Model created in COMSOL Multiphysics 6.4

Failure Prediction in a Layered Shell

Introduction

Laminated shells made of carbon fiber reinforced polymer (CRFP) are common in a large variety of applications due to their high strength to weight ratio. Evaluation of the structural integrity of a laminated shell for a set of applied loads is necessary to make the design of such structures reliable.

This example shows how to model laminated shells using a **Linear Elastic Material** model in the Shell interfaces available with the Structural Mechanics Module. The same example can be modeled using a **Linear Elastic Material, Layered** model in the Shell interface. The model using the latter approach can be found in the Verification Examples folder of the Composite Materials Module Application Library.

The structural integrity of a stack of shells with different fiber orientations is assessed through the parameters called Failure Index and Safety Factor, using different polynomial failure criteria. Because of the orientation, each ply will have different strength in the longitudinal and transversal direction, and hence different response to the loading. The analysis using a polynomial failure criterion is termed *first ply failure analysis*, where failure in any ply is considered as failure of the whole laminate. In this example, seven different polynomial criteria are compared.

The original model is a NAFEMS benchmark model, described in *Benchmarks for Membrane and Bending Analysis of Laminated Shells, Part 2: Strength Analysis* (Ref. 1). The COMSOL Multiphysics solutions are compared with the reference data.

Model Definition

The physical geometry of the problem consists of four square shells stacked above each other. The side length is 1 cm and each layer has thickness of 0.05 mm. The laminate (90/

–45/45/0) is subjected to an in-plane axial tensile load. The actual geometry of the laminate is shown in Figure 1.

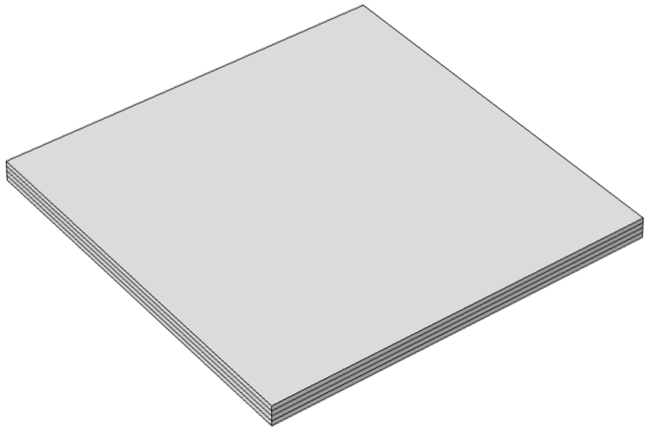

Figure 1: Geometry of layered shell with ply orientations 90/–45/45/0 from bottom to top.

MATERIAL PROPERTIES

The transversely isotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given in Table 1:

TABLE 1: MATERIAL PROPERTIES.

Material property	Value
$\{E_1, E_2, E_3\}$	{207, 7.6} (GPa)
G_{12}	5 (GPa)
$\{\upsilon_{12}, \upsilon_{23}\}$	{0. 3, 0}

The tensile, compressive, and shear strengths are given in Table 2.

TABLE 2: MATERIAL STRENGTHS IN MPA.

Material strengths	Value
$\{\sigma_{t1}, \sigma_{t2}, \sigma_{t3}\}$	{500, 5,5} (MPa)
$\{\sigma_{c1}, \sigma_{c2}, \sigma_{c3}\}$	{350, 75, 75} (MPa)
$\{\sigma_{ss23}, \sigma_{ss13}, \sigma_{ss12}\}$	{35, 35, 35} (MPa)

All material properties and strengths are given in the local material directions, where the first axis is aligned with the fiber orientation.

BOUNDARY CONDITIONS

The applied boundary conditions and loads on each node are given in the table below.

TABLE 3: NODE LOCATIONS AND BOUNDARY CONDITIONS.

Node	X (m)	Y (m)	Z (m)	Constrained DOF	Fx (N)	Fy (N)	Fz (N)
1 (1)	0	0	0	u, v, w, θ_x, θ_y, θ_z	0	0	0
2 (3)	0.01	0	0	θ_z	7.5	0	0
3 (4)	0.01	0.01	0	θ_z	7.5	0	0
4 (2)	0	0.01	0	u, θ_z	0	0	0

The numbers within parentheses are point numbers in COMSOL Multiphysics geometry. The boundary conditions provided in the benchmark specifications apply to the layered shell as a single entity. The rotation around the z-axis, θ_z, is automatically constrained so it does not need to be considered.

FAILURE CRITERIA

Six different failure criteria are used to predict the failure in the layered shell. These are Tsai–Wu anisotropic, Tsai–Wu orthotropic (plane stress version), Tsai–Hill (plane stress version), Hoffman, Azzi–Tsai–Hill, and Norris criteria.

The Hill criterion in Ref. 1 is called the Tsai–Hill criterion in COMSOL Multiphysics. For plane stress problems, a plane stress version of respective criteria must be used.

Ref. 1 does not give results for the Tsai–Wu anisotropic, Azzi–Tsai–Hill, and Norris criteria; so the analytical results for failure index and safety factor are here derived from the stress values given in Ref. 1.

The stresses from Ref. 1 are given in Table 4. Apart from σ_{11}, σ_{22}, and σ_{12}, all other stress components are either zero or negligible.

TABLE 4: STRESSES IN DIFFERENT PLIES.

Stresses	Ply 1	Ply 2	Ply 3	Ply 4
σ_{11} (MPa)	-5.128	12.59	8.520	9.357
σ_{22} (MPa)	4.407	1.983	0.125	-1.859
σ_{12} (MPa)	-1.663	2.572	-2.051	-0.5557

For all the selected polynomial criteria, the failure index (FI) is written as

$$\mathrm{FI} = \sigma_i F_{ij} \sigma_j + \sigma_i f_i \tag{1}$$

where σ_i is the 6-by-1 stress vector (sorted using Voigt notation), $\boldsymbol{F}_{ij}$ is a 6-by-6 symmetric matrix (fourth rank tensor) that contains the coefficients for the quadratic terms, and f_i is a 6-by-1 vector (second rank tensor) that contains the linear terms. A failure index equal to or greater than 1.0 indicates failure in the material. In order to find the safety factor SF, the applied stress in Equation 1 is multiplied by the safety factor SF, and the failure index FI is set equal to 1.0, which results in a quadratic equation of the form

$$a\,\mathrm{SF}^2 + b\,\mathrm{SF} = 1 \tag{2}$$

where $a = \sigma_i \boldsymbol{F}_{ij} \sigma_j$ and $b = \sigma_i f_i$.

The lowest positive root in Equation 2 is selected as the safety factor. Based on the stress values given in Table 4, the failure index and safety factor are computed for the criteria for which results in Ref. 1 are missing.

Tsai–Wu Anisotropic

For the Tsai–Wu anisotropic criterion, the material strength parameters are taken from Table 2 in order to obtain the same results as with the Tsai–Wu orthotropic criterion. This exercise is done in order to verify the correctness of the implementation. The nonzero elements in the second-rank tensor f are given below. Here, and in the following equations, repeated indices do not imply summation.

$$f_{ii} = \frac{1}{\sigma_{ti}} - \frac{1}{\sigma_{ci}};\quad i = 1, 2, 3 \tag{3}$$

The nonzero elements in the fourth rank tensor $\boldsymbol{F}$ are

$$\begin{aligned} &\boldsymbol{F}_{ii} = \frac{1}{\sigma_{ti}\sigma_{ci}};\quad i = 1, 2, 3 \\ &\boldsymbol{F}_{44} = \frac{1}{\sigma_{ss23}^2},\quad \boldsymbol{F}_{55} = \frac{1}{\sigma_{ss13}^2},\quad \boldsymbol{F}_{66} = \frac{1}{\sigma_{ss12}^2} \\ &\boldsymbol{F}_{ij} = -\frac{1}{2}\left(\sqrt{\boldsymbol{F}_{ii}\boldsymbol{F}_{jj}}\right);\quad i = 1, 2, 3 \end{aligned} \tag{4}$$

For the Tsai–Wu anisotropic criterion, the nonzero elements of the vector f_i and the matrix $\boldsymbol{F}_{ij}$ are given by Equation 3 and Equation 4. By taking values of stresses from Table 4, the failure index and safety factor are computed from Equation 1 and Equation 2, and given in Table 5 below.

TABLE 5: ANALYTIC VALUES OF FAILURE INDEX AND SAFETY FACTOR FOR TSAI–WU ANISOTROPIC CRITERION.

Index	Ply 1	Ply 2	Ply 3	Ply 4
FI	0.8840	0.3730	0.0199	-0.34309
SF	1.122	2.536	14.30	31.88

Azzi–Tsai–Hill

For the Azzi–Tsai–Hill criterion, all elements of the vector f_i are zero, while the nonzero elements of the matrix F_{ij} are given by Equation 5.

$$\begin{cases} \sigma_i \geq 0: & \left(F_{ii} = \dfrac{1}{\sigma_{ti}^2}\right) \\ \sigma_i < 0: & \left(F_{ii} = \dfrac{1}{\sigma_{ci}^2}\right) \end{cases} ; \quad i = 1, 2$$

$$F_{66} = \frac{1}{\sigma_{ss12}^2} \tag{5}$$

$$\begin{cases} \sigma_1 \geq 0: & \left(F_{12} = -\dfrac{1}{2\sigma_{t1}^2}\right) \\ \sigma_1 < 0: & \left(F_{12} = -\dfrac{1}{2\sigma_{c1}^2}\right) \end{cases}$$

By taking values of the stresses from Table 4, the failure index and safety factor are computed from Equation 1, Equation 2, and Equation 5, and given in Table 6 below.

TABLE 6: ANALYTIC VALUES OF FAILURE INDEX AND SAFETY FACTOR FOR AZZI–TSAI–HILL CRITERION.

Index	Ply 1	Ply 2	Ply 3	Ply 4
FI	0.7796	0.1632	0.00435	0.00128
SF	1.132	2.474	15.15	27.87

Norris

For the Norris criterion, all elements of the vector f_i are zero, while the nonzero elements of the matrix F_{ij} are given by Equation 6.

$$
\begin{cases}
\sigma_i \geq 0: & \left(F_{ii} = \dfrac{1}{\sigma_{ti}^2}\right) \\
\sigma_i < 0: & \left(F_{ii} = \dfrac{1}{\sigma_{ci}^2}\right)
\end{cases} \quad ; \quad i = 1, 2
$$

$$
F_{66} = \frac{1}{\sigma_{ss12}^2}
$$

$$
F_{12} = -\frac{1}{2}(\sqrt{F_{11}F_{22}}) \tag{6}
$$

By taking values of the stresses from Table 4, the failure index and safety factor are computed from Equation 1, Equation 2, and Equation 6, and given in Table 7 below.

TABLE 7: ANALYTIC VALUES OF FAILURE INDEX AND SAFETY FACTOR FOR NORRIS CRITERION.

Index	**Ply 1**	**Ply 2**	**Ply 3**	**Ply 4**
FI	0.7923	0.1533	0.0039	0.00168
SF	1.126	2.553	15.95	24.38

Note that for the current model, failure index and safety factor are computed at the midplane of each shell interface. However, COMSOL Multiphysics actually computes failure index, safety factor, damage index, and margin of safety at bottom, middle, and top surfaces of the shell, as well as the most critical of the three values.

Results and Discussion

The computed stresses are shown in Table 4, while Table 5 through Table 7 show the analytical values for failure index and safety factor (reserve factor) for certain failure criteria. For the Tsai–Wu orthotropic (plane stress version), Tsai–Hill (plane stress version), and Hoffman criteria, the failure index and safety factor are taken from Ref. 1. The results are compared with results from COMSOL Multiphysics.

TABLE 8: COMPARISON OF STRESSES FOR A LAYERED SHELL.

Ply	**σ_{11} (MPa), benchmark**	**σ_{11} (MPa), computed**	**σ_{22} (MPa), benchmark**	**σ_{22} (MPa), computed**	**σ_{12} (MPa), benchmark**	**σ_{12} (MPa), computed**
Ply 1	-5.128	-5.128	4.407	4.407	-1.663	-1.663
Ply 2	12.59	12.59	1.983	1.983	2.572	2.571
Ply 3	8.520	8.520	0.1256	0.1256	-2.051	-2.051
Ply 4	9.357	9.357	-1.859	-1.859	-0.5557	-0.5557

TABLE 9: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 1 (90 DEGREE PLY).

Criterion	**FI (benchmark or analytical)**	**FI, computed**	**SF (benchmark or analytical)**	**SF, computed**
Tsai–Wu orthotropic	0.8840	0.8841	1.122	1.1223
Hoffman	0.8811	0.8814	1.1253	1.1258
Tsai–Hill	0.7795	0.7796	1.1325	1.1325
Azzi–Tsai–Hill	0.7796	0.7796	1.132	1.1325
Norris	0.7923	0.7923	1.126	1.1234
Tsai–Wu anisotropic	0.8840	0.8841	1.122	1.1223

TABLE 10: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 2 (-45 DEGREE PLY).

Criterion	**FI (benchmark or analytical)**	**FI, computed**	**SF (benchmark or analytical)**	**SF, computed**
Tsai–Wu orthotropic	0.3730	0.3731	2.5367	2.5367
Hoffman	0.3763	0.3760	2.4944	2.4941
Tsai–Hill	0.1632	0.1632	2.4748	2.4748
Azzi–Tsai–Hill	0.1632	0.1632	2.474	2.4748
Norris	0.1533	0.1533	2.553	2.5534
Tsai–Wu anisotropic	0.37308	0.3731	2.536	2.5367

TABLE 11: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 3(45 DEGREE PLY).

Criterion	**FI (benchmark or analytical)**	**FI, computed**	**SF (benchmark or analytical)**	**SF, computed**
Tsai–Wu orthotropic	0.0199	0.01991	14.302	14.302
Hoffman	0.0200	0.02003	14.098	14.098
Tsai–Hill	0.0043	0.00435	15.157	15.157
Azzi–Tsai–Hill	0.0043	0.00435	15.15	15.157
Norris	0.0039	0.00392	15.95	15.954
Tsai–Wu anisotropic	0.0199	0.01991	14.30	14.302

TABLE 12: COMPARISON OF FAILURE INDEX (FI) AND SAFETY FACTORS (SF) FOR PLY 4 (0 DEGREE PLY).

Criterion	**FI (benchmark or analytical)**	**FI, computed**	**SF (benchmark or analytical)**	**SF, computed**
Tsai–Wu orthotropic	-0.3430	-0.3430	31.885	31.884
Hoffman	-0.3451	-0.3450	37.876	37.876
Tsai–Hill	0.00140	0.001359	27.12	27.124
Azzi–Tsai–Hill	0.00128	0.00128	27.87	27.877
Norris	0.00168	0.00168	24.38	24.388
Tsai–Wu anisotropic	-0.3430	-0.3430	31.88	31.884

For many industrial and real life applications, the safety factor (**SF**) is more useful than the failure index (**FI**). The safety factor (or reserve factor) gives a direct indication of how close the component is to failure. Figure 2 shows the Hoffman safety factor (**SF**) at the midplane for the different plies. Ply 1 (90-degree ply) is close to failure as expected because of its orientation, where fibers are perpendicular to the loading direction.

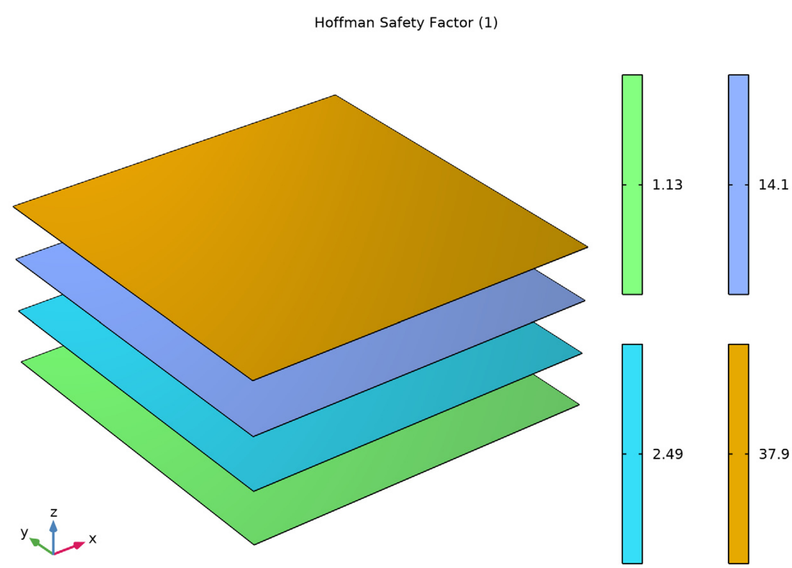

Figure 2: Hoffman safety factors at midplanes for a stack of shells.

The stress in the fiber direction in all plies are shown in Figure 3. The stress in ply 1 is the lowest in absolute value (and also compressive), but this layer is still more susceptible to failure due to the orientation of its fibers.

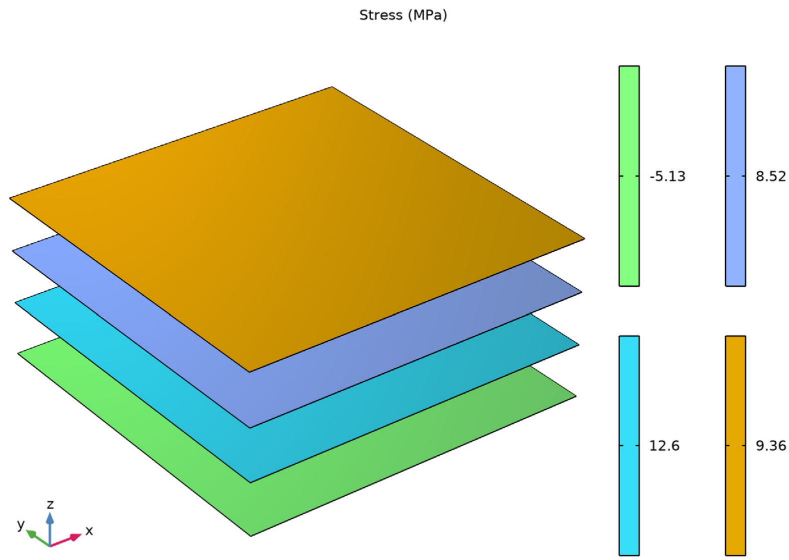

Figure 3: The stress in the fiber direction in a stack of shells.

Notes About the COMSOL Implementation

This layered shell is modeled using four separate Shell interfaces on top of each other. All four interfaces are located on the same boundary, and share the translational and rotational degrees of freedom. It is only the different values of the offset properties which describes the stacking.

The boundary conditions provided in the benchmark specifications apply to the layered shell as a single entity. When implemented in this model, special attention must be paid to the boundary condition stating that in one point, only the x-translation should be constrained. In the shell sense, this is a condition on the midsurface of the stack, which is between ply 2 and ply 3. Setting the degree of freedom u to zero, would in this case imply that also the rotation around the y-axis is constrained, since it would be applied on all layers. The intended boundary condition is instead implemented by stating that the x-displacement in ply 2 should be the negative of the x-displacement in ply 3.

Reference

1. P. Hopkins, *Benchmarks for Membrane and Bending Analysis of Laminated Shells, Part 2: Strength Analysis,* NAFEMS, 2005.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/failure_prediction_in_a_layered_shell`

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 > Shell (shell)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics > Shell (shell)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Structural Mechanics > Shell (shell)**.

7 Click **Add**.

8 In the **Select Physics** tree, select **Structural Mechanics > Shell (shell)**.

9 Click **Add**.

10 Click **Study**.

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

12 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Load the text file containing the material properties and material strengths.

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

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 `failure_prediction_in_a_layered_shell_material_properties.txt`.

DEFINITIONS

Set up three rotated coordinate systems.

Rotated System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for **Rotated System**, locate the **Rotation** section.

3 Find the **Euler angles** subsection. In the α text field, type `pi/2`.

Rotated System 3 (sys3)

1 Right-click **Rotated System 2 (sys2)** and choose **Duplicate**.

2 In the **Settings** window for **Rotated System**, locate the **Rotation** section.

3 Find the **Euler angles** subsection. In the α text field, type `-pi/4`.

Rotated System 4 (sys4)

1 Right-click **Rotated System 3 (sys3)** and choose **Duplicate**.

2 In the **Settings** window for **Rotated System**, locate the **Rotation** section.

3 Find the **Euler angles** subsection. In the α text field, type `pi/4`.

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Go to Plane Geometry**.

Work Plane 1 (wp1) > Plane Geometry

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

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

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

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

3 In the **Side length** text field, type `1e-2`.

4 Click **Build Selected**.

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

MATERIALS

Material 1 (mat1)

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

PLY 1

Activate **Advanced Physics** option from **Show** button.

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

2 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

3 Click **OK**.

The layered shell is modeled using four separate shell interfaces located on the same boundary (mesh surface), sharing the degrees of freedom. The stacking of the shells is done using a **Position** option in a **Thickness and Offset** feature. With this option the constraints and loads are transferred to the actual midplane of the shells without modeling it.

As the same degrees of freedom are to be shared by all shell interfaces, set the displacement field to **u** and the displacement of the shell normals to **ar** for Shell 2, Shell 3, and Shell 4.

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

The results given in the benchmark example are at the midplane of each shell layer. Set the **Default Through-Thickness Result Location** to zero for all shells.

4 In the **Settings** window for **Shell**, type `Ply 1` in the **Label** text field.

5 In the **Name** text field, type `shell1`.

6 Click to expand the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

7 Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Ply 1 (shell1)** 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`.

4 From the **Position** list, choose **User defined**.

5 In the $z_{\text{reloffset}}$ text field, type `-3`.

Linear Elastic Material 1

Choose the transversely isotropic solid model for the linear elastic material and assign **Rotated System 2** as **Shell Local System**.

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

4 Select the **Transversely isotropic** checkbox.

Shell Local System 1

1 In the **Model Builder** window, click **Shell Local System 1**.

2 In the **Settings** window for **Shell Local System**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

PLY 2

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell 2 (shell2)**.

2 In the **Settings** window for **Shell**, type `Ply 2` in the **Label** text field.

3 Locate the **Discretization** section. From the **Displacement field** list, choose **Linear**.

4 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

5 Click to expand the **Dependent Variables** section. In the **Displacement field (m)** text field, type `u`.

6 In the **Displacement of shell normals (1)** text field, type `ar`.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Ply 2 (shell2)** 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`.

4 From the **Position** list, choose **User defined**.

5 In the $z_{\text{reloffset}}$ text field, type `-1`.

Linear Elastic Material 1

Choose the transversely isotropic solid model for the linear elastic material and assign **Rotated System 3** as **Shell Local System**.

1 In the **Model Builder** window, 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**.
4 Select the **Transversely isotropic** checkbox.

Shell Local System 1

1 In the **Model Builder** window, click **Shell Local System 1**.
2 In the **Settings** window for **Shell Local System**, locate the **Coordinate System Selection** section.
3 From the **Coordinate system** list, choose **Rotated System 3 (sys3)**.

PLY 3

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell 3 (shell3)**.
2 In the **Settings** window for **Shell**, type `Ply 3` in the **Label** text field.
3 Locate the **Discretization** section. From the **Displacement field** list, choose **Linear**.
4 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.
5 Locate the **Dependent Variables** section. In the **Displacement field (m)** text field, type `u`.
6 In the **Displacement of shell normals (1)** text field, type `ar`.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Ply 3 (shell3)** 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`.
4 From the **Position** list, choose **User defined**.
5 In the $z_{\mathrm{reloffset}}$ text field, type `1`.

Linear Elastic Material 1

Choose the transversely isotropic solid model for the linear elastic material and assign **Rotated System 4** as **Shell Local System**.

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

4 Select the **Transversely isotropic** checkbox.

Shell Local System 1

1 In the **Model Builder** window, click **Shell Local System 1**.

2 In the **Settings** window for **Shell Local System**, locate the **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Rotated System 4 (sys4)**.

PLY 4

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell 4 (shell4)**.

2 In the **Settings** window for **Shell**, type `Ply 4` in the **Label** text field.

3 Locate the **Discretization** section. From the **Displacement field** list, choose **Linear**.

4 Locate the **Default Through-Thickness Result Location** section. In the z text field, type `0`.

5 Locate the **Dependent Variables** section. In the **Displacement field (m)** text field, type `u`.

6 In the **Displacement of shell normals (1)** text field, type `ar`.

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Ply 4 (shell4)** 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`.

4 From the **Position** list, choose **User defined**.

5 In the $z_{\mathrm{reloffset}}$ text field, type `3`.

Linear Elastic Material 1

Choose the transversely isotropic solid model for the linear elastic material.

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

4 Select the **Transversely isotropic** checkbox.

MATERIALS

Material 1 (mat1)

Select the material properties for the transversely isotropic material from Table 1.

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Material 1 (mat1)**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	{Evect1, Evect2}	`{E1, E2}`	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	`{nu12, nu23}`	1	Transversely isotropic
Shear modulus	{Gvect1}	`G`	N/m^2	Transversely isotropic
Density	rho	`1500`	kg/m^3	Basic

PLY 1 (SHELL1)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Ply 1 (shell1)** click **Linear Elastic Material 1**.

Safety: Tsai–Wu Orthotropic, Plane Stress Criterion

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

Safety 2, 3, 4, 5, 6, 7

1 Create six similar **Safety** nodes by duplicating the **Safety 1** node, and replace the failure criterion as given in the table below:

Name	Failure Criterion
Safety 2	`Hoffman`
Safety 3	`Tsai-Hill with Plane Stress option`
Safety 4	`Azzi-Tsai-Hill`
Safety 5	`Norris`
Safety 6	`Tsai-Wu anisotropic`

Select all **Safety** nodes under **Play 1 (shell1)** > **Linear Elastic Material 1**, and right-click to **Copy**. Then, go to **Linear Elastic Material 1** under **Play 2 (shell2)**, **Play 3 (shell3)**, and **Ply 4 (shell4)** and right-click to **Paste Multiple Items**.

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Tensile strengths	{sigmats1, sigmats2, sigmats3}	`{Sigmats1, Sigmats2, Sigmats3}`	Pa	Orthotropic strength parameters, Voigt notation
Compressive strengths	{sigmacs1, sigmacs2, sigmacs3}	`{Sigmacs1, Sigmacs2, Sigmacs3}`	Pa	Orthotropic strength parameters, Voigt notation
Shear strengths	{sigmass1, sigmass2, sigmass3}	`{Sigmass23, Sigmass13, Sigmass12}`	Pa	Orthotropic strength parameters, Voigt notation

Property	Variable	Value	Unit	Property group
Second rank tensor, Voigt notation	{F_s1, F_s2, F_s3, F_s4, F_s5, F_s6}	`{Fs11, Fs22, Fs33, 0, 0, 0}`	1/Pa	Anisotropic strength parameters, Voigt notation
Fourth rank tensor, Voigt notation	{F_f11, F_f12, F_f22, F_f13, F_f23, F_f33, F_f14, F_f24, F_f34, F_f44, F_f15, F_f25, F_f35, F_f45, F_f55, F_f16, F_f26, F_f36, F_f46, F_f56, F_f66} ; F_fij = F_fji	`{Ff11, Ff12, Ff22, Ff13, Ff23, Ff33, 0, 0, 0, Ff44, 0, 0, 0, 0, Ff55, 0, 0, 0, 0, 0, Ff66}`	$m^2 \cdot s^4/kg^2$	Anisotropic strength parameters, Voigt notation

PLY 1 (SHELL1)

Fixed Constraint 1

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

2 Select Point 1 only.

Apply a nodal tensile load of 15 N as an edge load. The load is shared by all shell midplanes, hence it is divided by 4 in order to keep a total value of 15 N.

Edge Load 1

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.

2 Select Edge 4 only.

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

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

`Ftotal/4`	x
0	y
0	z

Now select **Fixed Constraint** and **Edge Load** nodes under **Ply 1 (shell1)**, and right-click to **Copy**. Then go to **Ply 2 (shell2)**, **Ply 3 (shell3)**, and **Ply 4 (shell4)**; and right-click to **Paste Multiple Items**.

PLY 2 (SHELL2)

To enforce a fixed x direction translation on Node 2, apply the displacement `-u0` in the x direction to Point 2 of `shell2`, and the displacement `u0` in the x direction to the same point of `shell3`. Also add a **Global Equation** node under `shell2` for the additional degree of freedom `u0`.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Ply 2 (shell2)**.

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.
2 Select Point 2 only.
3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.
4 From the **Displacement in x direction** list, choose **Prescribed**.
5 In the u_{0x} text field, type `-u0`.
6 Click the **Show More Options** button in the **Model Builder** toolbar.
7 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.
8 Click **OK**.

Global Equations 1 (ODE1)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.
2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.
3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
u0		0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.
5 In the **Physical Quantity** dialog, type `displacement` in the text field.
6 In the tree, select **General** > **Displacement (m)**.
7 Click **OK**.

PLY 3 (SHELL3)

Prescribed Displacement/Rotation 1

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

2 Select Point 2 only.

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

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

5 In the u_{0x} text field, type `u0`.

MESH 1

Use a single quadrilateral element.

Free Quad 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.

2 Select Boundary 1 only.

Distribution 1

1 Right-click **Free Quad 1** and choose **Distribution**.

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

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type `1`.

5 Click **Build All**.

STUDY 1

Switch off the generation of default plots.

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

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

3 Clear the **Generate default plots** checkbox.

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Click **Apply**.

Cut Point 3D 1

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

2 Right-click **Results** > **Datasets** and choose **Cut Point 3D**.

3 In the **Settings** window for **Cut Point 3D**, locate the **Point Data** section.

4 In the **X** text field, type `0.5e-2`.

5 In the **Y** text field, type `0.5e-2`.

6 In the **Z** text field, type `0`.

Select the checkbox in the result node to enable automatic reevaluation of evaluation groups when the model is resolved.

7 In the **Model Builder** window, click **Results**.

8 In the **Settings** window for **Results**, locate the **Update of Results** section.

9 Select the **Reevaluate all evaluation groups after solving** checkbox.

Failure Indices in Ply 1

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

2 In the **Settings** window for **Evaluation Group**, type `Failure Indices in Ply 1` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

4 Locate the **Transformation** section. Select the **Transpose** checkbox.

Point Evaluation 1

1 Right-click **Failure Indices in Ply 1** and choose **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`shell1.emm1.sf1.f_im`	1	`Tsai-Wu orthotropic failure index, middle, plane stress`
`shell1.emm1.sf2.f_im`	1	`Hoffman failure index, middle`

Expression	Unit	Description
shell1.emm1.sf3.f_im	1	Tsai-Hill failure index, middle, plane stress
shell1.emm1.sf4.f_im	1	Azzi-Tsai-Hill failure index, middle
shell1.emm1.sf5.f_im	1	Norris failure index, middle
shell1.emm1.sf6.f_im	1	Tsai-Wu anisotropic failure index, middle

4 In the **Failure Indices in Ply 1** toolbar, click **Evaluate**.

Evaluation Group 2, 3, 4

Create three similar evaluation groups by duplicating the **Evaluation Group 1** node, and replace the word `shell1` in the **Expressions** by `shell2`, `shell3`, and `shell4` in **Point Evaluation** nodes in respective evaluation groups. Rename evaluation group nodes appropriately.

Select evaluation groups corresponding to the failure indices, and right click on **Group** to create a group.

Failure Indices

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

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

Safety Factors in Ply 1

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

2 In the **Settings** window for **Evaluation Group**, type `Safety Factors in Ply 1` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.

4 Locate the **Transformation** section. Select the **Transpose** checkbox.

Point Evaluation 1

1 Right-click **Safety Factors in Ply 1** and choose **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
shell1.emm1.sf1.s_fm	1	Tsai-Wu orthotropic safety factor, middle, plane stress
shell1.emm1.sf2.s_fm	1	Hoffman safety factor, middle

Expression	Unit	Description
shell1.emm1.sf3.s_fm	1	Tsai-Hill safety factor, middle, plane stress
shell1.emm1.sf4.s_fm	1	Azzi-Tsai-Hill safety factor, middle
shell1.emm1.sf5.s_fm	1	Norris safety factor, middle
shell1.emm1.sf6.s_fm	1	Tsai-Wu anisotropic failure index, middle

4 In the **Safety Factors in Ply 1** toolbar, click **Evaluate**.

Evaluation Group 6, 7, 8

Create three similar evaluation groups by duplicating the **Evaluation Group 5** node, and replace the word `shell1` in the **Expressions** by `shell2`, `shell3`, and `shell4` in **Point Evaluation** nodes in respective evaluation groups. Rename evaluation group nodes appropriately.

Select evaluation groups corresponding to the safety factors, and right click on **Group** to create a group.

Safety Factors

1. In the **Model Builder** window, under **Results** click **Group 2**.
2. In the **Settings** window for **Group**, type `Safety Factors` in the **Label** text field.

Stresses in Ply 1

1. In the **Results** toolbar, click **Evaluation Group**.
2. In the **Settings** window for **Evaluation Group**, type `Stresses in Ply 1` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.
4. Locate the **Transformation** section. Select the **Transpose** checkbox.

Point Evaluation 1

1. Right-click **Stresses in Ply 1** and choose **Point Evaluation**.
2. In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
shell1.Sl11	MPa	Second Piola-Kirchhoff stress, local coordinate system, 11 component

Expression	Unit	Description
`shell1.Sl22`	MPa	Second Piola-Kirchhoff stress, local coordinate system, 22 component
`shell1.Sl12`	MPa	Second Piola-Kirchhoff stress, local coordinate system, 12 component

4 In the **Stresses in Ply 1** toolbar, click **Evaluate**.

Evaluation Group 10, 11, 12

Create three similar evaluation groups by duplicating the **Evaluation Group 9** node, and replace the word `shell1` in the **Expressions** by `shell2`, `shell3`, and `shell4` in **Point Evaluation** nodes in respective evaluation groups, respectively. Rename evaluation group nodes appropriately.

Select evaluation groups corresponding to the stresses, and right click on **Group** to create a group.

Stresses

1 In the **Model Builder** window, under **Results** click **Group 3**.

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

To visualize stress in the fiber direction at the midplane of each ply, use four different **Layered Material Slice** plots and shift them in the z direction for better visualization. Use the **round** operator to get uniform color in each ply.

Stress (Ply)

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Stress (MPa)`.

5 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

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

7 From the **Displacement** list, choose **Absolute**.

8 In the **Cell displacement** text field, type `30*th`.

Surface 1

1 Right-click **Stress (Ply)** and choose **Surface**.

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

3 In the **Expression** text field, type `round(shell1.sl11)`.

Surface 2

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

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

3 In the **Expression** text field, type `round(shell2.sl11)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Cyclic**.

Surface 3

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

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

3 In the **Expression** text field, type `round(shell3.sl11)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Disco**.

Surface 4

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

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

3 In the **Expression** text field, type `round(shell4.sl11)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalDark**.

Stress (Ply)

1 In the **Model Builder** window, click **Stress (Ply)**.

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

3 From the **Position** list, choose **Right double**.

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

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

6 In the **Stress (Ply)** toolbar, click **Plot**.

To visualize the Hoffman safety factors in the layered shell, duplicate the **Stress** plot group.

Hoffman Safety Factors

1 Right-click **Stress (Ply)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Hoffman Safety Factors` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Hoffman Safety Factor (1)`.

Surface 1

1 In the **Model Builder** window, expand the **Hoffman Safety Factors** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `round(shell1.emm1.sf2.s_fm,3)`.

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 `round(shell2.emm1.sf2.s_fm,3)`.

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 `round(shell3.emm1.sf2.s_fm,3)`.

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 `round(shell4.emm1.sf2.s_fm,3)`.

Hoffman Safety Factors

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

2 In the **Model Builder** window, click **Hoffman Safety Factors**.

3 In the **Hoffman Safety Factors** toolbar, click **Plot**.

To visualize the combined safety factors in the layered shell, duplicate the **Hoffman Safety Factors** plot group.