Created in COMSOL Multiphysics 6.1

Inflation of a Square Hyperelastic Airbag

Introduction

The numerical treatment of thin structures with a membrane model is much simpler than with a shell model due to the assumption of zero bending stiffness. However, for some load cases this assumption is disadvantageous. For instance, the zero bending stiffness triggers wrinkling when a membrane is subjected to compressive stresses, which result in an equilibrium instability. This limitation can be overcome by the incorporation of a wrinkling model based on the tension field theory.

In this example, a square airbag made of a neo-Hookean hyperelastic material is inflated using pressurized air. The results are compared with the example presented in Ref. 1.

The example is similar to the model *Inflation of a Square Airbag* in the Structural Mechanics Module Application Library, where a linear elastic material model is used.

Model Definition

A square airbag, 1 mm in thickness and 1.2 m in diagonal length, is inflated using pressurized air. The membrane is modeled as a compressible neo-Hookean hyperelastic material. The material properties given in Table 1 are taken from Ref. 1.

TABLE 1: MATERIAL PROPERTIES.

Property	**Variable**	**Value**
Young's modulus	E	588 MPa
Poisson's ratio	ν	0.4

Only a quarter of the airbag is analyzed due to the intrinsic symmetry of the model.

Results and Discussions

Figure 1 shows the wrinkled regions in the inflated airbag. Apart from the central region, wrinkles develop everywhere.

The transverse displacement of the inflated airbag is shown in Figure 2. The distribution of the first and second principal stress at the end of the inflation process is shown in Figure 3 and Figure 4, respectively. The minimum value of the second principal stress is almost zero. Both figures show tensile principal stresses after inflation.

Figure 5 shows the displacement of three different material points as a function of the inflation pressure. These displacements are compared with the results given in Ref. 1. At the final pressure, the values match the reference values well. Figure 6 shows the variation in the first principal (tensile) stress in the midpoint of the airbag with the inflation pressure.

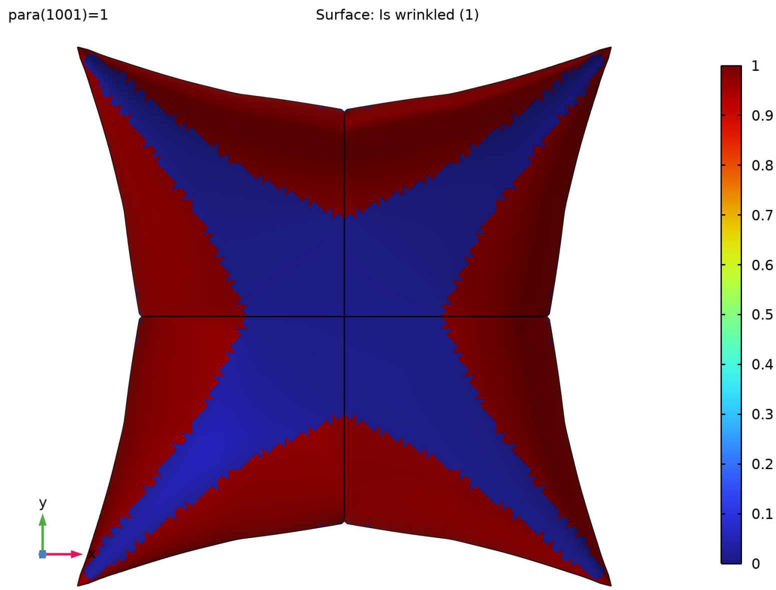

Figure 1: Wrinkled region in the inflated airbag.

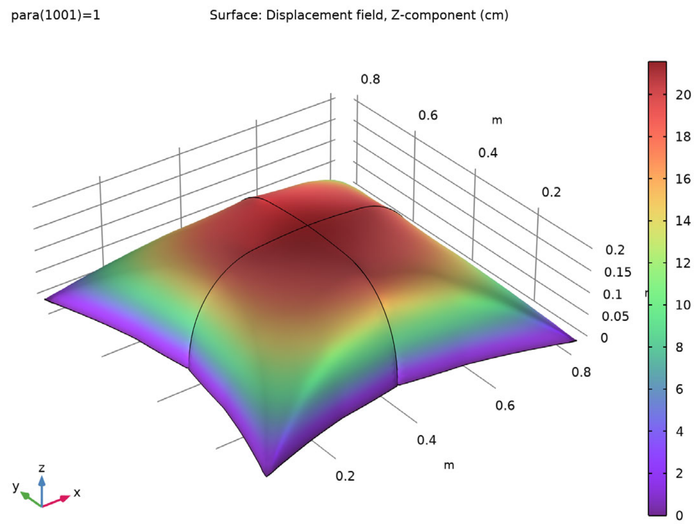

Figure 2: Transverse displacement after inflation.

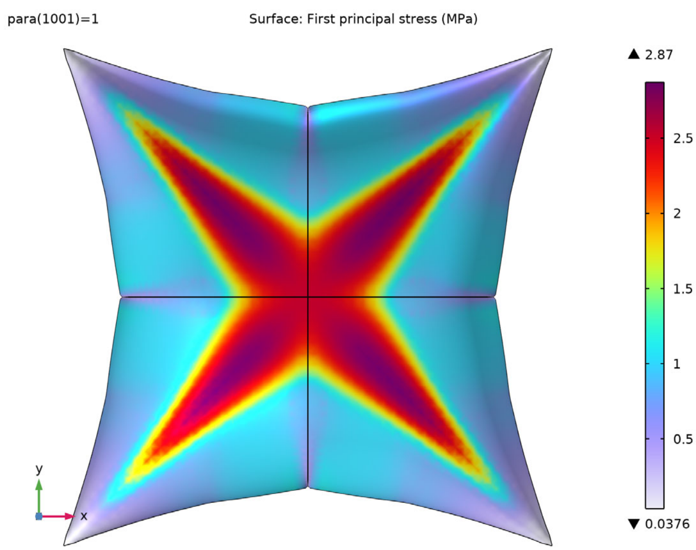

Figure 3: First principal stress after inflation.

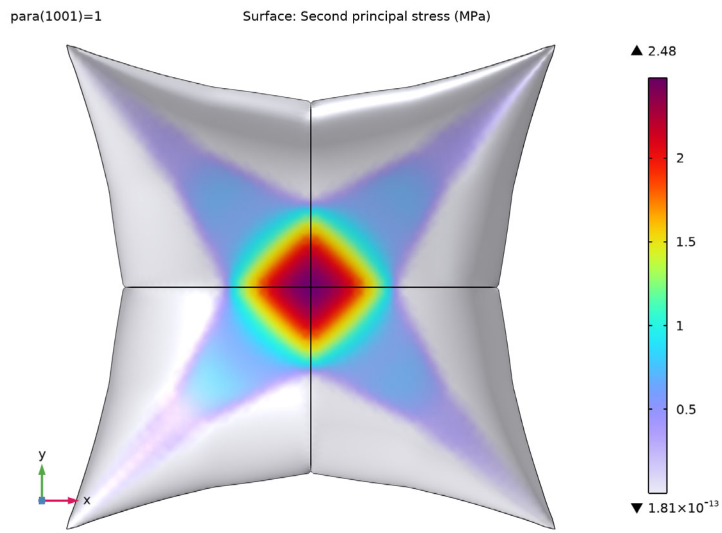

Figure 4: Second principal stress after inflation.

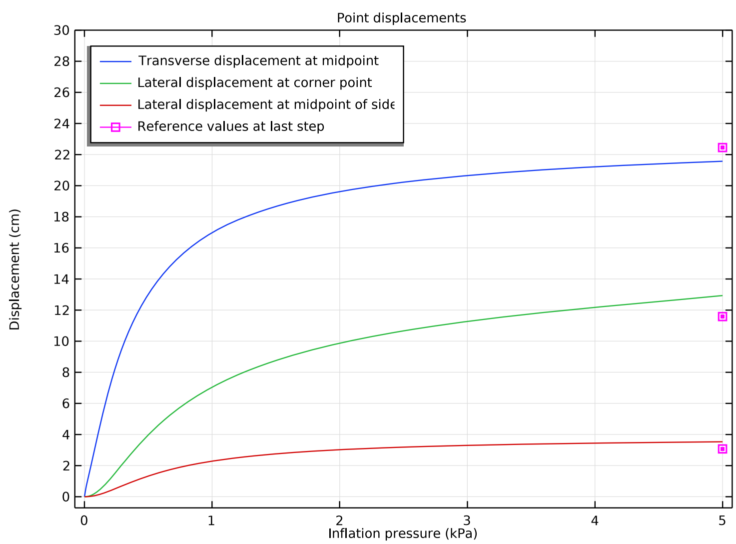

Figure 5: Displacements at different material points during inflation.

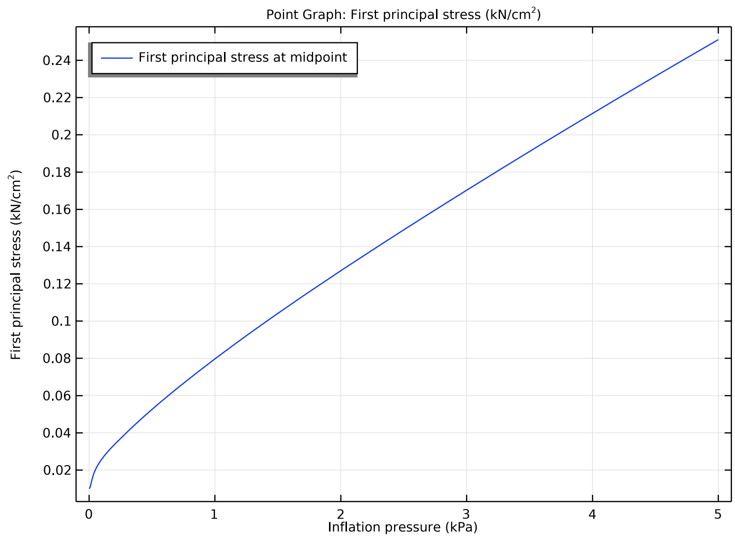

Figure 6: First principal stress in the airbag center during inflation.

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.

Since the unstressed membrane does not have any stiffness in the normal direction, a very small spring support is added to stabilize the model. To further improve the numerical stability, a load support is provided in the form of an edge load that decreases parametrically as the model becomes stable when the membrane is under tension.

Constant strain triangle (CST) elements are numerically stable for wrinkling problems. CST elements give constant strains, which in turn give constant stresses for homogeneous material properties, ensuring that the whole element is either wrinkled, slack, or taut. This type of elements are used for the analysis.

Reference

1. A. Diaby, A.L. Van, and C. Wielgosz, "Wrinkling and buckling of prestressed membranes," *Finite Elem. Anal. Des.,* vol. 42, no. 11, pp. 992–1001, 2006.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/membrane_airbag_inflation_hyperelastic`

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 **3D**.

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_airbag_inflation_hyperelastic_parameters.txt`.

DEFINITIONS

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `F`.

4 In the table, enter the following settings:

t	f(t)
0	100
1	1

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

Function	Unit
F	N/m

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

Argument	Unit
t	1

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 In the table, enter the following settings:

Name	Expression	Unit	Description
wmax_ref	22.45[cm]	m	Transverse displacement at midpoint, reference
u1max_ref	3.07[cm]	m	Lateral displacement at corner point, reference
u2max_ref	11.58[cm]	m	Lateral displacement at midpoint of side edge, reference

Due to symmetry, only a quarter of the geometry is constructed.

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node.

2 Right-click **Geometry 1** and choose **Work Plane**.

Work Plane 1 (wp1)>Square 1 (sq1)

1 In the **Work Plane** toolbar, click **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type L/2.

4 Locate the **Position** section. In the **xw** text field, type `L/2`.

5 In the **Work Plane** toolbar, click **Build All**.

MATERIALS

Material 1 (mat1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

When modeling wrinkling, constant strain triangular (CST) elements are a good choice due to their numerical stability. Use a triangular mesh and change the default **Quadratic** discretization to **Linear**.

MEMBRANE (MBRN)

1 In the **Settings** window for **Membrane**, click to expand the **Discretization** section.

2 From the **Displacement field** list, choose **Linear**.

Hyperelastic Material 1

1 Right-click **Component 1 (comp1)>Membrane (mbrn)** and choose **Material Models>Hyperelastic Material**.

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

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

4 Locate the **Hyperelastic Material** section. From the **Specify** list, choose **Young's modulus and Poisson's ratio**.

Wrinkling 1

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

Wrinkling 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Membrane (mbrn)>Hyperelastic Material 1** node, then click **Wrinkling 1**.

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**.

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`.

Symmetry 1

1. In the **Physics** toolbar, click **Edges** and choose **Symmetry**.
2. Select Edges 1 and 3 only.

Prescribed Displacement 1

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

Face Load 1

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 `-Pmax*para`.

Add a parametrically decreasing edge load on the outer edges in the lateral direction in form of a spring force in order to increase the numerical stability. Note that the local edge coordinate system is used.

Edge Load 1

1. In the **Physics** toolbar, click **Edges** and choose **Edge Load**.
2. Select Edges 2 and 4 only.
3. In the **Settings** window for **Edge Load**, locate the **Coordinate System Selection** section.
4. From the **Coordinate system** list, choose **Local edge system**.
5. Locate the **Force** section. Specify the $\mathbf{F}_L$ vector as

0	xl
-F(para)	yl
0	zl

Add a spring support in the thickness direction in order to improve numerical stability.

Spring Foundation 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.
2. In the **Settings** window for **Spring Foundation**, locate the **Boundary Selection** section.

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

4 Locate the **Spring** section. From the list, choose **Symmetric**.

5 In the $\mathbf{k}_A$ table, enter the following settings:

0	0	0
0	0	0
0	0	1e-3[N/m^3]

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 1 (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	EE	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	Nu	1	Young's modulus and Poisson's ratio
Density	rho	0	kg/m³	Basic

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

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

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

Distribution 1

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

2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type 25.

Convert 1

1 In the **Mesh** toolbar, click **Modify** and choose **Convert**.

2 In the **Settings** window for **Convert**, locate the **Element Split Method** section.

3 From the **Element split method** list, choose **Insert centerpoints**.

4 Click **Build All**.

Customize the study settings in order to achieve better convergence.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	range(0,0.001,1)	

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

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

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

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 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

5 Select the **Tuning of step size** check box.

6 In the **Maximum step size** text field, type `0.001`.

7 From the **Predictor** list, choose **Linear**.

8 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Fully Coupled 1**.

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

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

11 In the **Maximum number of iterations** text field, type `100`.

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

13 In the **Settings** window for **Study**, locate the **Study Settings** section.

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

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

Add a plot of the transverse displacement. Use Mirror operations to create a dataset of the entire airbag.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Mirror 3D 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **More 3D Datasets>Mirror 3D**.

3 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

4 From the **Plane** list, choose **XZ-planes**.

5 In the **Y-coordinate** text field, type `0.5*L`.

Mirror 3D 2

1 Right-click **Mirror 3D 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **yz-planes**.

5 In the **x-coordinate** text field, type `0.5*L`.

6 In the **Results** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Membrane>Displacement (mbrn)**.

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

4 In the **Results** toolbar, click **Add Predefined Plot**.

RESULTS

Transverse Displacement

1 In the **Model Builder** window, under **Results** click **Displacement (mbrn)**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 2**.

Surface 1

1 In the **Model Builder** window, expand the **Transverse Displacement** node, then click **Surface 1**.

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

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

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

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Rainbow>SpectrumLight** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Surface**, click to expand the **Quality** section.

9 From the **Smoothing threshold** list, choose **None**.

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

11 In the **Transverse Displacement** toolbar, click **Plot**.

First Principal Stress

1 In the **Model Builder** window, right-click **Transverse Displacement** and choose **Duplicate**.

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

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

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**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Membrane>Stress>Principal stresses>mbrn.sp1 - First principal stress - N/m²**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

6 Click **OK**.

First Principal Stress

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

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 **First Principal Stress** toolbar, click **Plot**.

5 Click the **Go to XY View** button in the **Graphics** toolbar.

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

Second Principal Stress

1 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**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Membrane>Stress>Principal stresses>mbrn.sp2 - Second principal stress - N/m²**.

Second Principal Stress

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

2 In the **Second Principal Stress** toolbar, click **Plot**.

Wrinkled Region

1 Right-click **Second Principal Stress** and choose **Duplicate**.

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

3 Locate the **Color Legend** section. Clear the **Show maximum and minimum values** check box.

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**.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Surface**, locate the **Quality** section.

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

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

Point Displacements

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Point Displacements` in the **Label** text field.

Point Graph 1

1 Right-click **Point Displacements** 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 `w`.

5 From the **Unit** list, choose **cm**.

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

7 In the **Expression** text field, type `Pmax*para`.

8 From the **Unit** list, choose **kPa**.

9 Click to expand 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
`Transverse displacement at midpoint`

Point Graph 2

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

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

3 Click **Clear Selection**.

4 Select Point 4 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `-u`.

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

Legends
Lateral displacement at corner point

Point Graph 3

1 Right-click **Point Graph 2** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Point 3 only.

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

Legends
Lateral displacement at midpoint of side edge

Global 1

1 In the **Model Builder** window, right-click **Point Displacements** and choose **Global**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 From the **Parameter selection (para)** list, choose **Last**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
wmax_ref	cm	Transverse displacement at midpoint, reference

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

7 In the **Expression** text field, type Pmax*para.

8 From the **Unit** list, choose **kPa**.

9 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Magenta**.

10 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

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

12 Set the **Number** value to **12**.

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

14 In the table, enter the following settings:

Legends
Reference values at last step

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
u1max_ref	cm	Lateral displacement at corner point, reference

4 Locate the **Legends** section. Clear the **Show legends** check box.

Global 3

1 Right-click **Global 2** and choose **Duplicate**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
u2max_ref	cm	Total displacement at midpoint of side edge, reference

Point Displacements

1 In the **Model Builder** window, click **Point Displacements**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

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

4 In the **Title** text area, type `Point displacements`.

5 Locate the **Plot Settings** section.

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

7 Select the **y-axis label** check box. In the associated text field, type `Displacement (cm)`.

8 Locate the **Axis** section. Select the **Manual axis limits** check box.

9 In the **y maximum** text field, type `30`.

10 In the **Point Displacements** toolbar, click **Plot**.

11 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

12 In the **Point Displacements** toolbar, click **Plot**.

Point Stress

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type `Inflation pressure (kPa)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Point Stress** 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 `mbrn.sp1`.

5 In the **Unit** field, type `kN/cm^2`.

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

7 In the **Expression** text field, type `Pmax*para`.

8 From the **Unit** list, choose **kPa**.

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
`First principal stress at midpoint`

Point Stress

1 In the **Model Builder** window, click **Point Stress**.

2 In the **Point Stress** toolbar, click **Plot**.