Created in COMSOL Multiphysics 6.2

Failure Prediction in a Laminated Composite Shell

Introduction

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

This example shows how to model laminated composite shells using the Linear Elastic Material, Layered model in the Shell interface available with the Composite Materials Module.

The structural integrity of a laminate with different fiber orientations in each ply is assessed through the parameters called Failure Index and Safety Factor, using different polynomial failure criteria. Because of the varying fiber orientation, each ply will have different stiffness in the longitudinal and transverse directions, 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.

This 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 stacked square plies/layers. The side length is 1 cm and each layer has a thickness of 0.05 mm as shown in Figure 1. The laminate is subjected to an in-plane axial tensile load and has a [90/–45/45/0] stacking sequence as shown in Figure 2.

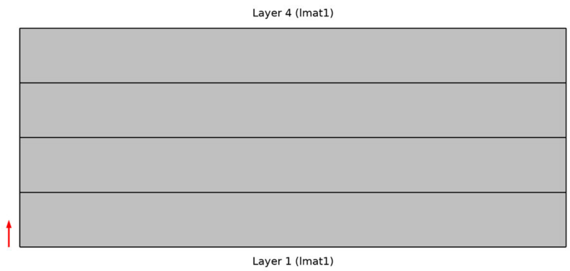

Figure 1: Cross-section view of the shell, showing the thickness (0.05 mm) of each ply.

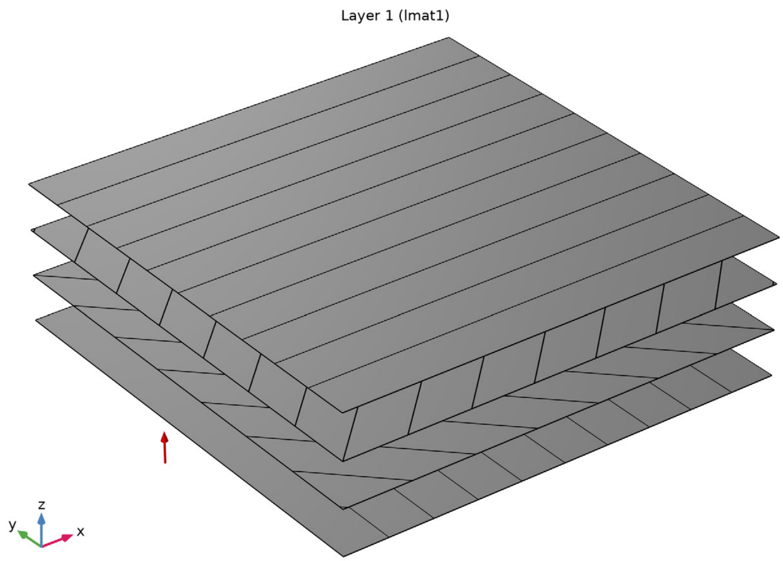

Figure 2: Stacking sequence [90/-45/45/0] of the laminate, from bottom to top, showing the fiber orientation in each ply.

MATERIAL PROPERTIES

All material properties and strengths are given in a layer coordinate system (local material directions of a ply), where the first axis is aligned with the fiber orientation. The composite plies are transversely isotropic; that is, directions 2 and 3 (and any other direction in the 23-plane) have equal stiffness properties. The ply material is therefore fully determined by seven material parameters for the Young's modulus, shear modulus, and Poisson's ratio (Table 1).

TABLE 1: MATERIAL PROPERTIES.

Material property	Value
$\{E_1, E_2\}$	{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.

Material strengths	Value
$\{\sigma_{ts1}, \sigma_{ts2}, \sigma_{ts3}\}$	{500, 5, 5} MPa
$\{\sigma_{cs1}, \sigma_{cs2}, \sigma_{cs3}\}$	{350, 75, 75} MPa
$\{\sigma_{ss23}, \sigma_{ss13}, \sigma_{ss12}\}$	{35,35,35} MPa

The local stresses are also evaluated in the layer coordinate system.

BOUNDARY CONDITIONS

The constraints and loads applied on each node of the laminate 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 the COMSOL Multiphysics geometry. The boundary conditions provided in the benchmark specifications are applied to the laminated composite shell as a single entity. The rotation around the z-axis, θ_z, is automatically constrained so it does not need to be considered.

It should be noted that since two point loads are prescribed at nodes, this benchmark can only be run with a single first-order element. This is the only case when such a specification can give a homogeneous stress state.

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 the 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), 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 $a = \sigma_i F_{ij} \sigma_j$ 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.

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

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

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

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 $\boldsymbol{F}_{ij}$ are given by Equation 5.

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

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

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

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.

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

$$
F_{66} = \frac{1}{\sigma_{ss12}^2} \tag{6}
$$

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

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, safety factor and stresses are computed at the midplane of each ply.

Results and Discussion

The computed stresses are shown in Table 4, while Table 5 to 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 LAMINATED COMPOSITE SHELL.

Ply	σ_{11} from benchmark	σ_{11}, computed	σ_{22} from benchmark	σ_{22}, computed	σ_{12} from benchmark	σ_{12}, computed
Ply 1	-5.128E6	-5.128E6	4.407E6	4.407E6	-1.663E6	-1.663E6
Ply 2	1.259E7	1.259E7	1.983E6	1.983E6	2.572E6	2.571E6
Ply 3	8.520E6	8.520E6	1.256E5	1.256E5	-2.051E6	-2.051E6
Ply 4	9.357E6	9.357E6	-1.859E6	-1.859E6	-5.557E5	-5.557E5

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 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 3 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. The von Mises stresses in all plies are shown in Figure 4. The stress in ply 1 is the lowest, but still this layer is still more susceptible to failure due to the orientation of its fibers.

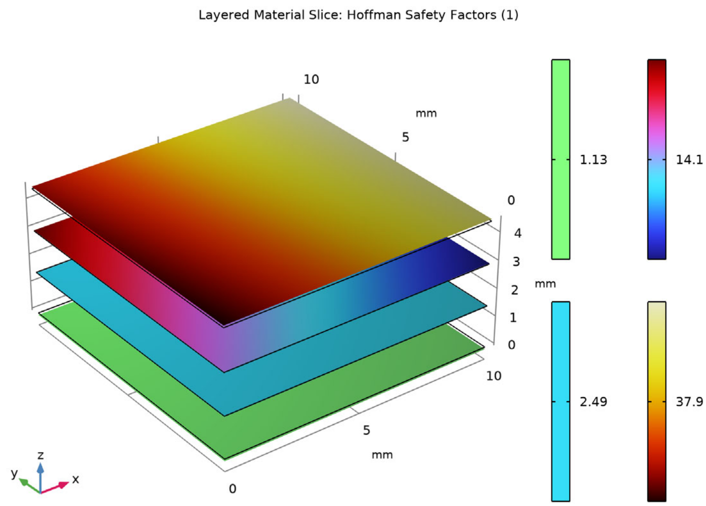

Figure 3: Hoffman safety factors at ply midplanes for the laminated composite shell.

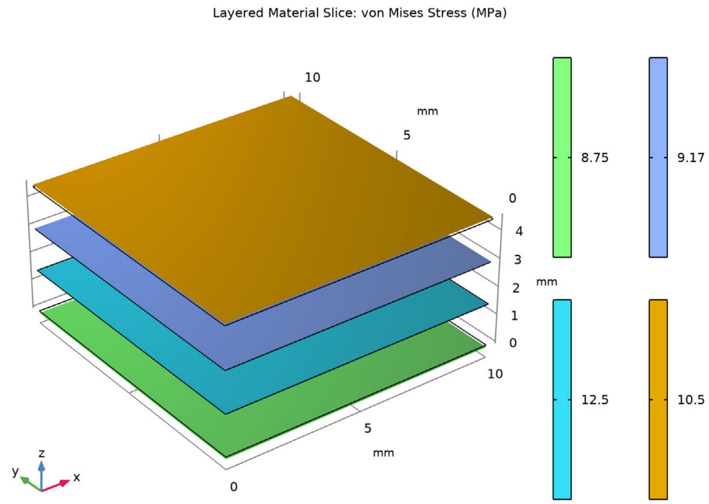

Figure 4: von Mises stress at ply midplanes for the laminated composite shell.

The distribution of the von Mises equivalent stress across the laminate is shown in the Figure 5. The interface between ply 2-ply 3 and ply 3-ply 4 experiences the maximum stresses due to bending of the laminate.

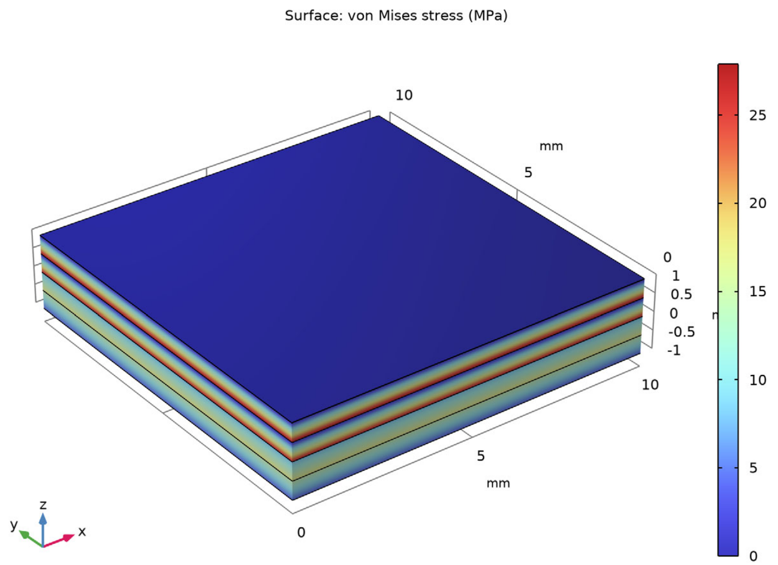

Figure 5: von Mises stress in the laminated composite shell.

The variation of the von Mises stress across the thickness at the middle of the laminate is shown in the Figure 6. It is evident that the von Mises stress is not continuous across the plies. The maximum Mises stress observed about 28–30 MPa in the third and fourth ply, which can be seen in Figure 5. The maximum stress observed at the bottom interface of the two top layers is caused by the bending of the laminate.

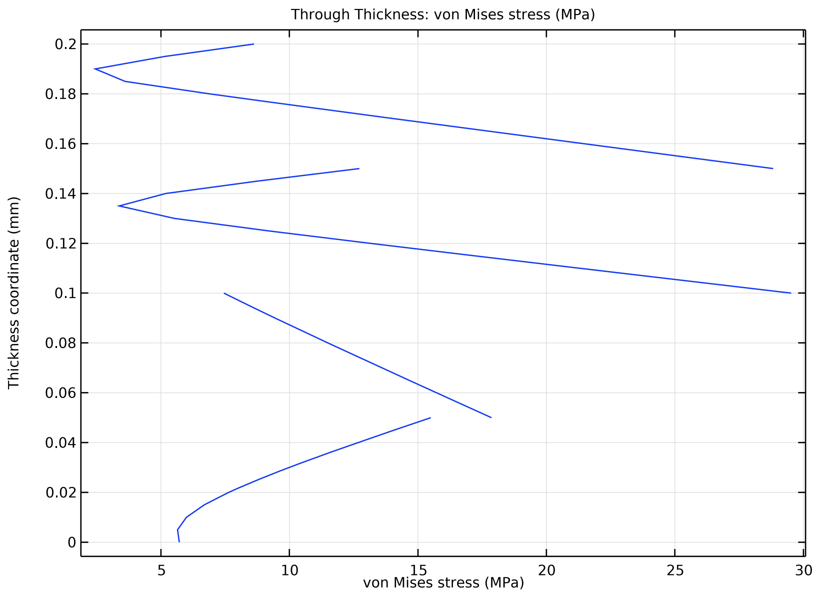

Figure 6: Through thickness variation of von Mises stress at the center of the laminated composite shell.

Notes About the COMSOL Implementation

- Modeling a composite laminated shell requires a surface geometry (2D), in general called a base surface, and a **Layered Material** node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. In the **Layered Material** node, you can model many layers stacked on top of each other having different thickness, material properties, and fiber orientations. You can also optionally specify the interface materials between the layers and control mesh elements in each layer.
- From a constitutive equations 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.
- To analyze the results in a composite shell, you can either create slice plot using a **Layered Material Slice** plot in order to see the in-plane variation of a quantity, or you can create a **Through Thickness** plot to see the out-of-plane variation of a quantity. In order to visualize the results as a 3D solid object, you can use the **Layered Material** dataset

which creates a virtual 3D solid object combining a surface geometry (2D) and the extra dimension (1D).

Reference

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

Application Library path: `Composite_Materials_Module/Verification_Examples/failure_prediction_in_a_laminated_composite_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 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Load the material properties and material strengths from a file.

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_laminated_composite_shell_materialproperties.txt`.

Material 1 (mat1)

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

Add a **Layered Material** node and assign appropriate thickness and rotation angles to each ply.

Layered Material: [90/-45/45/0]

1 Right-click **Materials** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, locate the **Layer Definition** section.

3 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 1	Material 1 (mat1)	90	th	1

4 Click **Add** three times.

5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 2	Material 1 (mat1)	-45	th	1
Layer 3	Material 1 (mat1)	45	th	1
Layer 4	Material 1 (mat1)	0	th	1

6 In the **Label** text field, type `Layered Material: [90/-45/45/0]`.

7 Locate the **Layer Definition** section. Click **Layer Cross-Section Preview** in the upper-right corner of the section.

8 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type `0.4`.

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

10 Locate the **Layer Definition** section. Click **Layer Cross-Section Preview** in the upper-right corner of the section.

11 Click **Layer Stack Preview** in the upper-right corner of the **Layer Definition** section.

GEOMETRY 1

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

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

3 From the **Length unit** list, choose **mm**.

Work Plane 1 (wp1)

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

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>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 10.

4 Click **Build Selected**.

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

MATERIALS

Layered Material Link 1 (llmat1)

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

SHELL (SHELL)

Activate **Advanced Physics Options**.

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

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

3 Click **OK**.

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

4 In the **Settings** window for **Shell**, click to expand the **Discretization** section.

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

Linear Elastic Material, Layered 1

Right-click **Component 1 (comp1)>Shell (shell)** and choose **Material Models>Linear Elastic Material, Layered**.

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

1 Select Boundary 1 only.

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

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

4 Select the **Transversely isotropic** check box.

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions>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

SHELL (SHELL)

Linear Elastic Material, Layered 1

In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Linear Elastic Material, Layered 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** check box.

5 Create five similar **Safety** nodes by duplicating the **Safety 1** node. Replace the failure criterion according to 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`

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions>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}	`{1/Sigmats1-1/Sigmacs1, 1/Sigmats2-1/Sigmacs2, 1/Sigmats3-1/Sigmacs3, 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	`{1/(Sigmats1*Sigmacs1), -0.5*sqrt(1/((Sigmats1*Sigmacs1)*(Sigmats2*Sigmacs2))), 1/(Sigmats2*Sigmacs2), -0.5*sqrt(1/((Sigmats1*Sigmacs1)*(Sigmats3*Sigmacs3))), -0.5*sqrt(1/((Sigmats2*Sigmacs2)*(Sigmats3*Sigmacs3))), 1/(Sigmats3*Sigmacs3), 0, 0, 0, 1/Sigmass23^2, 0, 0, 0, 0, 1/Sigmass13^2, 0, 0, 0, 0, 0, 1/Sigmass12^2}`	$m^2 \cdot s^4/kg^2$	Anisotropic strength parameters, Voigt notation

SHELL (SHELL)

Fixed Constraint 1

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

2 Select Point 1 only.

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**.

Apply a total tensile load of 15 N as an edge load.

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	x
0	y
0	z

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

Generation of default plots is switched off in order to create customized 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** check box.

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

RESULTS

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

Layered Material 1

1. In the **Model Builder** window, expand the **Results>Datasets** node.
2. Right-click **Results>Datasets** and choose **More Datasets>Layered Material**.
3. In the **Settings** window for **Layered Material**, locate the **Layers** section.
4. In the **Scale** text field, type `10`.

   Use an **Evaluation Group** instead of **Derived Values** to compute the failure indices, safety factors, and stresses.

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

5. In the **Model Builder** window, click **Results**.
6. In the **Settings** window for **Results**, locate the **Update of Results** section.
7. Select the **Reevaluate all evaluation groups after solving** check box.

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 **Layered Material 1**.
4. Locate the **Transformation** section. Select the **Transpose** check box.

Point Evaluation 1

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

   To compute the failure indices at the midplane of each ply, load the expressions from a file.

2. Select Point 4 only.
3. In the **Settings** window for **Point Evaluation**, locate the **Through-Thickness Location** section.
4. From the **Location definition** list, choose **Physical**.
5. In the **Local z-coordinate** text field, type `0.5*th`.
6. Locate the **Expressions** section. Click **Load from File**.
7. Browse to the model's Application Libraries folder and double-click the file `failure_prediction_in_a_laminated_composite_shell_failure_indices.txt`.

8 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 location appropriately in **Point Evaluation** nodes in the respective evaluation groups. Rename evaluation group nodes appropriately.

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 **Layered Material 1**.

4 Locate the **Transformation** section. Select the **Transpose** check box.

Point Evaluation 1

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

To compute the safety factors at the midplane of each ply, load the expressions from a file.

2 Select Point 4 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Through-Thickness Location** section.

4 From the **Location definition** list, choose **Physical**.

5 In the **Local z-coordinate** text field, type `0.5*th`.

6 Locate the **Expressions** section. Click **Load from File**.

7 Browse to the model's Application Libraries folder and double-click the file `failure_prediction_in_a_laminated_composite_shell_safety_factors.txt`.

8 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 location appropriately in **Point Evaluation** nodes in the respective evaluation groups. Rename evaluation group nodes appropriately.

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 **Layered Material 1**.

4 Locate the **Transformation** section. Select the **Transpose** check box.

Point Evaluation 1

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

To compute the stresses at the midplane of each ply, load the expressions from a file.

2 Select Point 4 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Through-Thickness Location** section.

4 From the **Location definition** list, choose **Physical**.

5 In the **Local z-coordinate** text field, type `0.5*th`.

6 Locate the **Expressions** section. Click **Load from File**.

7 Browse to the model's Application Libraries folder and double-click the file `failure_prediction_in_a_laminated_composite_shell_stresses.txt`.

8 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 location appropriately in **Point Evaluation** nodes in the respective evaluation groups. Rename evaluation group nodes appropriately.

To visualize von Mises stress 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.

von Mises 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 `von Mises 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 `Layered Material Slice: von Mises Stress (MPa)`.

5 Locate the **Color Legend** section. From the **Position** list, choose **Right double**.

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

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

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

9 In the **Cell displacement** text field, type `3E4*th`.

Ply 1

1 Right-click **von Mises Stress (Ply)** and choose **Layered Material Slice**.

2 In the **Settings** window for **Layered Material Slice**, type `Ply 1` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Shell>Stress>shell.misesGp - von Mises stress - N/m²**.

4 Locate the **Expression** section. In the **Expression** text field, type `round(shell.mises)`.

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

6 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Relative**.

7 In the **Local z-coordinate [-1,1]** text field, type `-0.75`.

8 Right-click **Ply 1** and choose **Duplicate**.

Ply 2

1 In the **Model Builder** window, under **Results>von Mises Stress (Ply)** click **Ply 1.1**.

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

3 Locate the **Through-Thickness Location** section. In the **Local z-coordinate [-1,1]** text field, type `-0.25`.

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

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

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

7 Click **OK**.

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

Ply 3

1 In the **Model Builder** window, under **Results>von Mises Stress (Ply)** click **Ply 2.1**.

2 In the **Settings** window for **Layered Material Slice**, type `Ply 3` in the **Label** text field.

3 Locate the **Through-Thickness Location** section. In the **Local z-coordinate [-1,1]** text field, type `0.25`.

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

5 In the **Color Table** dialog box, select **Wave>Disco** in the tree.

6 Click **OK**.

7 Right-click **Ply 3** and choose **Duplicate**.

Ply 4

1 In the **Model Builder** window, under **Results>von Mises Stress (Ply)** click **Ply 3.1**.

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

3 Locate the **Through-Thickness Location** section. In the **Local z-coordinate [-1,1]** text field, type `0.75`.

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

5 In the **Color Table** dialog box, select **Thermal>ThermalDark** in the tree.

6 Click **OK**.

von Mises Stress (Ply)

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

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

2 In the **Model Builder** window, right-click **von Mises Stress (Ply)** and choose **Duplicate**.

Hoffman Safety Factors (Ply)

1 In the **Model Builder** window, under **Results** click **von Mises Stress (Ply) 1**.

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

3 Locate the **Title** section. In the **Title** text area, type `Layered Material Slice: Hoffman Safety Factors (1)`.

Ply 1

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

2 In the **Settings** window for **Layered Material Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Shell>Safety>Hoffman>shell.llem1.lsf2.s_f - Hoffman safety factor - 1**.

Ply 2

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

2 In the **Settings** window for **Layered Material Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Shell>Safety>Hoffman>shell.llem1.lsf2.s_f - Hoffman safety factor - 1**.

Ply 3

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

2 In the **Settings** window for **Layered Material Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Shell>Safety>Hoffman>shell.llem1.lsf2.s_f - Hoffman safety factor - 1**.

Ply 4

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

2 In the **Settings** window for **Layered Material Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Shell>Safety>Hoffman>shell.llem1.lsf2.s_f - Hoffman safety factor - 1**.

Hoffman Safety Factors (Ply)

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

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

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

von Mises Stress (Laminate)

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

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

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

Surface 1

1 Right-click **von Mises Stress (Laminate)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Shell>Stress>shell.misesGp - von Mises stress - N/m²**.

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

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

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

6 Click **OK**.

7 In the **von Mises Stress (Laminate)** toolbar, click **Plot**.

Add a **Cut Point 3D** at the center of the geometry in order to visualize the through-thickness stress variation.

Cut Point 3D 1

1 In the **Results** toolbar, click **Cut Point 3D**.

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

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

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

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

von Mises Stress at Midpoint (Through-Thickness)

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `von Mises Stress at Midpoint (Through-Thickness)` in the **Label** text field.

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

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

Through Thickness 1

1 In the **von Mises Stress at Midpoint (Through-Thickness)** toolbar, click **More Plots** and choose **Through Thickness**.

2 In the **Settings** window for **Through Thickness**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `shell.mises`.

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

5 Locate the **y-Axis Data** section. From the **Unit** list, choose **mm**.

6 In the **von Mises Stress at Midpoint (Through-Thickness)** toolbar, click **Plot**.