



Model created in COMSOL Multiphysics 6.4

# Parametric Study on Cross-Ply Laminate Failure

## *Introduction*

---

Laminated composite shells made from graphite–epoxy fiber-reinforced plies are widely used in many applications due to their high strength-to-weight ratio. Evaluating their structural integrity under applied loads is essential for ensuring reliable designs.

This example demonstrates how to perform failure analysis of a laminated composite shells using the **Layered Shell** interface. The laminate is analyzed under multiple load cases and loaded to failure, considering in-plane stress states. Both classical and advanced failure criteria are described and compared.

This example analyzes a cross-ply laminate subjected to monotonic tension, monotonic compression, and biaxial tension to determine the failure loads as a function of the fiber angle. This work is inspired by a similar study presented in [Ref. 1](#).



Read more about the Composite Materials Module in COMSOL blog, [Introduction to the Composite Materials Module](#).

---

## *Model Definition*

---

The geometry is a rectangular plate measuring 450 mm in length and 400 mm in width. The laminate features a symmetric stacking sequence of four layers, each with a thickness of 0.10 mm. The fiber orientation of the cross-ply laminate is parameterized as illustrated in [Figure 1](#).

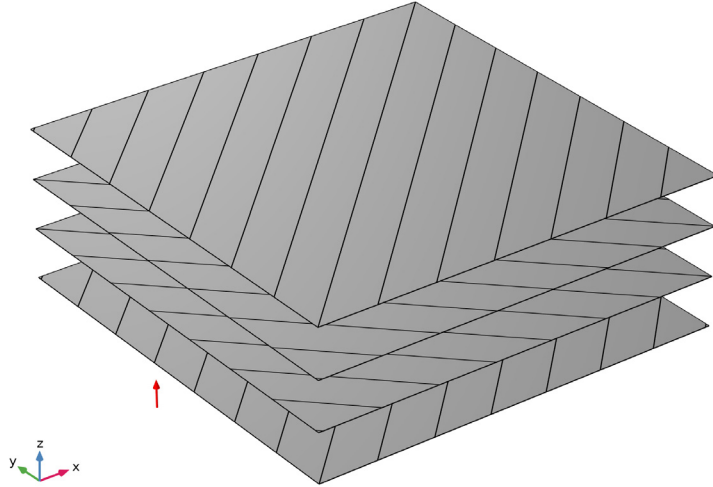


Figure 1: Stacking sequence from bottom to top, showing fiber orientation in each ply.

### MATERIAL PROPERTIES

All material properties and strengths are given in the layer coordinate system, where the first axis is aligned with the fiber orientation. The material parameters and failure coefficients are summarized in Table 1 and Table 2.

TABLE 1: ELASTIC MATERIAL PARAMETERS.

Elastic parameters	Value
$\{E_1, E_2\}$	{130.0, 11.0} GPa
$G_{12}$	5.0 GPa
$\{\nu_{12}, \nu_{23}\}$	{0.280, 0.350}

TABLE 2: FAILURE COEFFICIENTS.

Failure coefficients	Value
$\{\sigma_{ts1}, \sigma_{ts2}, \sigma_{ts3}\}$	{1700, 70.5, 70.5} MPa
$\{\sigma_{cs1}, \sigma_{cs2}, \sigma_{cs3}\}$	{1050, 240, 240} MPa
$\{\sigma_{ss23}, \sigma_{ss13}, \sigma_{ss12}\}$	{96,90,90} MPa

### BOUNDARY CONDITIONS

The laminate is subjected to three different load cases: monoaxial compression, monoaxial tension, and biaxial tension. For monoaxial compression and tension, Edge 1 is fixed in

the  $X$  and  $Z$  directions, and a boundary load is applied to Edge 3. For the biaxial tension case, Edge 1 is fixed in the  $X$  and  $Z$  directions, Edge 4 is fixed in the  $Y$  and  $Z$  directions, and a boundary load is applied to Edge 2 and Edge 3.

### FAILURE CRITERIA

Three failure criteria are employed to determine the failure loads: LaRC03, Tsai–Hill, and Hashin. The latter two are applied in their plane-stress formulations

#### LaRC03

This criterion is used to accurately predict the failure of unidirectional FRP laminates under in-plane stress. It is one of the most advanced failure models, comprising six phenomenological failure modes that describe matrix and fiber failure without relying on curve-fitting parameters. The model assumes brittle fracture for matrix compression failure and applies the action-plane concept from Mohr–Coulomb theory. It accounts for fiber kinking due to misalignment and tensile matrix cracking associated with interlaminar crack propagation.

The matrix failure criterion under transverse compression (LaRC03-1) is

$$g_{mC}(\sigma) = \left( \frac{\tau_{\text{eff},t}}{\sigma_{ss23}^i} \right)^2 + \left( \frac{\tau_{\text{eff},l}}{\sigma_{ss12}^i} \right)^2 - 1 \text{ for } \sigma_{11} \geq -\sigma_{cs2} \text{ and } \sigma_{22} < 0$$

where  $\tau_{\text{eff},t}$  and  $\tau_{\text{eff},l}$  are the effective shear stresses in the transverse and longitudinal directions, respectively,  $\sigma_{ss12}^i$  is the *in situ in-plane shear strength*,  $\sigma_{ss23}^i$  is the shear strength in the transverse direction, and  $\sigma_{cs2}$  is the compressive strength in the transverse direction. The effective shear stresses are functions of the fracture plane angle which is found by maximizing the Mohr–Coulomb effective stresses.

The matrix failure criterion under transverse tension (LaRC03-2) is

$$g_{mT}(\sigma) = (1-r) \left( \frac{\sigma_{22}}{\sigma_{ts2}^i} \right) + r \left( \frac{\sigma_{22}}{\sigma_{ts2}^i} \right)^2 + \left( \frac{\sigma_{12}}{\sigma_{ss12}^i} \right)^2 - 1 \text{ for } \sigma_{22} \geq 0$$

where  $\sigma_{ts2}^i$  is the *in situ transverse tensile strength*, and  $r$  is a dimensionless parameter based on fracture toughness.

The fiber failure criterion under longitudinal tension (LaRC03-3) is

$$g_{fT}(\varepsilon) = \frac{\varepsilon_{11}}{\varepsilon_{ts1}} - 1 \text{ for } \sigma_{11} \geq 0$$

The fiber failure criterion under matrix tension (LaRC03-4) is

$$g_{fCmT}(\sigma) = (1-r) \left( \frac{\sigma_{22}^m}{\sigma_{ts2}^i} \right) + r \left( \frac{\sigma_{22}^m}{\sigma_{ts2}^i} \right)^2 + \left( \frac{\sigma_{12}^m}{\sigma_{ss12}^i} \right)^2 - 1 \text{ for } \sigma_{22} \geq 0$$

The fiber failure criterion under matrix compression (LaRC03-5) is

$$g_{fCmC}(\sigma) = \left\langle \frac{|\sigma_{12}^m| + \sigma_{22}^m \eta_1}{\sigma_{ss12}^i} \right\rangle - 1 \text{ for } \sigma_{22} < 0$$

where  $\sigma_{ij}^m$  are the ply stresses transformed to the misalignment coordinate frame, and  $\eta_1$  is a nondimensional parameter based on the failure strength and fracture plane angle under uniaxial transverse compression.

The matrix failure criterion under biaxial compression (LaRC03-6) is

$$g_{mB}(\sigma) = \left( \frac{\tau_{eff,t}^m}{\sigma_{ss23}^i} \right)^2 + \left( \frac{\tau_{eff,l}^m}{\sigma_{ss12}^i} \right)^2 - 1 \text{ for } \sigma_{11} < -\sigma_{cs2} \text{ and } \sigma_{22} < 0$$

where the effective shear stresses in the transverse and longitudinal directions,  $\tau_{eff,t}^m$  and  $\tau_{eff,l}^m$ , are calculated from the stresses in the misaligned coordinates.

The failure criterion for the composite is then computed by selecting the most critical failure mode

$$g(\sigma) = \max(g_{mC}, g_{mT}, g_{fT}, g_{fCmT}, g_{fCmC}, g_{mB})$$

### Tsai-Hill

The Tsai-Hill criterion follows a polynomial description

$$g(\sigma) = \sigma_i F_{ij} \sigma_j - 1$$

where  $\sigma_i$  is the 6-by-1 stress vector (sorted using Voigt notation),  $F_{ij}$  is a 6-by-6 symmetric matrix that contains the coefficients for the quadratic terms. For the plane stress case, many of the coefficients are zero except for the following:

$$F_{11} = \frac{1}{\sigma_{ts1}^2} \quad \text{for } \sigma_{11} > 0, \text{ or } F_{11} = \frac{1}{\sigma_{cs1}^2} \quad \text{for } \sigma_{11} < 0$$

$$F_{22} = \frac{1}{\sigma_{ts2}^2} \quad \text{for } \sigma_{22} > 0, \text{ or } F_{22} = \frac{1}{\sigma_{cs2}^2} \quad \text{for } \sigma_{22} < 0$$

$$F_{66} = \frac{1}{\sigma_{ss12}^2}, F_{12} = -\frac{1}{2}(F_{11} + F_{22} - F_{33})$$

### Hashin

The Hashin failure theory is an extension of the Hashin–Rotem model that incorporates six failure modes, including fiber, matrix, and interlaminar failure, with distinctions between tension and compression. Stress interactions are considered in determining the tensile fiber failure mode, tensile matrix failure mode, and compressive matrix failure mode.

The fiber failure criterion in tension is defined by

$$g_{fT}(\sigma) = \frac{\sigma_{11}^2}{\sigma_{ts1}^2} + \frac{\sigma_{12}^2 + \sigma_{13}^2}{\sigma_{ss12}^2} - 1 \text{ for } \sigma_{11} \geq 0$$

The fiber failure criterion in compression is defined by

$$g_{fC}(\sigma) = \left| \frac{\sigma_{11}}{\sigma_{cs1}} \right| - 1 \text{ for } \sigma_{11} < 0$$

The matrix failure criterion in tension is defined by

$$g_{mT}(\sigma) = \frac{(\sigma_{22} + \sigma_{33})^2}{\sigma_{ts2}^2} + \frac{\sigma_{23}^2 - \sigma_{22}\sigma_{33}}{\sigma_{ss23}^2} + \frac{\sigma_{12}^2 + \sigma_{13}^2}{\sigma_{ss12}^2} - 1 \text{ for } \sigma_{22} + \sigma_{33} \geq 0$$

The matrix failure criterion in compression is defined by

$$g_{mC}(\sigma) = \left( \frac{\sigma_{cs2}^2}{4\sigma_{ss23}^2} - 1 \right) \frac{\sigma_{22} + \sigma_{33}}{\sigma_{cs2}} + \frac{(\sigma_{22} + \sigma_{33})^2}{4\sigma_{ss23}^2} + \frac{\sigma_{23}^2 - \sigma_{22}\sigma_{33}}{\sigma_{ss23}^2} + \frac{\sigma_{12}^2 + \sigma_{13}^2}{\sigma_{ss12}^2} - 1$$

for  $\sigma_{22} + \sigma_{33} < 0$

The interlaminar failure criterion in tension is defined by

$$g_{iT}(\sigma) = \frac{\sigma_{33}^2}{\sigma_{ts3}^2} - 1 \text{ for } \sigma_{33} \geq 0$$

The interlaminar failure criterion in compression is defined by

$$g_{iC}(\sigma) = \frac{\sigma_{33}^2}{\sigma_{cs3}^2} - 1 \text{ for } \sigma_{33} < 0$$

The failure criterion for the composite is computed by selecting the most critical failure mode

$$g(\sigma) = \max(g_{fT}, g_{fC}, g_{mT}, g_{mC}, g_{iT}, g_{iC})$$

The plane-stress version of the Hashin criterion is obtained by setting  $\sigma_{13} = \sigma_{23} = \sigma_{33} = 0$ . However, the interlaminar failure cannot then be predicted.

### Results and Discussion

The mechanical loading of the laminate is systematically increased until damage is detected, at which point the load is defined as the failure load. For each load case in this study, the plots in [Figure 2](#), [Figure 3](#), and [Figure 4](#) show the failure loads for the three failure criteria as functions of the fiber angle.

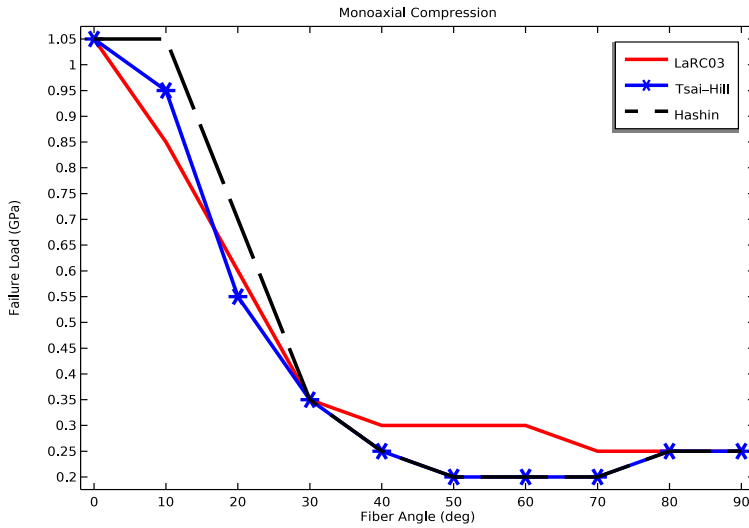


Figure 2: Monoaxial compression test.

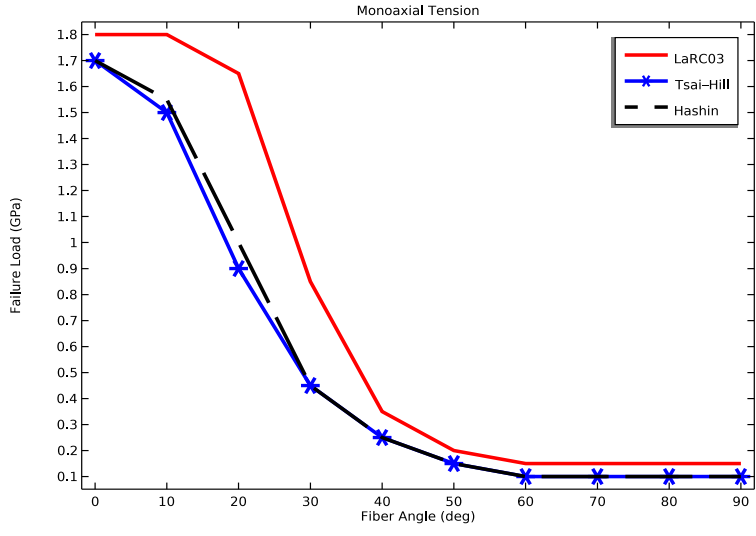


Figure 3: Monoaxial tension test.

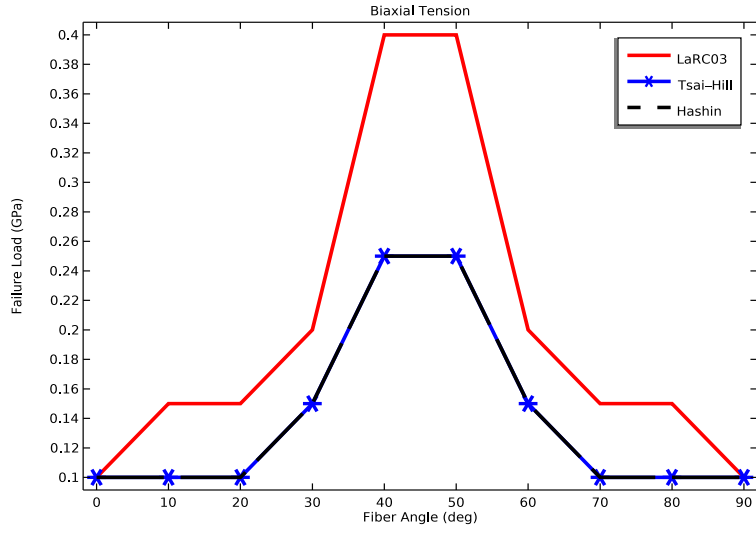


Figure 4: Biaxial tension test.

## *Notes About the COMSOL Implementation*

---

- Modeling a composite laminate as a layered shell requires a surface geometry, in general referred to as a base surface, and a **Layered Material** node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other having different thicknesses, material properties, and fiber orientations. You can optionally specify the interface materials between the layers, and control the number of through-thickness mesh elements for each layer.
- The **Layered Material Link** and **Layered Material Stack** have an option to transform given **Layered Material** into a symmetric or antisymmetric laminate. A repeated laminate can also be constructed using a transform option.
- The third direction for the selected coordinate system in the **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack** represents the normal direction of the **Layered Shell** or **Shell** physics. This is also the direction in which the layer stacking is interpreted from bottom to top, and therefore, it is crucial to know it during modeling. There are two ways to achieve this:
  - Using physics symbols: Go to the physics settings, find the **Physics Symbols** section, and select the **Enable physics symbols** checkbox. Then go to the material feature, for instance, **Linear Elastic Material**, to see the normal direction represented by green arrows in the geometry.
  - Using result templates: When a solution dataset is available, use the result template **Thickness and Orientation** to plot the normal direction.
- From a constitutive model point of view, you can either use the *Layerwise (LW)* theory based **Layered Shell** interface, or the *Equivalent Single Layer (ESL)* theory based **Linear Elastic Material, Layered** node in the **Shell** interface. The laminated composite presented in the current model is modeled using the **Layered Shell** interface.
- Since we are only interested in the failure loads, the **Stop Condition** node is included in our solver configurations to prevent the load from increasing further once damage is detected at the current fiber angle.

## *Reference*

---

I. P. Nali and E. Carrera, “A numerical assessment on two-dimensional failure criteria for composite layered structures,” *Composites Part B: Engineering*, vol. 43, no. 2, pp. 280–289, 2012; [doi.org/10.1016/j.compositesb.2011.06.018](https://doi.org/10.1016/j.compositesb.2011.06.018).

---

**Application Library path:** Composite\_Materials\_Module/  
Verification\_Examples/parametric\_study\_on\_crossply\_laminate\_failure


---

### *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 > Layered Shell (Ishell)**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies > Stationary**.
- 6 Click  **Done**.


#### **GLOBAL DEFINITIONS**

##### *Parameters I*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `parametric_study_on_crossply_laminate_failure_parameters.txt`.

Define the load case parameters.


##### *Load Case Parameters*

- 1 In the **Home** toolbar, click  **Parameters** and choose **Add > Parameters**.
- 2 In the **Settings** window for **Parameters**, type Load Case Parameters in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
xLoad	1	1	Load magnitude in x-direction
yLoad	0	0	Load magnitude in y-direction


In any node in the Model Builder, you can add comments to explain the settings. Right click on the node to select the **Properties and Comments** option to add the comment.

4 In the **Home** toolbar, click  **Parameter Case**.


5 In the **Settings** window for **Case**, type Monoaxial Compression in the **Label** text field.

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

Name	Expression	Description
xLoad	-1	Load magnitude in x-direction

7 In the **Home** toolbar, click  **Parameter Case**.

8 In the **Settings** window for **Case**, type Monoaxial Tension in the **Label** text field.

9 In the **Home** toolbar, click  **Parameter Case**.

10 In the **Settings** window for **Case**, type Biaxial Tension in the **Label** text field.

11 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Description
yLoad	1	Load magnitude in y-direction

In this model, three different failure criteria will be applied to determine failure. Define the relevant parameters to create safety cases for each criterion and to establish the stop condition in the study node.

### Safety Case


1 In the **Home** toolbar, click  **Parameters** and choose **Add > Parameters**.

2 In the **Settings** window for **Parameters**, type Safety Case in the **Label** text field.


3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
safety1	1	1	Used to turn on the LaRC03 criterion in the stop condition

Name	Expression	Value	Description
safety2	0	0	Used to turn on the Tsai-Hill criterion in the stop condition
safety3	0	0	Used to turn on the Hashin criterion in the stop condition

4 In the **Home** toolbar, click  **Parameter Case**.


5 In the **Settings** window for **Case**, type LaRC03 in the **Label** text field.

6 In the **Home** toolbar, click  **Parameter Case**.

7 In the **Settings** window for **Case**, type Tsai-Hill in the **Label** text field.

8 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Description
safety1	0	Used to turn on the LaRC03 criterion in the stop condition
safety2	1	Used to turn on the Tsai-Hill criterion in the stop condition

9 In the **Home** toolbar, click  **Parameter Case**.

10 In the **Settings** window for **Case**, type Hashin in the **Label** text field.

11 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Description
safety1	0	Used to turn on the LaRC03 criterion in the stop condition
safety3	1	Used to turn on the Hashin criterion in the stop condition

### *Unidirectional Fiber Composite*

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Model Builder** window, click **Material 1 (mat1)**.

3 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

4 In the **Material properties** tree, select **Basic Properties > Density**.

5 Click  **Add to Material**.

6 In the **Label** text field, type Unidirectional Fiber Composite.

7 Locate the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics > Linear Elastic Material > Transversely Isotropic**.

8 Click **+** **Add to Material**.

9 In the **Material properties** tree, select **Solid Mechanics > Strength Limits > Orthotropic Strength Parameters, Voigt Notation**.

10 Click **+** **Add to Material**.

11 In the **Model Builder** window, under **Global Definitions > Materials > Unidirectional Fiber Composite (mat1)** click **Basic (def)**.

12 In the **Settings** window for **Basic**, locate the **Output Properties** section.

13 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Density	rho	Rho	kg/m <sup>3</sup>	1x1

14 In the **Model Builder** window, under **Global Definitions > Materials > Unidirectional Fiber Composite (mat1)** click **Transversely isotropic (TransverselyIsotropic)**.

15 In the **Settings** window for **Transversely Isotropic**, locate the **Output Properties** section.

16 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Young's modulus	{Evect1, Evect2}	{E1, E2}	Pa	2x1
Poisson's ratio	{nuvect1, nuvect2}	{nu12, nu23}	1	2x1
Shear modulus	Gvect1	G	N/m <sup>2</sup>	1x1

17 In the **Model Builder** window, under **Global Definitions > Materials > Unidirectional Fiber Composite (mat1)** click **Orthotropic strength parameters, Voigt notation (OrthotropicStrengthParameters)**.

18 In the **Settings** window for **Orthotropic Strength Parameters, Voigt Notation**, locate the **Output Properties** section.

19 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Tensile strengths	{sigmats1, sigmats2, sigmats3}	{XT, YT, YT}	Pa	3x1
Compressive strengths	{sigmacs1, sigmacs2, sigmacs3}	{XC, YC, YC}	Pa	3x1

Property	Variable	Expression	Unit	Size
Shear strengths	{sigmass1, sigmass2, sigmass3}	{ST, SL, SL}	Pa	3x1
Ultimate tensile strain in longitudinal direction	epsilon1	epsT11	1	1x1

Now add a **Layered Material** node and assign appropriate thickness and rotation angles to each layer. The laminate is symmetric. It is sufficient to define only half the laminate in the **Layered Material** node. The transformation into a full laminate is performed through the layered material settings in the **Layered Material Link** node.

#### Laminate

- 1 In the **Model Builder** window, right-click **Materials** and choose **Layered Material**.
- 2 In the **Settings** window for **Layered Material**, type *Laminate* in the **Label** text field.
- 3 Locate the **Layer Definition** section. In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
Layer 1	Unidirectional Fiber Composite (mat1)	theta	0.7854 rad	th	1


- 4 Click  **Add**.

- 5 In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
Layer 2	Unidirectional Fiber Composite (mat1)	-theta	-0.7854 rad	th	1

## GEOMETRY I



### 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 (wpl) > 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 W.
- 4 In the **Height** text field, type H.
- 5 Click  **Build Selected**.

## DEFINITIONS

*Average 1 (aveop1)*

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Average**.
- 2 In the **Settings** window for **Average**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 From the **Selection** list, choose **All boundaries**.

Define the stop conditions in the **Variables** node.

*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
stopCondition1	safety1* aveop1(lshell.xdintopall(lshell.lemm1.sf1.d_i))>eps		Stop condition for LaRC03 criterion
stopCondition2	safety2* aveop1(lshell.xdintopall(lshell.lemm1.sf2.d_i))>eps		Stop condition for Tsai-Hill criterion
stopCondition3	safety3* aveop1(lshell.xdintopall(lshell.lemm1.sf3.d_i))>eps		Stop condition for Hashin criterion

## MATERIALS

### *Layered Material Link 1 (lmat1)*

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

The half laminate defined in the **Layered Material** node can be transformed into a full symmetric laminate using a transform option in the layered material settings.

- 2 In the **Settings** window for **Layered Material Link**, locate the **Layered Material Settings** section.
- 3 From the **Transform** list, choose **Symmetric**.
- 4 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type 0.5.
- 5 Locate the **Layered Material Settings** section. Click **Layer Cross-Section Preview** in the upper-right corner of the section.
- 6 Click **Layer Stack Preview** in the upper-right corner of the **Layered Material Settings** section.

## LAYERED SHELL (LSHELL)


Set the discretization for the displacement field to Linear in order to resemble the benchmark example.

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Layered Shell (lshell)**.
- 2 In the **Settings** window for **Layered Shell**, 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) > Layered Shell (lshell)** click **Linear Elastic Material 1**.
- 2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.
- 3 Select the **Transversely isotropic** checkbox.


### *Safety 1: LaRC03*

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Safety**.
- 2 In the **Settings** window for **Safety**, type Safety 1: LaRC03 in the **Label** text field.
- 3 Locate the **Failure Model** section. From the **Failure criterion** list, choose **LaRC03**.

### Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

#### Safety 2: Tsai–Hill

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Safety**.
- 2 In the **Settings** window for **Safety**, type Safety 2: Tsai-Hill in the **Label** text field.
- 3 Locate the **Failure Model** section. From the **Failure criterion** list, choose **Tsai–Hill**.
- 4 Select the **Use plane stress formulation** checkbox.


### Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

#### Safety 3: Hashin


- 1 In the **Physics** toolbar, click  **Attributes** and choose **Safety**.
- 2 In the **Settings** window for **Safety**, type Safety 3: Hashin in the **Label** text field.
- 3 Locate the **Failure Model** section. From the **Failure criterion** list, choose **Hashin**.
- 4 Select the **Use plane stress formulation** checkbox.

#### Boundary Load 1

- 1 In the **Physics** toolbar, click  **Edges** and choose **Boundary Load**.
- 2 Select Edge 4 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 Specify the  $\mathbf{f}_A$  vector as


xLoad\*loadPara x

#### Boundary Load 2

- 1 In the **Physics** toolbar, click  **Edges** and choose **Boundary Load**.
- 2 Select Edge 3 only.
- 3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
- 4 Specify the  $\mathbf{f}_A$  vector as


yLoad\*loadPara y

#### Prescribed Displacement 1

- 1 In the **Physics** toolbar, click  **Edges** and choose **Prescribed Displacement**.
- 2 Select Edge 1 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 From the **Displacement in z direction** list, choose **Prescribed**.

#### *Prescribed Displacement 2*

- 1 In the **Physics** toolbar, click  **Edges** and choose **Prescribed Displacement**.
- 2 Select Edge 2 only.
- 3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
- 4 From the **Displacement in y direction** list, choose **Prescribed**.
- 5 From the **Displacement in z direction** list, choose **Prescribed**.

### **MESH 1**

#### *Mapped 1*

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



#### *Distribution 1*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Edges 1 and 2 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 10.
- 5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

### **STUDY 1**

Set up three parametric sweeps: first for fiber angle, second for load cases, and third for failure criterion.


#### *Theta Sweep*

- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, type Theta Sweep in the **Label** text field.
- 3 Locate the **Study Settings** section. Click  **Add**.

4 In the table, enter the following settings:


Parameter name	Parameter value list	Parameter unit
theta (Fiber angle)	range (0, 10, 90)	deg

#### Load Case Switch

- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, type Load Case Switch in the **Label** text field.
- 3 Locate the **Study Settings** section. From the **Sweep type** list, choose **Parameter switch**.
- 4 Click **+ Add**.
- 5 In the table, enter the following settings:

Switch	Cases	Case numbers
Load Case Parameters	All	range(1,1,3)

#### Safety Case Switch

- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, type Safety Case Switch in the **Label** text field.
- 3 Locate the **Study Settings** section. From the **Sweep type** list, choose **Parameter switch**.
- 4 Click **+ Add**.
- 5 In the table, enter the following settings:


Switch	Cases	Case numbers
Safety Case	All	range(1,1,3)

#### 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** checkbox.
- 4 Click **+ Add**.
- 5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
loadPara (Load parameter)	range (0, 0.05, 1.8)	GPa

*Solution 1 (soll)*

1 In the **Study** toolbar, click  **Show Default Solver**.


2 In the **Model Builder** window, expand the **Solution 1 (soll)** node.

Define three stop conditions that will terminate the parametric solver and advance to the next fiber angle as soon as any one of them is satisfied. A condition is considered fulfilled when the surface-averaged damage index exceeds zero according to the selected failure criterion. The applicable condition depends on the current safety case.

3 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (soll) > Stationary Solver 1** node.


4 Right-click **Study 1 > Solver Configurations > Solution 1 (soll) > Stationary Solver 1 > Parametric 1** and choose **Stop Condition**.

5 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

6 Click  **Add**.

7 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
comp1.stopCondition1	True ( $\geq 1$ )	√	Stop if LaRC03 damage index is greater than zero

8 Click  **Add**.

9 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
comp1.stopCondition2	True ( $\geq 1$ )	√	Stop if Tsai-Hill damage index is greater than zero


10 Click  **Add**.

11 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
comp1.stopCondition3	True ( $\geq 1$ )	√	Stop if Hashin damage index is greater than zero

12 Locate the **Output at Stop** section. From the **Add solution** list, choose **Steps before and after stop**.

Disable the default plots for this study.

- 13 In the **Model Builder** window, click **Study 1**.
- 14 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 15 Clear the **Generate default plots** checkbox.
- 16 In the **Study** toolbar, click  **Compute**.

## RESULTS

Use evaluation groups to compute the failure loads and averaged damage index. Select the checkbox in the result node to enable automatic reevaluation of evaluation groups when the model is resolved.

- 1 In the **Model Builder** window, click **Results**.
- 2 In the **Settings** window for **Results**, locate the **Update of Results** section.
- 3 Select the **Reevaluate all evaluation groups after solving** checkbox.


### *Layered Material 1*

- 1 In the **Model Builder** window, expand the **Results** node.
- 2 Right-click **Results** > **Datasets** and choose **More Datasets** > **Layered Material**.
- 3 In the **Settings** window for **Layered Material**, locate the **Data** section.
- 4 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

### *Failure Load and Averaged Damage Index: LaRC03*

- 1 In the **Model Builder** window, right-click **Results** and choose **Node Group**.
- 2 In the **Settings** window for **Group**, type Failure Load and Averaged Damage Index: LaRC03 in the **Label** text field.

### *Monoaxial Compression, LaRC03*


- 1 In the **Results** toolbar, click  **Evaluation Group**.
- 2 In the **Settings** window for **Evaluation Group**, type Monoaxial Compression, LaRC03 in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.
- 4 From the **Safety Case, Load Case Parameters** list, choose **From list**.
- 5 In the **Safety Case, Load Case Parameters** list box, select **1: LaRC03, Monoaxial Compression**.
- 6 From the **Parameter selection (loadPara)** list, choose **Last**.
- 7 Click to expand the **Format** section. From the **Include parameters** list, choose **Off**.

### *Surface Average 1*


- 1 Right-click **Monoaxial Compression, LaRC03** and choose **Average > Surface Average**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.
- 4 In the table, enter the following settings:

<b>Expression</b>	<b>Unit</b>	<b>Description</b>
theta	deg	Fiber angle
loadPara	GPa	Load parameter
lshell.lemm1.sf1.d_i	1	LaRC03 damage index


### *Monoaxial Compression, LaRC03*

- 1 In the **Model Builder** window, click **Monoaxial Compression, LaRC03**.
- 2 In the **Monoaxial Compression, LaRC03** toolbar, click  **Evaluate**.

### *Monoaxial Tension, LaRC03*

- 1 Right-click **Monoaxial Compression, LaRC03** and choose **Duplicate**.
- 2 In the **Settings** window for **Evaluation Group**, type Monoaxial Tension, LaRC03 in the **Label** text field.
- 3 Locate the **Data** section. In the **Safety Case, Load Case Parameters** list box, select **2: LaRC03, Monoaxial Tension**.
- 4 In the **Monoaxial Tension, LaRC03** toolbar, click  **Evaluate**.


### *Biaxial Tension, LaRC03*

- 1 Right-click **Monoaxial Tension, LaRC03** and choose **Duplicate**.
- 2 In the **Settings** window for **Evaluation Group**, type Biaxial Tension, LaRC03 in the **Label** text field.
- 3 Locate the **Data** section. In the **Safety Case, Load Case Parameters** list box, select **3: LaRC03, Biaxial Tension**.
- 4 In the **Biaxial Tension, LaRC03** toolbar, click  **Evaluate**.

### *Failure Load and Averaged Damage Index: Tsai-Hill*

- 1 In the **Model Builder** window, right-click **Results** and choose **Node Group**.
- 2 In the **Settings** window for **Group**, type Failure Load and Averaged Damage Index: Tsai-Hill in the **Label** text field.

### *Monoaxial Compression, Tsai-Hill*

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


- 2 In the **Settings** window for **Evaluation Group**, type Monoaxial Compression, Tsai-Hill in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material I**.
- 4 From the **Safety Case, Load Case Parameters** list, choose **From list**.
- 5 In the **Safety Case, Load Case Parameters** list box, select **4: Tsai-Hill, Monoaxial Compression**.
- 6 From the **Parameter selection (loadPara)** list, choose **Last**.
- 7 Locate the **Format** section. From the **Include parameters** list, choose **Off**.

#### *Surface Average 1*


- 1 Right-click **Monoaxial Compression, Tsai-Hill** and choose **Average > Surface Average**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.
- 4 In the table, enter the following settings:

<b>Expression</b>	<b>Unit</b>	<b>Description</b>
theta	deg	Fiber angle
loadPara	GPa	Load parameter
lshell.lemm1.sf2.d_i	1	Tsai-Hill damage index, plane stress

#### *Monoaxial Compression, Tsai-Hill*

- 1 In the **Model Builder** window, click **Monoaxial Compression, Tsai-Hill**.
- 2 In the **Monoaxial Compression, Tsai-Hill** toolbar, click  **Evaluate**.

#### *Monoaxial Tension, Tsai-Hill*

- 1 Right-click **Monoaxial Compression, Tsai-Hill** and choose **Duplicate**.
- 2 In the **Settings** window for **Evaluation Group**, type Monoaxial Tension, Tsai-Hill in the **Label** text field.
- 3 Locate the **Data** section. In the **Safety Case, Load Case Parameters** list box, select **5: Tsai-Hill, Monoaxial Tension**.
- 4 In the **Monoaxial Tension, Tsai-Hill** toolbar, click  **Evaluate**.

#### *Biaxial Tension, Tsai-Hill*

- 1 Right-click **Monoaxial Tension, Tsai-Hill** and choose **Duplicate**.
- 2 In the **Settings** window for **Evaluation Group**, type Biaxial Tension, Tsai-Hill in the **Label** text field.

3 Locate the **Data** section. In the **Safety Case, Load Case Parameters** list box, select **6: Tsai–Hill, Biaxial Tension**.


4 In the **Biaxial Tension, Tsai–Hill** toolbar, click  **Evaluate**.

*Failure Load and Averaged Damage Index: Hashin*

1 In the **Model Builder** window, right-click **Results** and choose **Node Group**.

2 In the **Settings** window for **Group**, type Failure Load and Averaged Damage Index: Hashin in the **Label** text field.

*Monoaxial Compression, Hashin*

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

2 In the **Settings** window for **Evaluation Group**, type Monoaxial Compression, Hashin in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.

4 From the **Safety Case, Load Case Parameters** list, choose **From list**.

5 In the **Safety Case, Load Case Parameters** list box, select **7: Hashin, Monoaxial Compression**.

6 From the **Parameter selection (loadPara)** list, choose **Last**.

7 Locate the **Format** section. From the **Include parameters** list, choose **Off**.

*Surface Average 1*

1 Right-click **Monoaxial Compression, Hashin** and choose **Average > Surface Average**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.

4 In the table, enter the following settings:

<b>Expression</b>	<b>Unit</b>	<b>Description</b>
theta	deg	Fiber angle
loadPara	GPa	Load parameter
lshell.1emm1.sf3.d_i	1	Hashin damage index, plane stress


*Monoaxial Compression, Hashin*

1 In the **Model Builder** window, click **Monoaxial Compression, Hashin**.


2 In the **Monoaxial Compression, Hashin** toolbar, click  **Evaluate**.

*Monoaxial Tension, Hashin*

1 Right-click **Monoaxial Compression, Hashin** and choose **Duplicate**.


- 2 In the **Settings** window for **Evaluation Group**, type Monoaxial Tension, Hashin in the **Label** text field.
- 3 Locate the **Data** section. In the **Safety Case, Load Case Parameters** list box, select **8: Hashin, Monoaxial Tension**.
- 4 In the **Monoaxial Tension, Hashin** toolbar, click  **Evaluate**.

#### *Biaxial Tension, Hashin*

- 1 Right-click **Monoaxial Tension, Hashin** and choose **Duplicate**.
- 2 In the **Settings** window for **Evaluation Group**, type Biaxial Tension, Hashin in the **Label** text field.
- 3 Locate the **Data** section. In the **Safety Case, Load Case Parameters** list box, select **9: Hashin, Biaxial Tension**.
- 4 In the **Biaxial Tension, Hashin** toolbar, click  **Evaluate**.

Create ID plots to compare failure loads for each load cases based on different failure criterion.

#### *Monoaxial Compression*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Monoaxial Compression in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **None**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 5 In the **Title** text area, type Monoaxial Compression.
- 6 Locate the **Plot Settings** section.
- 7 Select the **x-axis label** checkbox. In the associated text field, type Fiber Angle (deg).
- 8 Select the **y-axis label** checkbox. In the associated text field, type Failure Load (GPa).
- 9 Locate the **Grid** section. Clear the **Show grid** checkbox.

#### *LaRC03*

- 1 Right-click **Monoaxial Compression** and choose **Table Graph**.
- 2 In the **Settings** window for **Table Graph**, type LaRC03 in the **Label** text field.
- 3 Locate the **Data** section. From the **Source** list, choose **Evaluation group**.
- 4 From the **x-axis data** list, choose **Fiber angle (deg)**.
- 5 From the **Plot columns** list, choose **Manual**.
- 6 In the **Columns** list box, select **Load parameter (GPa)**.

- 7 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.
- 8 From the **Width** list, choose **3**.
- 9 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 10 From the **Legends** list, choose **Manual**.
- 11 In the table, enter the following settings:

---

**Legends**

---

LaRC03

---

*Tsai-Hill*

- 1 Right-click **LaRC03** and choose **Duplicate**.
- 2 In the **Settings** window for **Table Graph**, type *Tsai-Hill* in the **Label** text field.
- 3 Locate the **Data** section. From the **Evaluation group** list, choose **Monoaxial Compression, Tsai-Hill**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 5 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.
- 6 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

---

*Tsai-Hill*

---

*Hashin*

- 1 Right-click **Tsai-Hill** and choose **Duplicate**.
- 2 In the **Settings** window for **Table Graph**, type *Hashin* in the **Label** text field.
- 3 Locate the **Data** section. From the **Evaluation group** list, choose **Monoaxial Compression, Hashin**.
- 4 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.
- 5 Find the **Line markers** subsection. From the **Marker** list, choose **None**.
- 6 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
- 7 Locate the **Legends** section. In the table, enter the following settings:

---

**Legends**

---

*Hashin*

---

*Monoaxial Compression*

- 1 In the **Model Builder** window, click **Monoaxial Compression**.

2 In the **Monoaxial Compression** toolbar, click  **Plot**.

#### *Monoaxial Tension*

- 1 Right-click **Monoaxial Compression** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Monoaxial Tension in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type Monoaxial Tension.

#### *LaRC03*

- 1 In the **Model Builder** window, expand the **Monoaxial Tension** node, then click **LaRC03**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **Evaluation group** list, choose **Monoaxial Tension, LaRC03**.


#### *Tsai–Hill*

- 1 In the **Model Builder** window, click **Tsai–Hill**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **Evaluation group** list, choose **Monoaxial Tension, Tsai–Hill**.

#### *Hashin*

- 1 In the **Model Builder** window, click **Hashin**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **Evaluation group** list, choose **Monoaxial Tension, Hashin**.

#### *Monoaxial Tension*

- 1 In the **Model Builder** window, click **Monoaxial Tension**.
- 2 In the **Monoaxial Tension** toolbar, click  **Plot**.

#### *Biaxial Tension*

- 1 Right-click **Monoaxial Tension** and choose **Duplicate**.
- 2 In the **Settings** window for **ID Plot Group**, type Biaxial Tension in the **Label** text field.
- 3 Locate the **Title** section. In the **Title** text area, type Biaxial Tension.

#### *LaRC03*

- 1 In the **Model Builder** window, expand the **Biaxial Tension** node, then click **LaRC03**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **Evaluation group** list, choose **Biaxial Tension, LaRC03**.

#### *Tsai–Hill*


- 1 In the **Model Builder** window, click **Tsai–Hill**.

- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **Evaluation group** list, choose **Biaxial Tension, Tsai–Hill**.

#### *Hashin*

- 1 In the **Model Builder** window, click **Hashin**.
- 2 In the **Settings** window for **Table Graph**, locate the **Data** section.
- 3 From the **Evaluation group** list, choose **Biaxial Tension, Hashin**.

#### *Biaxial Tension*

- 1 In the **Model Builder** window, click **Biaxial Tension**.
- 2 In the **Biaxial Tension** toolbar, click  **Plot**.