Model created in COMSOL Multiphysics 6.4

Wrinkling of Cylindrical Membranes with Varying Thickness

Introduction

The numerical treatment of wrinkling in membranes can be handled by two approaches, see Ref. 1. In the first approach, out-of-plane geometric nonlinearities are treated as constitutive nonlinearities through a modification of the strain energy density, which is called the relaxed strain energy approach. In contrast, the second method involves direct modifications of the deformation gradient instead of the constitutive relation. The second approach is more general and applicable to anisotropic membranes, and this method is implemented in COMSOL Multiphysics.

In this example, wrinkling is studied in a cylindrical membrane of nonuniform thickness that is being stretched axially and filled with water internally. The membrane material is described with an incompressible Mooney–Rivlin model. Depending on the level of axial stretch and internal pressure, certain portions of the membrane undergo wrinkling. The results of the two approaches of handling wrinkling are compared with each other as well as with the results from the example presented in Ref. 1. In the case of the relaxed strain energy approach, the total strain energy for a Mooney–Rivlin material, which is a combination of the full and the relaxed strain energy, is taken from Ref. 1.

Model Definition

The model example is taken from Ref. 1. A cylindrical membrane of radius 10 mm and initial height of 80 mm is first stretched axially and subsequently filled with water. The membrane is modeled with an incompressible two-parameter Mooney–Rivlin hyperelastic material. The material properties are given in Table 1.

TABLE 1: MATERIAL PROPERTIES.

Property	**Variable**	**Value**
Mooney–Rivlin parameter C_1	C_1	0.2111 MPa
Mooney–Rivlin parameter C_2	C_2	2.111 kPa

The membrane thickness varies along the height as $\mathrm{th} = \mathrm{th}_\mathrm{m}(2(1-M)(Z/H_i)+M)$, where H_i is initial the height, th_m is the mean thickness of 0.1 mm, and M is a parameter controlling the variation in thickness, here taken to be 0.5.

Results and Discussions

Figure 1 and Figure 2 show the wrinkled regions in the stretched and inflated cylinder by both approaches at different levels of water column height. The results match with each other closely; the slight variation in the wrinkled region with the relaxed strain-energy approach comes from the transition zone approximation of the step function.

After stretching, the central portion of the membrane is wrinkled. As the water level increases, the extent of the wrinkled region reduces and finally disappears at around $z_w = 70$ mm. The results are in agreement with those published in Ref. 1.

The first and second principal stresses in the inflated cylinder are shown in Figure 3 and Figure 4, respectively. The stresses from the modified deformation gradient approach matches with the relaxed strain energy approach (see the results from the model). Note that the second principal stress is nonnegative in the whole membrane.

Figure 5 and Figure 6 show the variations in the principal stresses along the height of cylinder after the prestretch. The results from both approaches agree with each other; moreover, the second principal stress is nonnegative in the wrinkled region. Similarly, Figure 7 shows the variation in the third principal strain after prestretch, and both approaches give the same results.

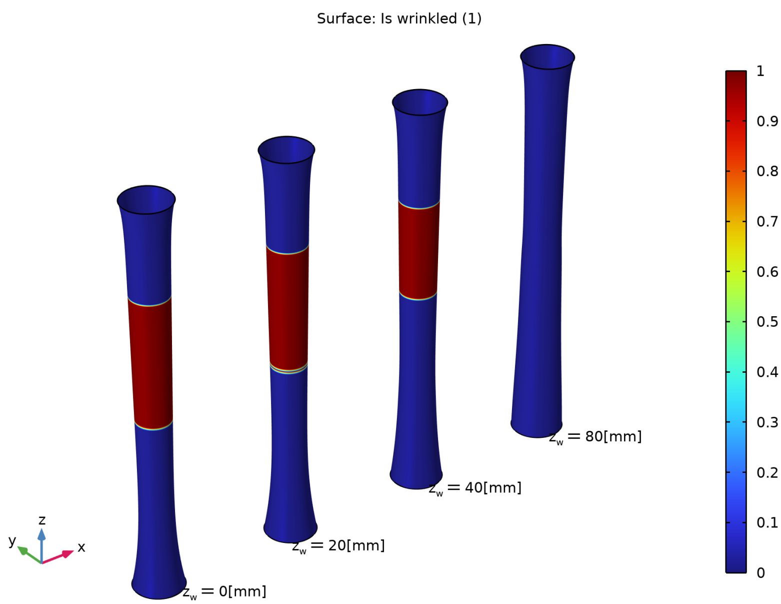

Figure 1: Wrinkled region in the inflated cylinder at different water heights computed with the modified deformation gradient approach.

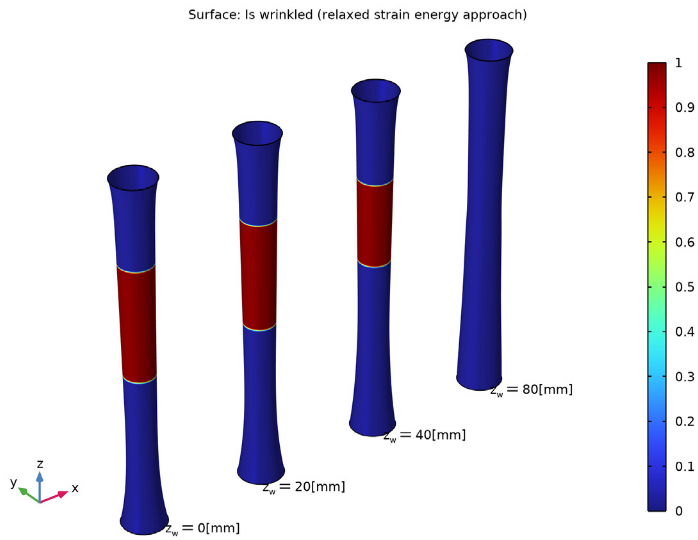

Figure 2: Wrinkled region in the inflated cylinder at different water heights computed with the relaxed strain energy approach.

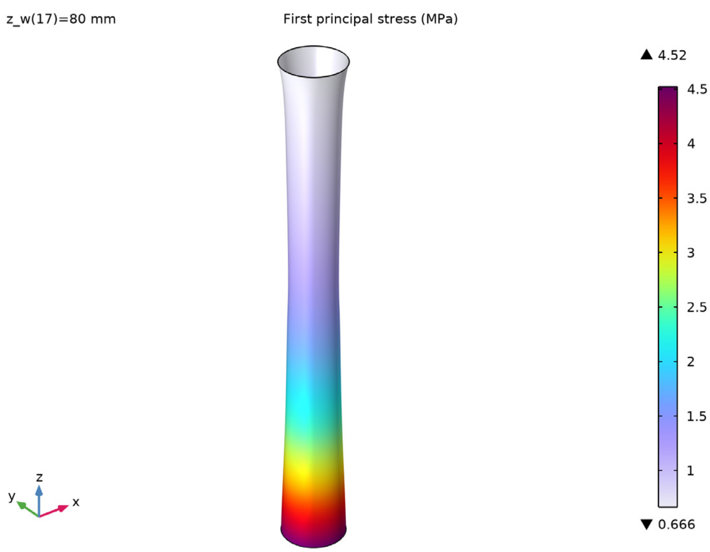

Figure 3: First principal stress in the inflated cylinder computed with the modified deformation gradient approach.

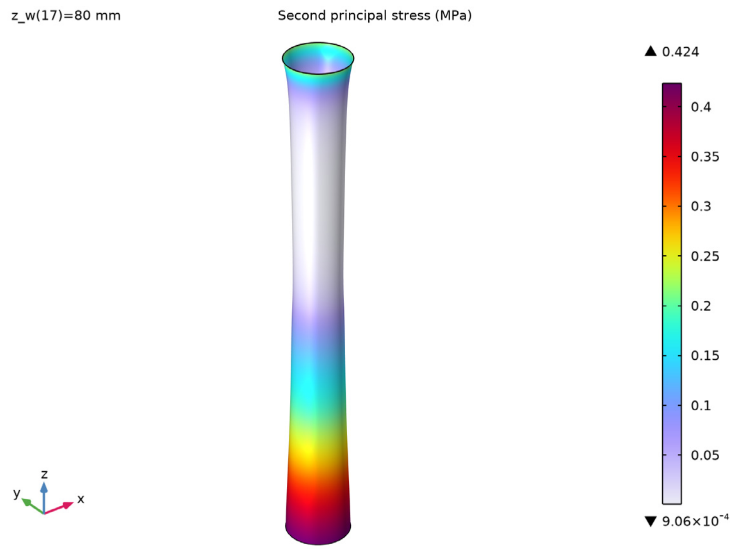

Figure 4: Second principal stress in the inflated cylinder with the modified deformation gradient approach.

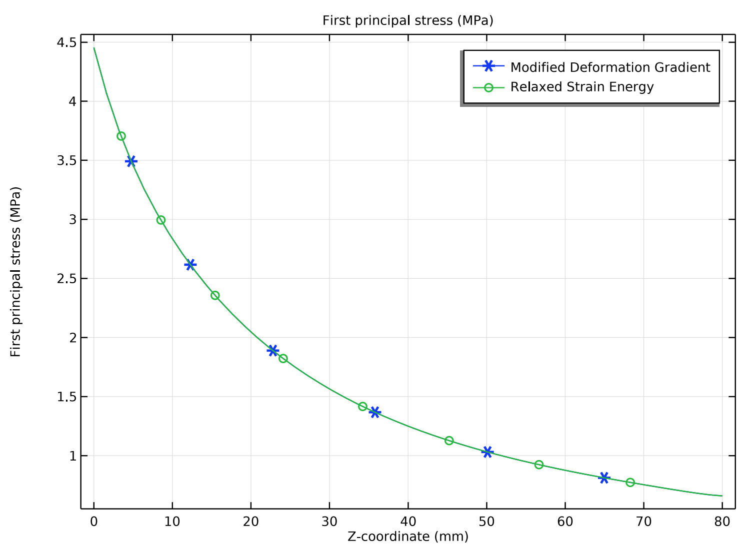

Figure 5: First principal stress in the cylinder after prestretch.

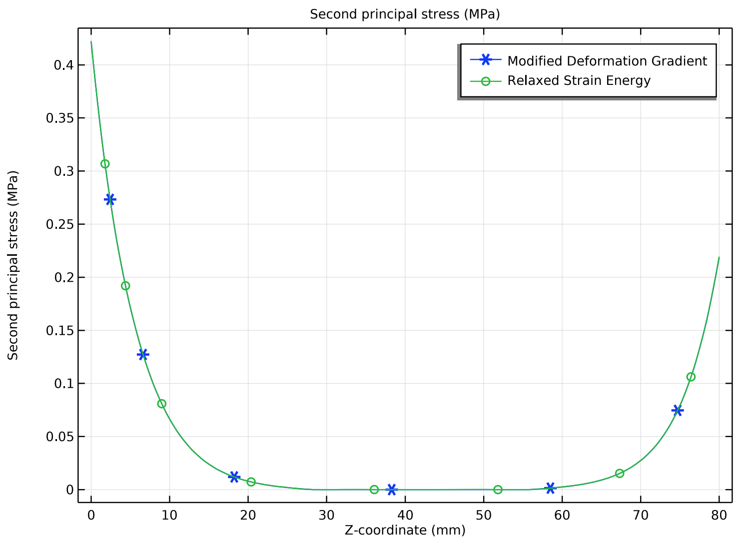

Figure 6: Second principal stress in the cylinder after prestretch.

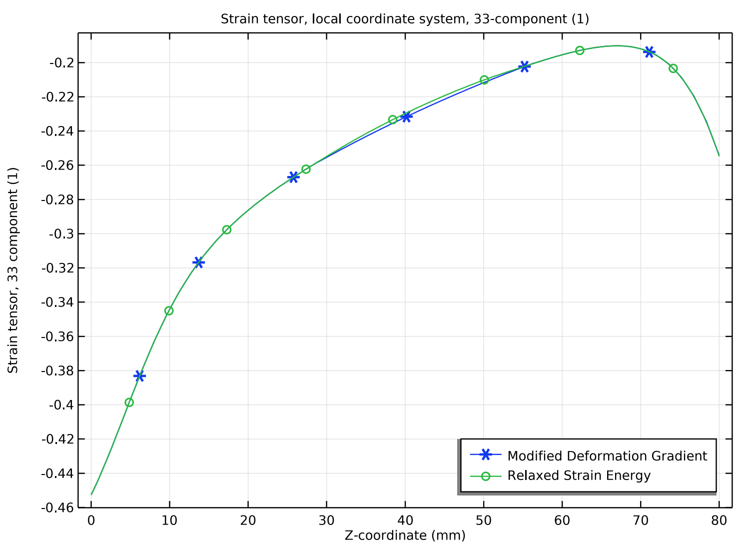

Figure 7: Third principal strain in the cylinder after prestretch.

Notes About the COMSOL Implementation

A wrinkling model based on the modified deformation gradient is incorporated within the membrane theory using the **Wrinkling** feature, which solves a set of nonlinear equations with the Newton–Raphson method.

The strain energy density based on the relaxed strain energy approach is taken from Ref. 1 and implemented through the **User defined** option under the **Hyperelastic Material** node. A weak contribution is added to enforce material incompressibility. Furthermore, wrinkling is identified to occur in the limit when the second principal stress turns negative (uniaxial stress conditions). Because the material is isotropic and incompressible, the wrinkling condition can thus be formulated in terms of the in-plane principal stretches as

$$\lambda_2 \le \frac{1}{\sqrt{\lambda_1}} \Rightarrow \lambda_2\sqrt{\lambda_1} \le 1 \tag{1}$$

Reference

1. A. Patil, “Inflation and instabilities of hyperelastic membranes,” *PhD Thesis,* KTH, 2016.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/membrane_varying_thickness`

Modeling Instructions

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

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Membrane (mbrn)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Model Parameters

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

2 In the **Settings** window for **Parameters**, type `Model Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

DEFINITIONS

Step 1 (step1)

1 In the **Definitions** toolbar, click f(X) **More Functions** and choose **Step**.

2 In the **Settings** window for **Step**, click to expand the **Smoothing** section.

3 Locate the **Parameters** section. In the **Location** text field, type `1`.

4 Locate the **Smoothing** section. In the **Size of transition zone** text field, type `0.008`.

Variables 1

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

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

3 Click **Load from File**.

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

GEOMETRY 1

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

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

3 From the **Length unit** list, choose **mm**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **r** text field, type `Ri`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **r** text field, type `Ri`.

7 In the **z** text field, type `Hi`.

8 Click **Build Selected**.

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 `th`.

To model wrinkling using the built-in modified deformation gradient approach, add a Mooney–Rivlin hyperelastic material with a **Wrinkling** subnode.

Hyperelastic Material (Modified Deformation Gradient)

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Hyperelastic Material**, type `Hyperelastic Material (Modified Deformation Gradient)` in the **Label** text field.

4 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **Mooney–Rivlin, two parameters**.

5 From the **Compressibility** list, choose **Incompressible**.

6 From the C_{10} list, choose **User defined**. In the associated text field, type `C1`.

7 From the C_{01} list, choose **User defined**. In the associated text field, type `C2`.

Wrinkling 1

1 In the **Physics** toolbar, click **Attributes** and choose **Wrinkling**.

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

3 From the **Termination criterion for local method** list, choose **Step size or residual**.

To model wrinkling using the relaxed strain energy approach, add a user-defined hyperelastic material.

Hyperelastic Material (Relaxed Strain Energy)

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

The energy expression considers incompressibility implicitly, hence we do not need to use the built-in mixed formulation coming with **Nearly incompressible** or **Incompressible** models. Therefore, choose the **Compressible** option.

2 In the **Settings** window for **Hyperelastic Material**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 In the **Label** text field, type `Hyperelastic Material (Relaxed Strain Energy)`.

5 Locate the **Hyperelastic Material** section. From the **Material model** list, choose **User defined**.

6 In the W_s text field, type `WT`.

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

8 In the **Show More Options** dialog, click **Select All**.

9 Click **OK**.

Add a **Weak Contribution** to determine the normal strain for the relaxed strain energy approach.

Weak Contribution (incompressibility)

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Weak Contribution**, locate the **Weak Contribution** section.

4 In the **Weak expression** text field, type `(-1+mbrn.Jel)*test(mbrn.unn)`.

5 In the **Label** text field, type `Weak Contribution (incompressibility)`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 1 only.

Prescribed Displacement (Prestretch)

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 In the **Settings** window for **Prescribed Displacement**, type `Prescribed Displacement (Prestretch)` in the **Label** text field.

3 Select Point 2 only.

4 Locate the **Prescribed Displacement** section. From the **Displacement in r direction** list, choose **Prescribed**.

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

6 In the u_{0z} text field, type `w_appl`.

Face Load (Fluid Pressure)

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Face Load**, locate the **Force** section.

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

5 In the p text field, type `P`.

6 In the **Label** text field, type `Face Load (Fluid Pressure)`.

MESH 1

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

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

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

Set up two study steps: one for prestretching, and another for inflation. Use the prestretch solution as initial values for the inflation step.

STUDY (MODIFIED DEFORMATION GRADIENT)

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

2 In the **Settings** window for **Study**, type `Study (Modified Deformation Gradient)` in the **Label** text field.

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

Prestretch

1 In the **Model Builder** window, under **Study (Modified Deformation Gradient)** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, type `Prestretch` in the **Label** text field.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Hyperelastic Material (Relaxed Strain Energy)** and **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Weak Contribution (incompressibility)**.

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

6 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Face Load (Fluid Pressure)**.

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

Inflation

1 In the **Study** toolbar, click **Stationary**.

2 In the **Settings** window for **Stationary**, type `Inflation` in the **Label** text field.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Hyperelastic Material (Relaxed Strain Energy)** and **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Weak Contribution (incompressibility)**.

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

6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
z_w (Height of water column)	`range(0,5,80)`	mm

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

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

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

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

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

STUDY (RELAXED STRAIN ENERGY)

1 In the **Settings** window for **Study**, type `Study (Relaxed Strain Energy)` in the **Label** text field.

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

Prestretch

1 In the **Model Builder** window, under **Study (Relaxed Strain Energy)** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, type `Prestretch` in the **Label** text field.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Membrane (mbrn), Controls spatial frame** > **Face Load (Fluid Pressure)**.

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

Inflation

1 In the **Study** toolbar, click **Stationary**.

2 In the **Settings** window for **Stationary**, type `Inflation` in the **Label** text field.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
z_w (Height of water column)	`range(0,5,80)`	`mm`

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

RESULTS

Set default units for result presentation.

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 **General** > **Displacement (m)** 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
Displacement	m	mm

8 Click **Add Physical Quantity**.

9 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** 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
Stress tensor	N/m^2	MPa

13 Click **Apply**.

Add a plot from **Result Templates** and edit it to show the wrinkled region in the membrane.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study (Modified Deformation Gradient)/Solution 1 (sol1)** > **Membrane** > **Stress, 3D (mbrn)**.

4 Click the **Add Result Template** button in the window toolbar.

RESULTS

Stress, 3D (mbrn)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 Click **Go to Source**.

Revolution 2D

1 In the **Model Builder** window, under **Results** > **Datasets** click **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.

3 In the **Start angle** text field, type `0`.

4 In the **Revolution angle** text field, type `360`.

Wrinkled Region

1 In the **Model Builder** window, under **Results** click **Stress, 3D (mbrn)**.

2 In the **Settings** window for **3D Plot Group**, type `Wrinkled Region` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (z_w (mm))** list, choose **0**.

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

5 Find the **Solution** subsection. Clear the **Solution** checkbox.

Surface 1

1 In the **Model Builder** window, expand the **Wrinkled Region** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Membrane** > **Wrinkling** > **mbrn.iswrinkled - Is wrinkled - 1**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Rainbow**.

4 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

5 From the **Smoothing** list, choose **None**.

Solution Array 1

1 Right-click **Surface 1** and choose **Solution Array**.

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

3 From the **Parameter selection (z_w)** list, choose **From list**.

4 In the **Parameter values (z_w (mm))** list, choose **0**, **20**, **40**, and **80**.

Wrinkled Region

1 In the **Model Builder** window, under **Results** click **Wrinkled Region**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Plot Array** section.

3 In the **Relative padding** text field, type 2.

4 In the **Model Builder** window, click **Wrinkled Region**.

Table Annotation 1

1 In the **Wrinkled Region** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, click to expand the **Plot Array** section.

3 Locate the **Data** section. From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
5	-1	0	\[\textrm{z}_\textrm{w}=\textrm{0[mm]}\]
55	-1	0	\[\textrm{z}_\textrm{w}=\textrm{20[mm]}\]
120	-1	0	\[\textrm{z}_\textrm{w}=\textrm{40[mm]}\]
180	-1	0	\[\textrm{z}_\textrm{w}=\textrm{80[mm]}\]

5 Select the **LaTeX markup** checkbox.

6 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

Wrinkled Region

1 In the **Model Builder** window, click **Wrinkled Region**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **New view**.

4 In the **Wrinkled Region** toolbar, click **Plot**.

5 Click the **Show Grid** button in the **Graphics** toolbar.

Add another plot and edit it to show the first principal stress in 3D.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study (Modified Deformation Gradient)/Solution 1 (sol1)** > **Membrane** > **Stress, 3D (mbrn)**.

3 Click the **Add Result Template** button in the window toolbar.

4 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

First Principal Stress

1 In the **Settings** window for **3D Plot Group**, type `First Principal Stress` in the **Label** text field.

2 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

3 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

Surface 1

1 In the **Model Builder** window, expand the **First Principal Stress** node, then click **Surface 1**.

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

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

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

5 Click the **Show Grid** button in the **Graphics** toolbar.

Duplicate the plot group to also show the second principal stress.

Second Principal Stress

1. In the **Model Builder** window, right-click **First Principal Stress** and choose **Duplicate**.
2. In the **Settings** window for **3D Plot Group**, type `Second Principal Stress` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Second Principal Stress** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `mbrn.sp2`.
4. In the **Second Principal Stress** toolbar, click **Plot**.

First Principal Stress, Second Principal Stress, Wrinkled Region

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Wrinkled Region**, **First Principal Stress**, and **Second Principal Stress**.
2. Right-click and choose **Group**.

Modified Deformation Gradient

In the **Settings** window for **Group**, type `Modified Deformation Gradient` in the **Label** text field.

Relaxed Strain Energy

1. Right-click **Modified Deformation Gradient** and choose **Duplicate**.

   Duplicate the node group **Modified Deformation Gradient** to show the plots corresponding to the relaxed strain energy approach.
2. In the **Settings** window for **Group**, type `Relaxed Strain Energy` in the **Label** text field.

Revolution 2D 1

1. In the **Model Builder** window, right-click **Revolution 2D** and choose **Duplicate**.
2. In the **Settings** window for **Revolution 2D**, locate the **Data** section.
3. From the **Dataset** list, choose **Study (Relaxed Strain Energy)/Solution 3 (sol3)**.

Relaxed Strain Energy

In the **Model Builder** window, expand the **Results** > **Relaxed Strain Energy** node.

Surface 1

1. In the **Model Builder** window, expand the **Results** > **Relaxed Strain Energy** > **Wrinkled Region 1** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `iswrinkled`.

Wrinkled Region 1

1 In the **Model Builder** window, click **Wrinkled Region 1**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 1**.

4 In the **Wrinkled Region 1** toolbar, click **Plot**.

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

First Principal Stress 1

1 In the **Model Builder** window, click **First Principal Stress 1**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 1**.

4 In the **First Principal Stress 1** toolbar, click **Plot**.

Second Principal Stress 1

1 In the **Model Builder** window, click **Second Principal Stress 1**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 1**.

4 In the **Second Principal Stress 1** toolbar, click **Plot**.

Third Principal Strain after Prestretch

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

2 In the **Settings** window for **1D Plot Group**, type `Third Principal Strain after Prestretch` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study (Modified Deformation Gradient)/Solution Store 1 (sol2)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Strain tensor, 33 component (1)`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Line Graph 1

1 Right-click **Third Principal Strain after Prestretch** and choose **Line Graph**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `mbrn.el33`.

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

6 In the **Expression** text field, type `Z`.

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

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

9 In the **Number** text field, type `6`.

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

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

12 In the table, enter the following settings:

Legends
Modified Deformation Gradient

Line Graph 2

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

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

3 From the **Dataset** list, choose **Study (Relaxed Strain Energy)/Solution Store 2 (sol4)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. In the **Number** text field, type `8`.

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

Legends
Relaxed Strain Energy

Third Principal Strain after Prestretch

1 In the **Model Builder** window, click **Third Principal Strain after Prestretch**.

2 In the **Third Principal Strain after Prestretch** toolbar, click **Plot**.

First Principal Stress after Prestretch

1 Right-click **Third Principal Strain after Prestretch** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `First Principal Stress after Prestretch` in the **Label** text field.

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

4 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Line Graph 1

1 In the **Model Builder** window, expand the **First Principal Stress after Prestretch** node, then click **Line Graph 1**.

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

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

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

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

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

First Principal Stress after Prestretch

1 In the **Model Builder** window, click **First Principal Stress after Prestretch**.

2 In the **First Principal Stress after Prestretch** toolbar, click **Plot**.

Second Principal Stress after Prestretch

1 Right-click **First Principal Stress after Prestretch** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Second Principal Stress after Prestretch` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Second principal stress (MPa)`.

Line Graph 1

1 In the **Model Builder** window, expand the **Second Principal Stress after Prestretch** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `mbrn.sp2`.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

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

3 In the **Expression** text field, type `mbrn.sp2`.

4 In the **Second Principal Stress after Prestretch** toolbar, click **Plot**.