Created in COMSOL Multiphysics 6.2

# Torsion of a Circular Membrane

## *Introduction*

The numerical treatment of thin structures with a membrane theory is much simpler than with a shell theory 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 which is an equilibrium instability. This undesirable limitation can be overcome with the incorporation of a numerical wrinkling model that removes such instabilities.

In this example, a torque is applied to the inner edge of a circular annulus-shaped membrane while the outer edge is fixed. This causes wrinkling due to torsion. The example has been studied by different researchers (see Ref. 1, Ref. 2, Ref. 3, and Ref. 4). The results are similar albeit with some differences. The mesh pattern and discretization used in these research papers are different even though the same geometry, material, and boundary conditions are used. In this example, the model is set up in four different ways with respect to mesh pattern and discretization order, corresponding to each of the research papers.

## *Model Definition*

The geometry consists of a thin circular membrane with an outer radius of 12.5 m and an inner radius of 5 m. it is fixed on the outer edge. A prescribed rotation of 10 degrees is applied on the inner edge.

The linear elastic material is modeled using both isotropic and orthotropic properties. The linear stress-strain relationship defined in Ref. 1 on compliance form reads

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} \dfrac{1}{E_1} & \dfrac{-\nu_{12}}{E_2} & 0 \\ \dfrac{-\nu_{12}}{E_2} & \dfrac{1}{E_2} & 0 \\ 0 & 0 & \dfrac{1}{G_{12}} \end{bmatrix}^{-1} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ 2\varepsilon_{xy} \end{bmatrix} \tag{1}$$

For the isotropic case, Young's modulus is $E = 100$ kPa and Poisson's ratio is $\nu = 0.3$. Note that in Ref. 4, Poisson's ratio is reported as $\nu = 0.45$. This is assumed to be a typing mistake, since they refer to Ref. 1 for material properties.

In the orthotropic case, the Young's moduli are $E_{11} = 100$ kPa and $E_{22}$=1000 kPa, Poisson's ratio is $\nu_{12} = 0.3$, and the shear modulus is $G_{12} = 38.5$ kPa. The elements of the elasticity matrix in the linear stress strain relationship $\sigma = C\,\varepsilon$ are given in Table 1.

### MATERIAL PROPERTIES

TABLE 1: MATERIAL PROPERTIES.

| Property | Variable | Isotropic Model | Orthotropic Model |
|---|---|---|---|
| Elasticity matrix, 11 element | $C_{11}$ | 109.89 kPa | 100.91 kPa |
| Elasticity matrix, 12 element | $C_{12}$ | 32.967 kPa | 30.272 kPa |
| Elasticity matrix, 22 element | $C_{22}$ | 109.89 kPa | 1000.91 kPa |
| Elasticity matrix, 33 element | $C_{33}$ | 76.923 kPa | 77 kPa |

The constitutive relation for a linear elastic material used in COMSOL Multiphysics is

$$[\sigma]_{3\times 3} = [D]_{6\times 6} : [\varepsilon]_{3\times 3}$$

where the elements of $D$ are related to the elements in $C$ as

$$D_{11} = C_{11}, D_{12} = C_{12}, D_{22} = C_{22}, D_{33} = C_{22}, D_{44} = \frac{C_{33}}{2}, D_{55} = \frac{C_{33}}{2}, D_{66} = \frac{C_{33}}{2}$$

It is assumed that the stiffness in the third direction is the same as in the second direction, since the out-of-plane data is not available in Ref. 1. The elements $D_{33}$, $D_{55}$, and $D_{66}$, cannot be equal to zero because the Membrane interface includes degrees of freedom for the out-of-plane strains. However, the choice of out-of-plane data has no effect on the results.

Although the material data in principle can be entered using the **Isotropic** or **Orthotropic** options in the **Linear Elastic Material** subnode, the elasticity matrix computed in COMSOL Multiphysics does not match the values given in Ref. 1 even for identical Young's modulus, Poisson's ratio, and shear modulus. The reason for the discrepancy is the difference between three-dimensional and two-dimensional membrane theory. The authors in Ref. 1 used two-dimensional membrane theory where the thickness direction is not part of the constitutive model, while COMSOL Multiphysics employs a full three-dimensional representation where the membrane thickness enters explicitly. This gives a different elasticity matrix for the same material data. In order to match the values in Ref. 1, the elasticity matrix is first computed, and then these values are entered under the **Anisotropic** option in the **Linear Elastic Material** node.

### FINITE ELEMENT MESH

In Ref. 1 and Ref. 4, three-noded triangular mesh elements are used, but the mesh patterns differs. In Ref. 3, four-noded quadrilateral elements are used with 120 mesh elements, while Ref. 2 uses nine-noded quadrilateral elements with 180 mesh elements. This gives four different mesh pattern to investigate, as shown in Figure 1, Figure 2, Figure 3, and Figure 4.

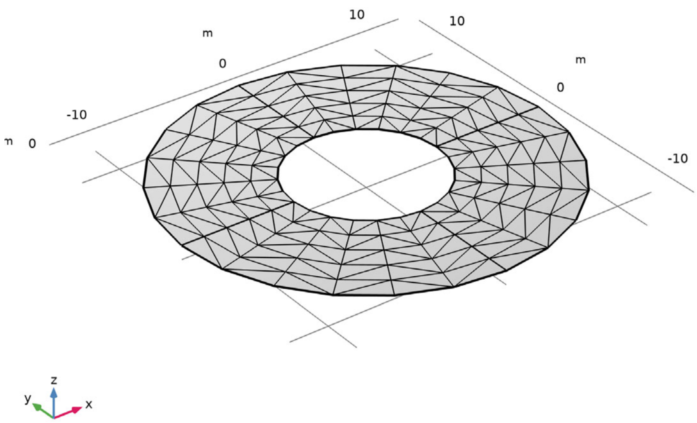


*Figure 1: The triangular mesh with 240 mesh elements from* Ref. 1.

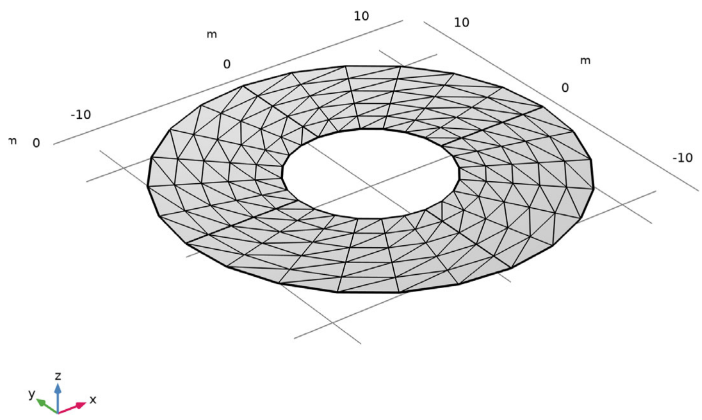


*Figure 2: The triangular mesh with 240 elements from* Ref. 4.

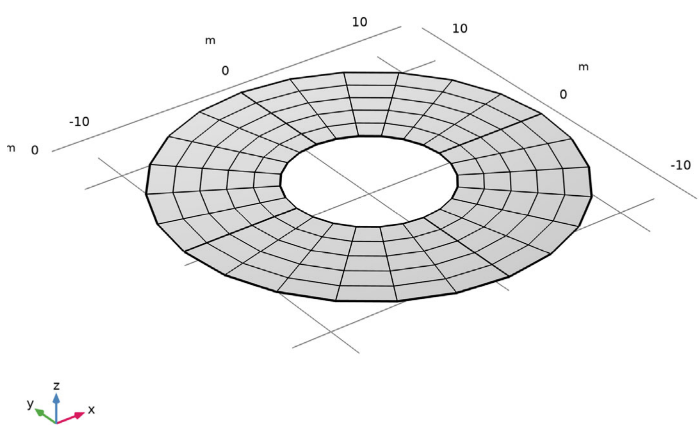


*Figure 3: The quadrilateral mesh with 120 elements from* Ref. 3.

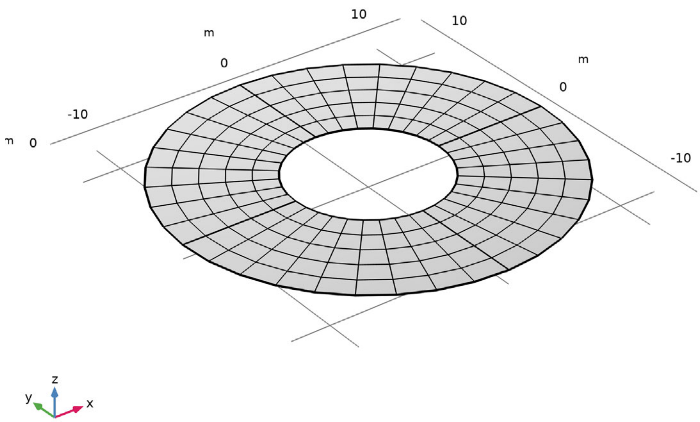


*Figure 4: The quadrilateral mesh with 180 elements from* Ref. 2.

## *Results and Discussions*

In all the following figures, four plots corresponding to four combinations of mesh pattern and discretization are presented. The order of the plots is the following and matches the legends position when multiple legends are used:

- Upper left — Three-noded triangular elements as shown in Figure 1
- Lower left — Three-noded triangular elements as shown in Figure 2
- Upper right — Four-noded quadrilateral elements as shown in Figure 3
- Lower right — Nine-noded quadrilateral elements as shown in Figure 4

Another point to note is that in all the following figures, **Resolution** and **Smoothing** in the plot quality section are set to **No Refinement** and **None** respectively, in order to represent the computed values as close as possible. Using smoothing would give significantly better plots, but it would hide some interesting features in some of the solutions.

Figure 5 and Figure 6 show the wrinkled regions for isotropic and orthotropic materials, respectively. For the isotropic case, the wrinkled region is symmetric and spreads in almost all elements except the outermost ones, close to the fixed outer boundary. For the orthotropic case, the wrinkled region is not symmetric, and it is localized near the inner edge. Depending on the selected mesh pattern and discretization, the wrinkling region varies significantly.

The first principal stress and the tensile direction for the isotropic and orthotropic cases are shown in Figure 7 and Figure 8, respectively. Figure 9 and Figure 10 show the second principal stress along the wrinkling direction. The lowest value of the second principal stress is almost zero, as is expected since the membrane cannot sustain compressive stresses. The principal stress pattern and directions match those presented in Ref. 1.

The maximum values of the first principal stress obtained with COMSOL Multiphysics for the different cases are given in Table 2 and Table 3. Some papers did not provide the values of maximum first principal stress explicitly. These are marked as Not Available (NA). When only the maximum value of the first principal stress is concerned, the worst match between reference values and COMSOL values is found for the three-noded triangular elements presented in Ref. 4, while a good match is found for three-noded triangular elements presented in Ref. 1. The COMSOL values for the mesh pattern and discretizations presented in Ref. 2 and Ref. 3 are in line with target values.

It is clear that the results are highly mesh and discretization dependent. In particular, the linear elements are performing badly in this example. The four-noded quadrilateral elements show a triangular stress pattern similar to the three-noded triangular elements, see Figure 7 and Figure 8. It is clear that the nine-noded quadrilateral elements are performing better considering the obtained maximum values as well as the stress and wrinkling pattern.

TABLE 2: MAXIMUM PRINCIPAL STRESS FOR ISOTROPIC MATERIAL.

| From | $\sigma_{p1}$ from Ref. 1 | $\sigma_{p1}$ from Ref. 2 | $\sigma_{p1}$ from Ref. 3 | $\sigma_{p1}$ from Ref. 4 |
|---|---|---|---|---|
| Reference | 26.9E3 | 26E3 | NA | 25.9E3 |
| COMSOL | 25.90E3 | 22.88E3 | 26.9E3 | 17.48E3 |

TABLE 3: MAXIMUM PRINCIPAL STRESS FOR ORTHOTROPIC MATERIAL.

| From | $\sigma_{p1}$ from Ref. 1 | $\sigma_{p1}$ from Ref. 2 | $\sigma_{p1}$ from Ref. 3 | $\sigma_{p1}$ from Ref. 4 |
|---|---|---|---|---|
| Reference | 270.5E3 | NA | NA | 230E3 |
| COMSOL | 235.7E3 | 247E3 | 296.5E3 | 163.4E3 |

The maximum values of wrinkling measure obtained with COMSOL Multiphysics are compared to the reference values in Table 4. The agreement between reference values and computed value are good.

TABLE 4: MAXIMUM VALUES OF WRINKLING MEASURE FOR ISOTROPIC MATERIAL.

| **From** | **β in material frame from** Ref. 3 |
|---|---|
| Reference | 0,19118 |
| COMSOL | 0.20824 |

The choice of mesh pattern shown in Figure 2 seems not desirable for this example. The physics is symmetric while the mesh is antisymmetric. If the counterclockwise torque (same as all references) is replaced by a clockwise torque, then the results are the same for all mesh and discretization patterns except the one used in Ref. 4. With clockwise torque the maximum first principal stress for isotropic and orthotropic material reached 26.03 kPa and 270.23 kPa, respectively. This can be compared with 17.48 kPa and 163.4 kPa obtained with counterclockwise torque. It can be noted that these values match what is reported in Ref. 4 much better.

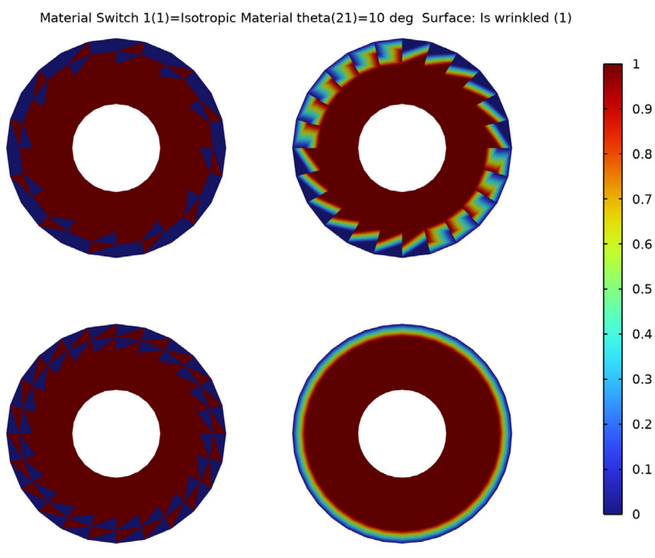


*Figure 5: Wrinkled region with isotropic material for different mesh types.*

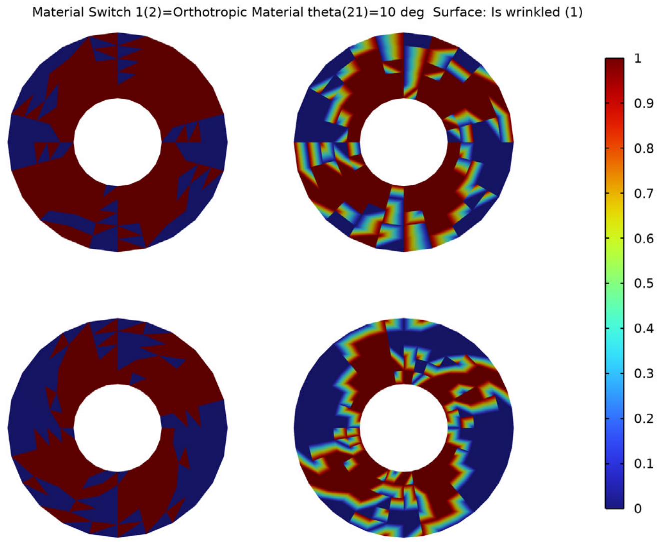


*Figure 6: Wrinkled region with orthotropic material for different mesh types.*

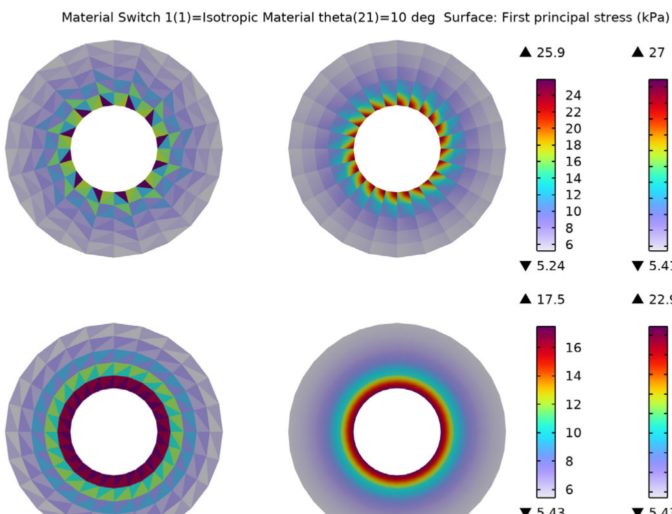


*Figure 7: First principal stress with isotropic material for different mesh types.*

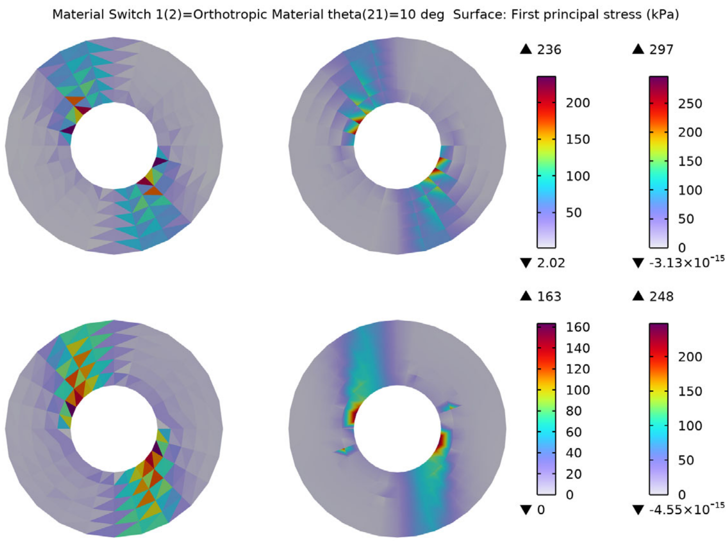


*Figure 8: First principal stress with orthotropic material for different mesh types.*

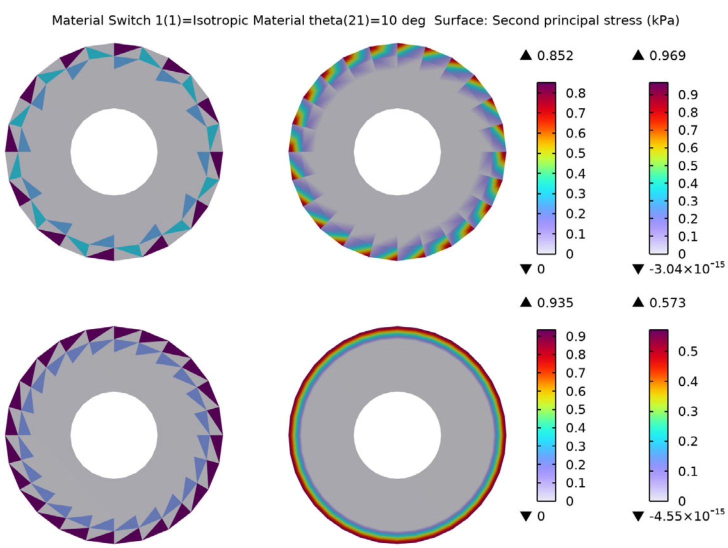


*Figure 9: Second principal stress with isotropic material for different mesh types.*

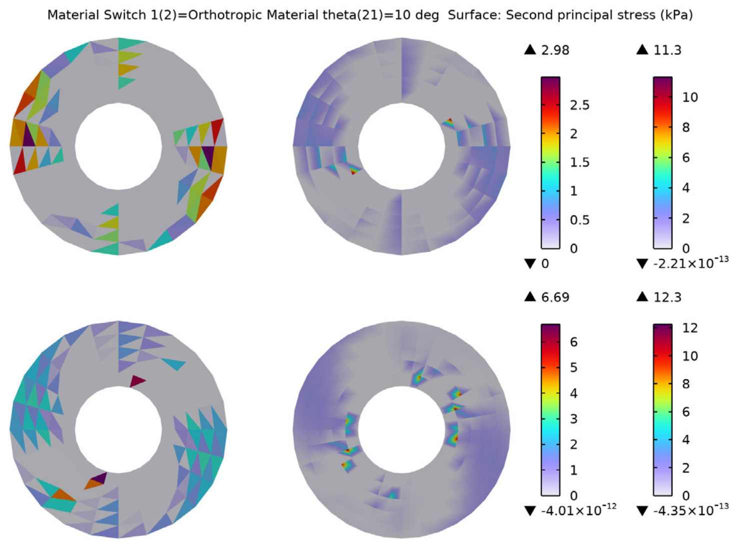


*Figure 10: Second principal stress with orthotropic material for different mesh types.*

## 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 using 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.

## References

1. D.G. Roddeman, “Finite element analysis of wrinkling membranes,” *Commun. Numer. Methods Eng.*, vol. 7, pp. 299–307, 1991.

2. K. Lu, M. Accorsi, and J. Leonard, “Finite element analysis of membrane wrinkling,” *Int. J. Numer. Methods Eng.*, vol. 50, pp. 1017–1038, 2000.

3. H. Schoop, L. Taenzer, and J. Homig, “Wrinkling of nonlinear membranes,” *Comput. Mech.*, vol. 29, pp. 68–74, 2002.

4. A. Jarasjarungkiat, R. Wuchner, and K.U. Bletzinger, “A wrinkling model based on material modification for isotropic and orthotropic membranes,” *Comput. Methods Appl. Mech. Eng.*, vol. 197, pp. 773–788, 2008.

---

**Application Library path:** `Structural_Mechanics_Module/Buckling_and_Wrinkling/membrane_torsion`

---

## *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_torsion_parameters.txt`.

#### *Isotropic Material Properties*

1 In the **Home** toolbar, click Pi **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_torsion_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_torsion_orthotropic_properties.txt`.

For this model four different mesh patterns are used. Import the meshed geometries to save modeling time.

### TRIANGULAR MESH, PATTERN 1

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

2 In the **Settings** window for **Mesh**, type `Triangular Mesh, Pattern 1` in the **Label** text field.

*Import 1*

1 In the **Mesh** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `membrane_torsion_tria_mesh1.mphbin`.

5 Click **Import**.

### TRIANGULAR MESH, PATTERN 2

1 In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

2 In the **Settings** window for **Mesh**, type `Triangular Mesh, Pattern 2` in the **Label** text field.

3 In the **Mesh** toolbar, click **Import**.

1 In the **Settings** window for **Import**, locate the **Import** section.

2 Click **Browse**.

3 Browse to the model's Application Libraries folder and double-click the file `membrane_torsion_tria_mesh2.mphbin`.

4 Click **Import**.

*Finalize*

In the **Model Builder** window, right-click **Finalize** and choose **Build All**.

## QUADRILATERAL MESH, PATTERN 1

1 In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

2 In the **Settings** window for **Mesh**, type `Quadrilateral Mesh, Pattern 1` in the **Label** text field.

*Import 1*

1 In the **Mesh** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `membrane_torsion_quad_mesh1.mphbin`.

5 Click **Import**.

*Finalize*

In the **Model Builder** window, right-click **Finalize** and choose **Build All**.

## QUADRILATERAL MESH, PATTERN 2

1 In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

2 In the **Settings** window for **Mesh**, type `Quadrilateral Mesh, Pattern 2` in the **Label** text field.

*Import 1*

1 In the **Mesh** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `membrane_torsion_quad_mesh2.mphbin`.

5 Click **Import**.

*Finalize*

In the **Model Builder** window, right-click **Finalize** and choose **Build All**.

The first study runs with linear shape order of the elements. Change the default **Quadratic** discretization to **Linear**.

## MEMBRANE (MBRN)

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

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

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

### *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 **Anisotropic**.

### *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**.

### *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 Edges 2, 3, 7, and 11 only.

### *Prescribed Displacement 1*

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

2 Select Edges 4, 5, 8, and 10 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 `(R11-1)*X+R12*Y+R13*Z`.

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

7 In the $u_{0y}$ text field, type `R21*X+(R22-1)*Y+R23*Z`.

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

**9** In the $u_{0z}$ text field, type `R31*X+R32*Y+(R33-1)*Z`.

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] |

The fourth study runs with quadratic shape order of the elements, so add a **Discretization** node. To do so, you first need to activate advanced physics options.

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

**7** In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

**8** Click **OK**.

*Quadratic Discretization*

**1** In the **Physics** toolbar, click **Global** and choose **Discretization**.

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

**3** From the **Displacement field** list, choose **Quadratic Lagrange**.

**4** In the **Label** text field, type `Quadratic Discretization`.

Add a **Material Switch** node in order to run the 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**.

**2** In the **Settings** window for **Material**, type `Isotropic 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 |
|---|---|---|---|---|
| Elasticity matrix | {D11, D12, D22, D13, D23, D33, D14, D24, D34, D44, D15, D25, D35, D45, D55, D16, D26, D36, D46, D56, D66} ; Dij = Dji | `{DD11_iso, DD12_iso, DD22_iso, 0, 0, DD33_iso, 0, 0, 0, DD44_iso, 0, 0, 0, 0, DD55_iso, 0, 0, 0, 0, 0, DD66_iso}` | Pa | Anisotropic |
| Density | rho | 0 | kg/m³ | Basic |

*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 |
|---|---|---|---|---|
| Elasticity matrix | {D11, D12, D22, D13, D23, D33, D14, D24, D34, D44, D15, D25, D35, D45, D55, D16, D26, D36, D46, D56, D66} ; Dij = Dji | `{DD11_orth, DD12_orth, DD22_orth, 0, 0, DD33_orth, 0, 0, 0, DD44_orth, 0, 0, 0, 0, DD55_orth, 0, 0, 0, 0, 0, DD66_orth}` | Pa | Anisotropic |
| Density | rho | 0 | kg/m³ | Basic |

The first study is for three-noded triangular elements, so choose the first mesh pattern.

Customize the study settings in order to achieve a better convergence.

## STUDY: THREE NODED TRIANGULAR (PATTERN 1)

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

**2** In the **Settings** window for **Study**, type `Study: Three Noded Triangular (Pattern 1)` in the **Label** text field.

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

*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**.

*Step 1: Stationary*

**1** In the **Model Builder** window, 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 |
|---|---|---|
| theta (Rotation) | `range(0,0.5,10)` | `deg` |

*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: Three Noded Triangular (Pattern 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** From the **Predictor** list, choose **Linear**.

**6** In the **Model Builder** window, under **Study: Three Noded Triangular (Pattern 1)>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Fully Coupled 1**.

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

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

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

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

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

Add a second study for three-noded triangular elements having the second mesh pattern.

## ADD STUDY

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

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

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

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

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

## STUDY: THREE NODED TRIANGULAR (PATTERN 2)

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

2 In the **Settings** window for **Study**, type `Study: Three Noded Triangular (Pattern 2)` in the **Label** text field.

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

### *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**.

### *Step 1: Stationary*

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

2 In the **Settings** window for **Stationary**, click to expand the **Mesh Selection** section.

3 In the table, enter the following settings:

| Component | Mesh |
|---|---|
| Component 1 | Triangular Mesh, Pattern 2 |

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

5 Click **Add**.

6 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| theta (Rotation) | `range(0,0.5,10)` | `deg` |

Customize the study settings as was done in **Study 1** in order to achieve a better convergence.

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

Add a third study for four-noded quadrilateral elements having the third mesh pattern.

### ADD STUDY

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

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

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

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

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

### STUDY: FOUR NODED QUADRILATERAL (PATTERN 3)

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study: Four Noded Quadrilateral (Pattern 3)` in the **Label** text field.

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

*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**.

*Step 1: Stationary*

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

2 In the **Settings** window for **Stationary**, locate the **Mesh Selection** section.

3 In the table, enter the following settings:

| Component | Mesh |
|---|---|
| Component 1 | Quadrilateral Mesh, Pattern 1 |

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

5 Click **Add**.

6 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| theta (Rotation) | `range(0,0.5,10)` | `deg` |

Customize the study settings as was done in **Study 1** in order to achieve a better convergence.

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

Add a fourth study for nine-noded quadrilateral elements having the fourth mesh pattern.

### ADD STUDY

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

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

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

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

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

### STUDY: NINE NODED QUADRILATERAL (PATTERN 4)

1 In the **Model Builder** window, click **Study 4**.

2 In the **Settings** window for **Study**, type `Study: Nine Noded Quadrilateral (Pattern 4)` in the **Label** text field.

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

#### *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**.

#### *Step 1: Stationary*

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

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

5 From the **Discretization** list, choose **Quadratic Discretization**.

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

| Component | Mesh |
|---|---|
| Component 1 | Quadrilateral Mesh, Pattern 2 |

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

8 Click **Add**.

9 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| theta (Rotation) | `range(0,0.5,10)` | `deg` |

Customize the study settings as was done in **Study 1** in order to achieve a better convergence.

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: Three Noded Triangular (Pattern 1)/Parametric Solutions 1 (sol2)**.

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

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

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

7 From the **Array shape** list, choose **Square**.

#### *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 **Resolution** list, choose **No refinement**.

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

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

6 In the **Row index** text field, type `1`.

7 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 **Data** section.

3 From the **Dataset** list, choose **Study: Three Noded Triangular (Pattern 2)/Parametric Solutions 2 (sol6)**.

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

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

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

7 Locate the **Plot Array** section. In the **Row index** text field, type 0.

8 Right-click **Surface 2** and choose **Duplicate**.

### *Surface 3*

1 In the **Model Builder** window, click **Surface 3**.

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

3 From the **Dataset** list, choose **Study: Four Noded Quadrilateral (Pattern 3)/ Parametric Solutions 3 (sol10)**.

4 Locate the **Plot Array** section. In the **Row index** text field, type 1.

5 In the **Column index** text field, type 1.

6 Right-click **Surface 3** and choose **Duplicate**.

### *Surface 4*

1 In the **Model Builder** window, click **Surface 4**.

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

3 From the **Dataset** list, choose **Study: Nine Noded Quadrilateral (Pattern 4)/ Parametric Solutions 4 (sol14)**.

4 Locate the **Plot Array** section. In the **Row index** text field, type 0.

### *Wrinkled Region*

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

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

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

4 Find the **Layout** subsection. Clear the **Use parameter indicator for solution and phase** check box.

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

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

7 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

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

### *First Principal Stress*

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

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

3 Click to expand the **Window Settings** section. Locate the **Color Legend** section. From the **Position** list, choose **Right double**.

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

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

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

7 Click **OK**.

### *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 `mbrn.sp1`.

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

5 Locate the **Inherit Style** section. From the **Plot** list, choose **None**.

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

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

8 Click **OK**.

### *Surface 3*

1 In the **Model Builder** window, click **Surface 3**.

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

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

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

5 Locate the **Inherit Style** section. From the **Plot** list, choose **None**.

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

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

8 Click **OK**.

*Surface 4*

1 In the **Model Builder** window, click **Surface 4**.

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

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

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

5 Locate the **Inherit Style** section. From the **Plot** list, choose **None**.

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

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

8 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 **Color Legend** section.

3 Select the **Show maximum and minimum values** check box.

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

5 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**, locate the **Expression** section.

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

*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 `mbrn.sp2`.

*Surface 3*

1 In the **Model Builder** window, click **Surface 3**.

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

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

*Surface 4*

**1** In the **Model Builder** window, click **Surface 4**.

**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**.

*Second Principal Stress*

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

*Maximum Wrinkling Measure (Isotropic)*

**1** In the **Results** toolbar, click **Evaluation Group**.

**2** In the **Settings** window for **Evaluation Group**, type `Maximum Wrinkling Measure (Isotropic)` in the **Label** text field.

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

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

**5** From the **Parameter selection (theta)** list, choose **Last**.

*Surface Maximum 1*

**1** Right-click **Maximum Wrinkling Measure (Isotropic)** and choose **Maximum>Surface Maximum**.

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

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

**4** Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Membrane>Wrinkling>mbrn.lemm1.wr1.Beta - Wrinkling measure, material frame - 1**.

**5** Right-click **Surface Maximum 1** and choose **Duplicate**.

*Surface Maximum 2*

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

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

**3** From the **Dataset** list, choose **Study: Three Noded Triangular (Pattern 2)/Parametric Solutions 2 (sol6)**.

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

**5** From the **Parameter selection (theta)** list, choose **Last**.

6 Right-click **Surface Maximum 2** and choose **Duplicate**.

*Surface Maximum 3*

1 In the **Model Builder** window, click **Surface Maximum 3**.
2 In the **Settings** window for **Surface Maximum**, locate the **Data** section.
3 From the **Dataset** list, choose **Study: Four Noded Quadrilateral (Pattern 3)/ Parametric Solutions 3 (sol10)**.
4 Right-click **Surface Maximum 3** and choose **Duplicate**.

*Surface Maximum 4*

1 In the **Model Builder** window, click **Surface Maximum 4**.
2 In the **Settings** window for **Surface Maximum**, locate the **Data** section.
3 From the **Dataset** list, choose **Study: Nine Noded Quadrilateral (Pattern 4)/ Parametric Solutions 4 (sol14)**.

*Maximum Wrinkling Measure (Isotropic)*

1 In the **Model Builder** window, click **Maximum Wrinkling Measure (Isotropic)**.
2 In the **Settings** window for **Evaluation Group**, locate the **Transformation** section.
3 Select the **Transpose** check box.
4 In the **Maximum Wrinkling Measure (Isotropic)** toolbar, click **Evaluate**.

*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**.

*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**.

### *Surface 2*

1 In the **Model Builder** window, expand the **Wrinkled Region 1** node, then click **Surface 2**.

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

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

### *Surface 3*

1 In the **Model Builder** window, click **Surface 3**.

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

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

### *Surface 4*

1 In the **Model Builder** window, click **Surface 4**.

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

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

4 In the **Wrinkled Region 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**.

### *Surface 2*

1 In the **Model Builder** window, expand the **First Principal Stress 1** node, then click **Surface 2**.

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

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

### *Surface 3*

1 In the **Model Builder** window, click **Surface 3**.

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

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

### *Surface 4*

1 In the **Model Builder** window, click **Surface 4**.

2 In the **Settings** window for **Surface**, 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, under **Results>Orthotropic Material** 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**.

*Surface 2*

1 In the **Model Builder** window, expand the **Second Principal Stress 1** node, then click **Surface 2**.

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

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

*Surface 3*

1 In the **Model Builder** window, click **Surface 3**.

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

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

*Surface 4*

1 In the **Model Builder** window, click **Surface 4**.

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

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

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

*Maximum Wrinkling Measure (Isotropic)*

In the **Model Builder** window, under **Results** right-click **Maximum Wrinkling Measure (Isotropic)** and choose **Duplicate**.

*Maximum Wrinkling Measure (Orthotropic)*

1 In the **Model Builder** window, under **Results** click **Maximum Wrinkling Measure (Isotropic) 1**.

2 In the **Settings** window for **Evaluation Group**, type `Maximum Wrinkling Measure (Orthotropic)` in the **Label** text field.

3 Locate the **Data** section. From the **Material Switch 1** list, choose **Last**.

*Surface Maximum 2*

1 In the **Model Builder** window, expand the **Maximum Wrinkling Measure (Orthotropic)** node, then click **Surface Maximum 2**.

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

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

*Surface Maximum 3*

1 In the **Model Builder** window, click **Surface Maximum 3**.

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

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

*Surface Maximum 4*

1 In the **Model Builder** window, click **Surface Maximum 4**.

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

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

4 In the **Maximum Wrinkling Measure (Orthotropic)** toolbar, click **=** **Evaluate**.