.

12 Click **OK**.

MEMBRANE (MBRN)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Membrane (mbrn)** click **Thickness and Offset 1**.

2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 In the d_0 text field, type `H`.

Neo-Hookean

Repeat the setup of the material models and boundary conditions for the Membrane interface.

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**.

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

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 `C10`.

6 From the C_{01} list, choose **User defined**. In the associated text field, type `C01`.

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)
1	mu1	alpha1
2	mu2	alpha2
3	mu3	alpha3

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*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 From the **Displacement in z direction** list, choose **Prescribed**.

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. From the **Displacement in x3 direction** list, choose **Prescribed**.

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. From the **Displacement in n direction** list, choose **Prescribed**.

6 In the u_{0n} text field, type `-1[mm]`.

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`.

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 (ODE2)

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 (ut_0) (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, type `pressure` in the text field.

6 In the tree, select **General** > **Pressure (Pa)**.

7 Click **OK**.

8 In the **Settings** window for **Global Equations**, locate the **Units** section.

9 Click **Select Source Term Quantity**.

10 In the **Physical Quantity** dialog, type `length` in the text field.

11 In the tree, select **General** > **Length (m)**.

12 Click **OK**.

Before building the mesh and solving, define variables for the analytical expressions of inflation pressure and hoop stress for Ogden's model.

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
p_Ogden	2*(H/Ri)*(mu1*(stretch^(alpha1-3)-stretch^(-2*alpha1-3))+mu2*(stretch^(alpha2-3)-stretch^(-2*alpha2-3))+mu3*(stretch^(alpha3-3)-stretch^(-2*alpha3-3)))	Pa	Pressure (Ogden, analytical)
sp1_Ogden	mu1*(stretch^alpha1-stretch^(-2*alpha1))+mu2*(stretch^alpha2-stretch^(-2*alpha2))+mu3*(stretch^alpha3-stretch^(-2*alpha3))	Pa	Hoop stress (Ogden, analytical)

MESH 1

Edge 1

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

Distribution 1

1 Right-click **Edge 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type `50`.
4 Click **Build All**.

STUDY: PRESTRETCH

The first study solves for the prestretch analysis.

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

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** checkbox.
4 In the tree, select **Component 1 (comp1)** > **Shell (shell), Controls spatial frame**.
5 Click **Control Frame Deformation**.
6 In the 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 (ODE1)**.
7 Right-click and choose **Disable**.
8 In the 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 (ODE2)**.
9 Right-click and choose **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 the **Add Study** button in the window toolbar.

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

STUDY: NEO-HOOKEAN

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

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

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** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Shell (shell), Controls spatial frame**.

5 Click **Control Frame Deformation**.

6 In the 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 Right-click and choose **Disable**.

8 In the 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 Right-click and choose **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** checkbox.

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)`	1

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 1**.

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

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

5 In the **Model Builder** window, expand the **Study: Neo-Hookean** > **Solver Configurations** > **Solution 2 (sol2)** > **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, under **Study: Neo-Hookean** > **Solver Configurations** > **Solution 2 (sol2)** > **Dependent Variables 1** 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 1** node, then click **Parametric 1**.

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, under **Study: Neo-Hookean** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** click **Fully Coupled 1**.

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 the **Add Study** button in the window toolbar.

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

STUDY: MOONEY–RIVLIN

1 In the **Settings** window for **Study**, type `Study: Mooney-Rivlin` in the **Label** text field.

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

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** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Shell (shell), Controls spatial frame**.

5 Click **Control Frame Deformation**.

6 In the 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 Right-click and choose **Disable**.

8 In the 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 Right-click and choose **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** checkbox.

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)	1

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, under **Study: Mooney–Rivlin** > **Solver Configurations** > **Solution 3 (sol3)** > **Dependent Variables 1** 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, under **Study: Mooney–Rivlin** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** 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 the **Add Study** button in the window toolbar.

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

STUDY: OGDEN

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

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

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** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Shell (shell), Controls spatial frame**.

5 Click **Control Frame Deformation**.

6 In the 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 Right-click and choose **Disable**.

8 In the 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 Right-click and choose **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** checkbox.

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)`	1

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 **Study: Ogden** > **Solver Configurations** > **Solution 4 (sol4)** > **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, under **Study: Ogden** > **Solver Configurations** > **Solution 4 (sol4)** > **Dependent Variables 1** 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 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, under **Study: Ogden** > **Solver Configurations** > **Solution 4 (sol4)** > **Stationary Solver 1** 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 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 the **Add Study** button in the window toolbar.

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

STUDY: VARGA

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

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

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** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Shell (shell), Controls spatial frame**.

5 Click **Control Frame Deformation**.

6 In the 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 Right-click and choose **Disable**.

8 In the 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 Right-click and choose **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** checkbox.

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)`	1

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 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: Varga** > **Solver Configurations** > **Solution 5 (sol5)** > **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, under **Study: Varga** > **Solver Configurations** > **Solution 5 (sol5)** > **Dependent Variables 1** 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 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, under **Study: Varga** > **Solver Configurations** > **Solution 5 (sol5)** > **Stationary Solver 1** 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**.

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Click **Add Physical Quantity**.

9 In the **Physical Quantity** dialog, select **General** > **Pressure (Pa)** in the tree.

10 Click **OK**.

11 In the **Settings** window for **Preferred Units**, locate the **Units** section.

12 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Pressure	Pa	kPa

13 Click **Apply**.

Neo-Hookean

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

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.sp1`.

4 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** checkbox. In the associated text field, type `0.05`.

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

5 In the **Stress (shell)** toolbar, click **Plot**.

Stress (mbrn)

1 In the **Results** toolbar, click **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.sp1`.

4 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

5 In the **Tube radius expression** text field, type `3`.

6 Select the **Radius scale factor** checkbox. In the associated text field, type `1.5E-4`.

7 From the **Color table** list, choose **RainbowLight**.

Deformation 1

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.

6 Select the **Scale factor** checkbox. In the associated text field, type `0.05`.

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

8 In the **Stress (mbrn)** toolbar, click **Plot**.

To reproduce Figure 3, proceed as follows.

Inflation Pressure

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D 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.

6 Select the **y-axis label** checkbox. In the associated text field, type `Inflation pressure (kPa)`.

7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

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`.

12 Click to expand the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

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 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 - 1**.

8 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Red**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

11 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 From the **Positioning** list, choose **Interpolated**.

6 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 `p_Ogden`.

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 **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. In the **y maximum** text field, type `60`.

6 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

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 `shell.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`.

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 `shell.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`.

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 `shell.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`.

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 `sp1_Ogden`.

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 `shell.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 In the **First Principal Stress** toolbar, click **Plot**.

Finally, to reproduce Figure 5, proceed as follows.

Deformed Thickness

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D 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.

7 Select the **y-axis label** checkbox. 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 `shell.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 - 1**.

8 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 From the **Positioning** list, choose **Interpolated**.

10 Locate the **Legends** section. Select the **Show legends** checkbox.

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

12 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 From the **Unit** list, choose **mm**.

5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type 10.

6 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 - 1**.

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

6 From the **Positioning** list, choose **Interpolated**.

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

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

9 In the table, enter the following settings:

Legends
Solid Mechanics

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