Created in COMSOL Multiphysics 6.2

Uniaxial Stretching of a Rectangular Membrane

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, when a membrane is subjected to compressive stresses, it may trigger wrinkling when these reach a critical level defined by the bending stiffness. This undesirable limitation can be overcome with the incorporation of a wrinkling model that removes these instabilities.

In this example, a rectangular membrane is stretched uniaxially, which results in wrinkling in the central region. The wrinkling region and stress distribution depend on the geometry, loading conditions, and material properties. The analytical results are used to verify the numerical solution.

Model Definition

A rectangular sheet 1 m in length, 0.5 m in width, and 1 mm thick is stretched in the longitudinal direction.

One of the short edges is fixed, while a prescribed displacement of 1 mm is applied on the opposite edge. The long edges are unconstrained.

The sheet is made of a linear elastic material, and both isotropic and orthotropic scenarios are analyzed. The material properties are given in Table 1.

MATERIAL PROPERTIES

TABLE 1: MATERIAL PROPERTIES.

Property	Variable	Isotropic Case	Orthotropic Case
Young's modulus, 1 direction	E_1	100 kPa	100 kPa
Young's modulus, 2 direction	E_2	100 kPa	20 kPa
Young's modulus, 3 direction	E_3	100 kPa	20 kPa
Poisson's ratio, 12 direction	ν_{12}	0.3	0.3
Poisson's ratio, 23 direction	ν_{23}	0.3	0
Poisson's ratio, 13 direction	ν_{13}	0.3	0
Shear modulus, 12 direction	G_{12}	$E_1/(2(1+\nu_{12}))$	38.5 kPa
Shear modulus, 23 direction	G_{23}	G_{12}	G_{12}
Shear modulus, 13 direction	G_{13}	G_{12}	G_{12}

Results and Discussions

The numerical and analytical results are compared side by side in the same figure. For this particular problem, wrinkling occurs in the 2$^{\mathrm{nd}}$ local direction as uniaxial stretching is applied in the 1$^{\mathrm{st}}$ local direction.

The analytical expression for the wrinkling measure for isotropic and orthotropic materials is derived from the equality

$$S_{22} = D_{12}e_{11} + D_{22}e_{22} + D_{23}e_{33} + \beta D_{22} = 0$$

which gives the expression for the wrinkling measure as

$$\beta = -\frac{D_{12}e_{11} + D_{22}e_{22} + D_{23}e_{33}}{D_{22}}$$

Figure 1 and Figure 2 show the wrinkled regions for the isotropic and orthotropic cases. The results match the analytical values for both scenarios. For the isotropic case, the whole central region is wrinkled, while for the orthotropic case, only the areas along the long edges are wrinkled.

Figure 3 and Figure 4 show the measure of wrinkling in the membrane. Again, in both isotropic and orthotropic cases the results match the analytical expressions. The region of maximum wrinkling is the region where maximum compressive stress could have developed without a wrinkling model.

The first principal stress and the tensile direction for the isotropic and orthotropic cases are shown in Figure 5 and Figure 6. Figure 7 and Figure 8 show the second principal stress along the wrinkling direction. As expected, the direction of wrinkling is perpendicular to the loading direction in uniaxial stretching. The lowest value of the second principal stress is nearly zero. Both figures show no compressive stresses.

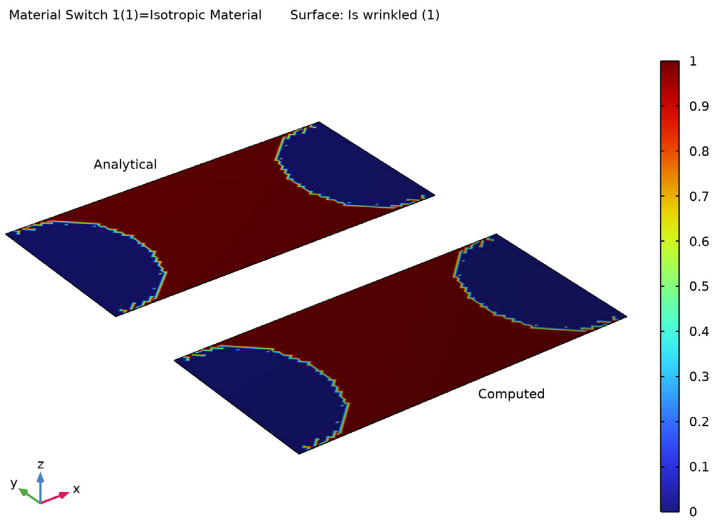

Figure 1: Wrinkled region, isotropic material.

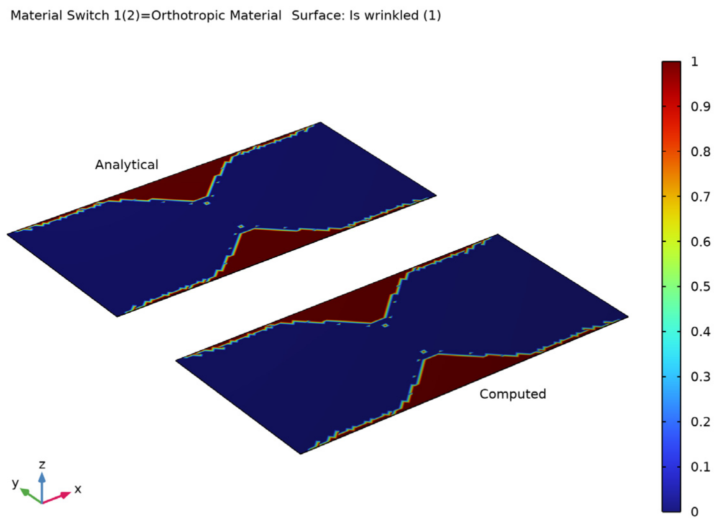

Figure 2: Wrinkled region, orthotropic material.

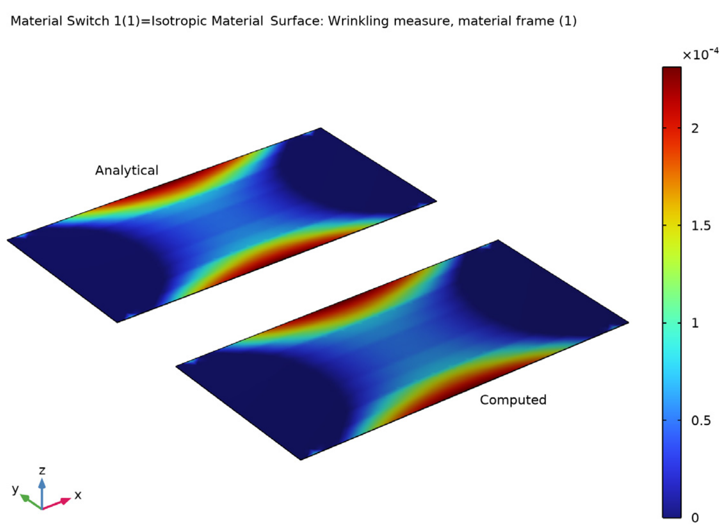

Figure 3: Measure of wrinkling, isotropic material.

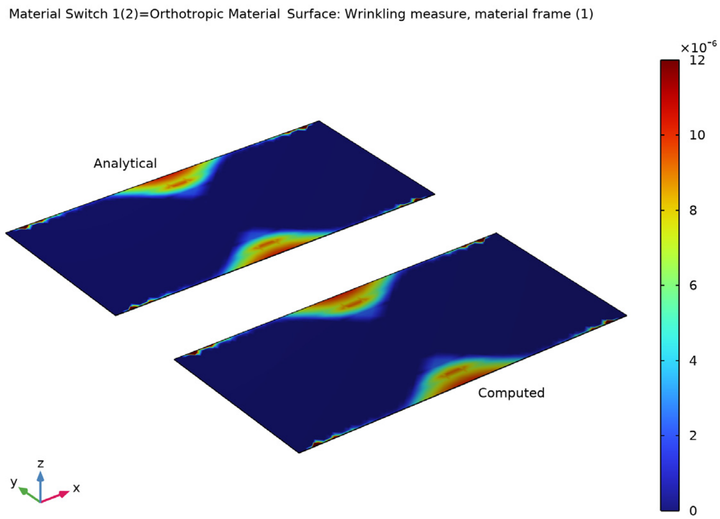

Figure 4: Measure of wrinkling, orthotropic material.

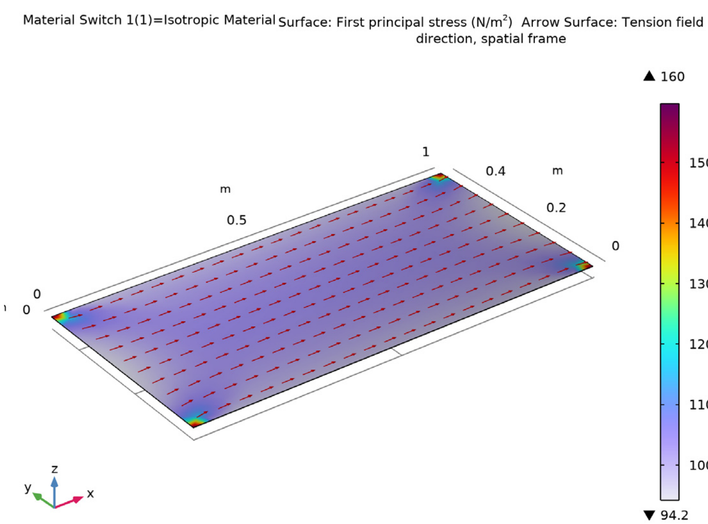

Figure 5: First principal stress, isotropic material.

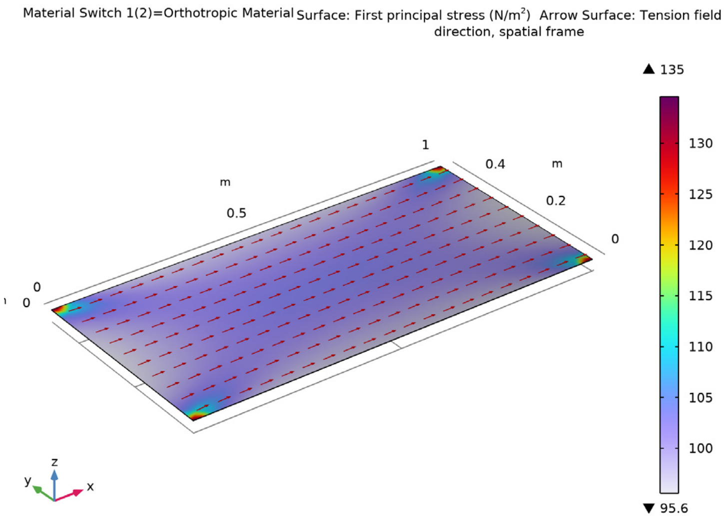

Figure 6: First principal stress, orthotropic material.

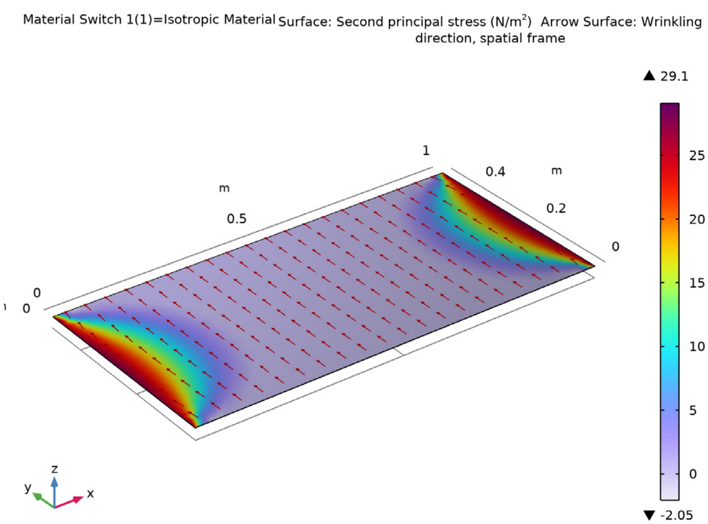

Figure 7: Second principal stress, isotropic material.

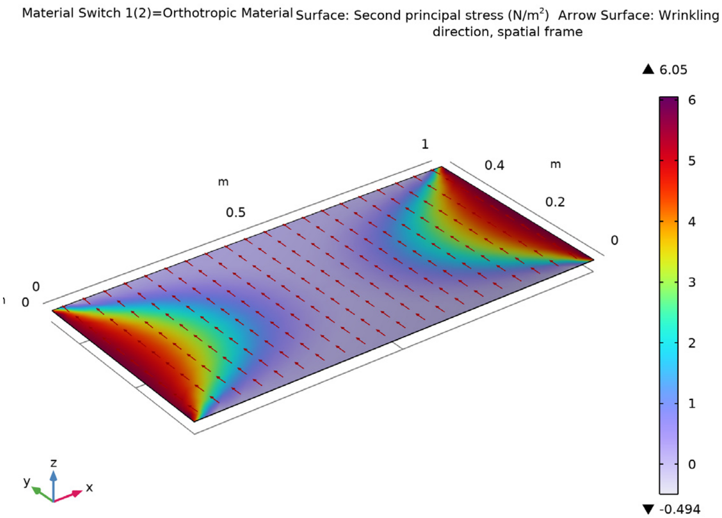

Figure 8: Second principal stress, orthotropic material.

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 stiffness in the normal direction, a very small spring support is added in order to stabilize the model.

Application Library path: `Structural_Mechanics_Module/Buckling_and_Wrinkling/membrane_uniaxial_stretching`

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

Geometric Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, type `Geometric Parameters` in the **Label** text field.
3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
th	1[mm]	0.001 m	Thickness of rectangular sheet
L	1[m]	1 m	Length of rectangular sheet
W	0.5[m]	0.5 m	Width of rectangular sheet

Isotropic Material Properties

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.
2 In the **Settings** window for **Parameters**, type `Isotropic Material Properties` 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_uniaxial_stretching_isotropic_properties.txt`.

Orthotropic Material Properties

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.
2 In the **Settings** window for **Parameters**, type `Orthotropic Material Properties` 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_uniaxial_stretching_orthotropic_properties.txt`.

DEFINITIONS

Variables 1

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

2 Right-click **Definitions** and choose **Variables**.

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

4 In the table, enter the following settings:

Name	Expression	Unit	Description
`Beta_ana`	`max(-(mbrn.D12*mbrn.eel11+mbrn.D22*mbrn.eel22+mbrn.D23*mbrn.eel33)/mbrn.D22,0)`		`Wrinkling measure, analytical`
`iswrinkled_ana`	`Beta_ana>0`		`Is wrinkled, analytical`

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `L`.

4 In the **Height** text field, type `W`.

5 Click **Build Selected**.

MEMBRANE (MBRN)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Membrane (mbrn)** click **Linear Elastic Material 1**.

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

3 From the **Material symmetry** list, choose **Orthotropic**.

Wrinkling 1

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

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`.

Fixed Constraint 1

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

2 Select Edge 1 only.

Prescribed Displacement 1

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

2 Select Edge 4 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in x direction** list, choose **Prescribed**.

5 In the u_{0x} text field, type `1[mm]`.

6 From the **Displacement in y direction** list, choose **Prescribed**.

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

Add a spring support in the thickness direction in order to achieve numerical stability for this problem.

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]

Add a **Material Switch** node in order to run same study with different materials.

MATERIALS

Material Switch 1 (sw1)

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

Isotropic Material

1 In the **Model Builder** window, right-click **Material Switch 1 (sw1)** and choose **Blank Material**.

   Orthotropic elastic properties are asked by the material due to the settings in the physics features. The isotropic material can however be defined by adding the isotropic properties manually. The orthotropic properties will be calculated automatically by synchronization rules.

2 In the **Settings** window for **Material**, type `Isotropic Material` in the **Label** text field.

3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material>Young's Modulus and Poisson's Ratio**.

4 Click **Add to Material**.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`E_iso`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`nu_iso`	1	Young's modulus and Poisson's ratio
Density	rho	0	kg/m³	Basic

   Material Switch 1 may show a warning. You can ignore it since the orthotropic material properties are well defined by the synchronization rules.

Orthotropic Material

1 In the **Model Builder** window, right-click **Material Switch 1 (sw1)** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Orthotropic 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
Young's modulus	{Evector1, Evector2, Evector3}	`{E1_orth, E2_orth, E3_orth}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{nu12_orth, nu23_orth, nu13_orth}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{G12_orth, G23_orth, G13_orth}`	N/m^2	Orthotropic
Density	rho	0	kg/m^3	Basic

MESH 1

Mapped 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.
2. Select Boundary 1 only.

Size

1. In the **Model Builder** window, click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Finer**.

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**.

STUDY 1

Material Sweep

1. In the **Study** toolbar, click **Material Sweep**.
2. In the **Settings** window for **Material Sweep**, locate the **Study Settings** section.
3. Click **Add**.

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 **Fully Coupled 1**.

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

5 From the **Termination criterion** list, choose **Solution and residual**.

6 In the **Maximum number of iterations** text field, type `50`.

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

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

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

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

RESULTS

Wrinkled Region

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Material Switch 1** list, choose **Isotropic Material**.

5 Click to expand the **Plot Array** section. Select the **Enable** check box.

6 From the **Array axis** list, choose **y**.

7 In the **Relative padding** text field, type `0.5`.

Surface 1

1 In the **Wrinkled Region** toolbar, click **Surface**.

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 Click to expand the **Quality** section. From the **Smoothing** list, choose **None**.

4 Right-click **Surface 1** and choose **Duplicate**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

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

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

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

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

Wrinkled Region

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

Annotation 1

1 In the **Wrinkled Region** toolbar, click **Annotation**.

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

3 In the **Text** text field, type `Computed`.

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

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

6 Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

Wrinkled Region

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

Annotation 2

1 In the **Wrinkled Region** toolbar, click **Annotation**.

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

3 In the **Text** text field, type `Analytical`.

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

5 In the **Y** text field, type `W`.

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

7 From the **Anchor point** list, choose **Lower right**.

8 Locate the **Plot Array** section. Select the **Manual indexing** check box.

9 In the **Index** text field, type `1`.

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

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.

6 Right-click **Wrinkled Region** and choose **Duplicate**.

Wrinkling Measure

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

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

Surface 1

1 In the **Model Builder** window, expand the **Wrinkling Measure** 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.lemm1.wr1.Beta - Wrinkling measure, material frame - 1**.

3 Locate the **Quality** section. From the **Smoothing** list, choose **Inside material domains**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

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

4 Locate the **Quality** section. From the **Smoothing** list, choose **Inside material domains**.

5 In the **Wrinkling Measure** toolbar, click **Plot**.

First Principal Stress

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 From the **Material Switch 1** list, choose **Isotropic Material**.

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

Surface 1

1 Right-click **First Principal Stress** and choose **Surface**.

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.sp1Gp - First principal stress - N/m²**.

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

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

5 Click **OK**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **First Principal Stress** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Membrane>Wrinkling>mbrn.lemm1.wr1.tnx,...,mbrn.lemm1.wr1.tnz - Tension field direction, spatial frame**.

First Principal Stress

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

2 Right-click **First Principal Stress** and choose **Duplicate**.

Second Principal Stress

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

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.sp2Gp - Second principal stress - N/m²**.

Arrow Surface 1

1 In the **Model Builder** window, click **Arrow Surface 1**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Membrane>Wrinkling>mbrn.lemm1.wr1.wnx,...,mbrn.lemm1.wr1.wnz - Wrinkling direction, spatial frame**.

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

Second Principal Stress

In the **Model Builder** window, collapse the **Results>Second Principal Stress** node.

Now, group all plots for isotropic materials, and duplicate them for plotting results with orthotropic material.

First Principal Stress, Second Principal Stress, Wrinkled Region, Wrinkling Measure

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Wrinkled Region**, **Wrinkling Measure**, **First Principal Stress**, and **Second Principal Stress**.

2 Right-click and choose **Group**.

Isotropic Material

1 In the **Settings** window for **Group**, type `Isotropic Material` in the **Label** text field.

2 Right-click **Isotropic Material** and choose **Duplicate**.

Orthotropic Material

1 In the **Model Builder** window, under **Results** click **Isotropic Material 1**.

2 In the **Settings** window for **Group**, type `Orthotropic Material` in the **Label** text field.

Wrinkled Region 1

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

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

3 From the **Material Switch 1** list, choose **Orthotropic Material**.

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

Wrinkling Measure 1

1 In the **Model Builder** window, click **Wrinkling Measure 1**.

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

3 From the **Material Switch 1** list, choose **Orthotropic Material**.

Surface 1

1 In the **Model Builder** window, expand the **Wrinkling Measure 1** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 Set the **Maximum** value to **1.2E-5**.

5 In the **Wrinkling Measure 1** toolbar, click **Plot**.

First Principal Stress 1

1 In the **Model Builder** window, under **Results>Orthotropic Material** click **First Principal Stress 1**.

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

3 From the **Material Switch 1** list, choose **Orthotropic Material**.

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 **Material Switch 1** list, choose **Orthotropic Material**.

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