Created in COMSOL Multiphysics 6.2

Stacking Sequence Optimization

Introduction

Composite laminates are synthetic structures and there is always a possibility to optimize the design in terms of the number of layers, the material of each layer, the thickness of each layer, and the stacking sequence for the specified loading conditions. Designers need to estimate how safe the composite material is for a chosen application and under given loading conditions. With suitable failure criteria, the performance of the composite can be assessed and optimized in order to reduce the failure index or increase safety factor for specified loading conditions.

This example illustrates how to optimize the stacking sequence in a composite laminate based on the Hashin failure criterion. The composite laminate considered for the analysis has six layers with a symmetric layup. A carbon–epoxy material with transversely isotropic material properties is used for the lamina. An optimization analysis is performed to find the optimum fiber orientation in each layer under specified loading conditions with the objective of minimizing the maximum failure index in the laminate. The derivative-free BOBYQA optimization solver is applied to find the optimum stacking sequence.

Model Definition

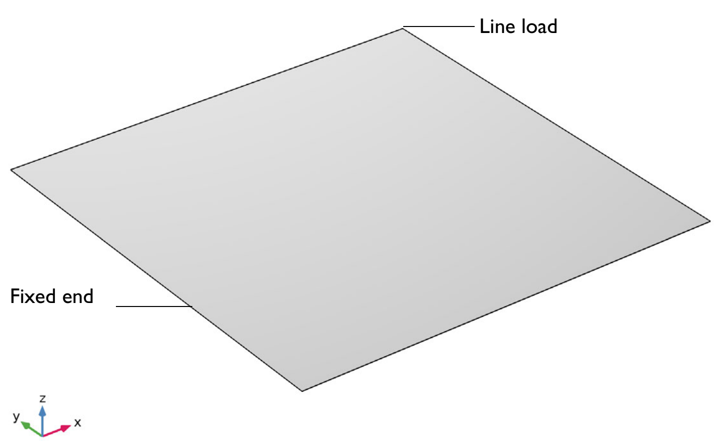

Figure 1: Model geometry of a composite laminate.

GEOMETRY AND BOUNDARY CONDITIONS

The geometry of a composite laminate with a side length of 0.5 m is shown in Figure 1. The following boundary conditions are applied:

- The left side of the composite laminate is fixed;
- A total load of 2 kN is applied to the top-right corner f the laminate in the form of a line load as shown in Figure 2.

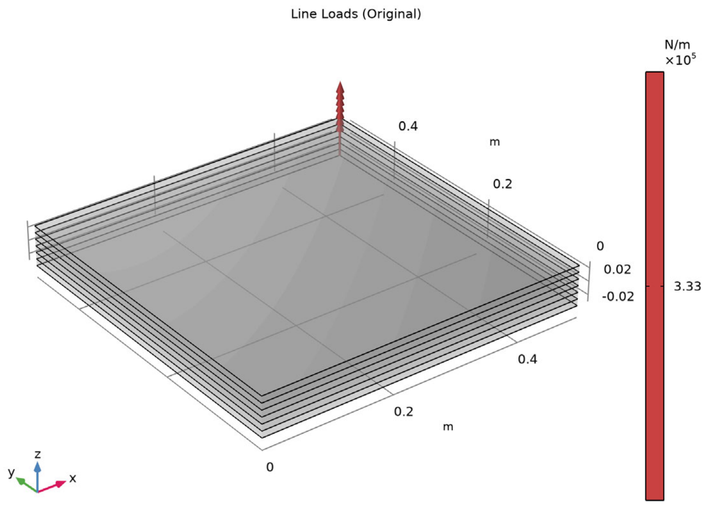

Figure 2: A 3D representation of the composite geometry together with the applied line load. Note that the geometry is scaled by a factor of 10 in the thickness direction for visualization purposes.

STACKING SEQUENCE

The laminate considered for the analysis consists of 6 layers with a symmetric layup. The original ply angles are assumed to be zero and are optimized to minimize the maximum failure index in the laminate under the loading conditions described above. The through-thickness view and the original layup of the laminate can be seen in Figure 3 and Figure 4, respectively.

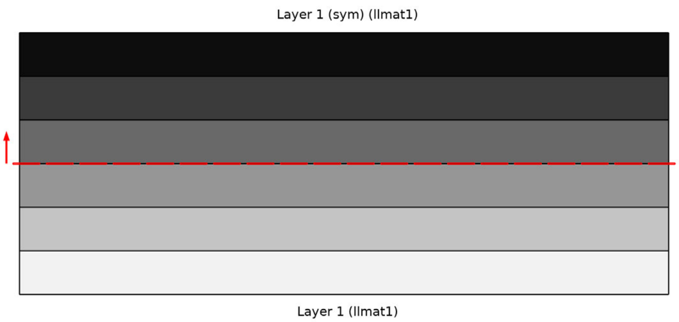

Figure 3: Through-thickness view of the laminate.

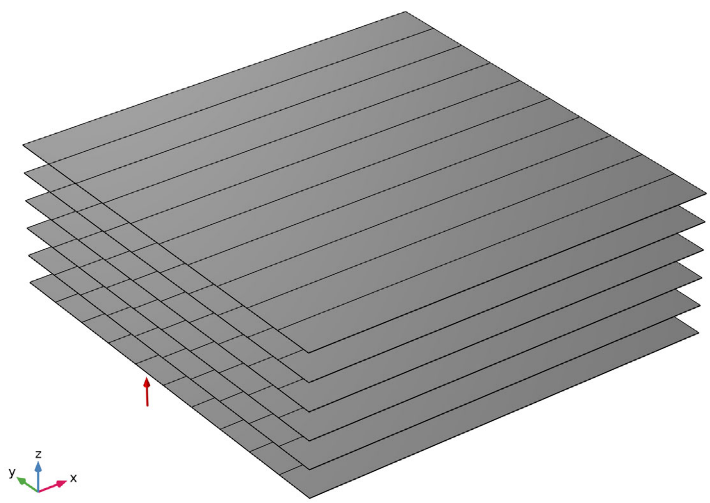

Figure 4: Stacking sequence $[0]_6$ of the original layup of the laminate.

MATERIAL PROPERTIES

Each ply of composite panel is assumed to be made of carbon fibers in an epoxy resin. The homogenized transversely isotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) and material strengths are given below:

TABLE 1: MATERIAL PROPERTIES OF A PLY.

Material property	**Value**
$\{E_1, E_2\}$	{134, 9.2} GPa
G_{12}	4.8 GPa
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.28, 0.28}
$\{\sigma_{T1}, \sigma_{T2}, \sigma_{T3}\}$	{2000, 80, 80} MPa
$\{\sigma_{C1}, \sigma_{C2}, \sigma_{C3}\}$	{1600, 200, 200} MPa
$\{\sigma_{S12}, \sigma_{S23}, \sigma_{S13}\}$	{150, 150, 150} MPa

LAYUP OPTIMIZATION

In the original layup, all plies are assumed to be aligned with the laminate coordinate system axis; in other words, their ply angles are zero. The objective is to optimize the ply angles in order to minimize the maximum failure index in the entire laminate.

The laminate considered here consists of 6 plies with a symmetric layup so effectively three ply angles are the control variables for the optimization problem. The initial value of the control variables is 0 degrees and the lower and upper bounds are –90 degrees and 90 degrees, respectively. A parametric optimization is performed using the BOBYQA method in order to find the optimum stacking sequence.

Results and Discussion

The Hashin failure criterion is a well-known failure criterion for composites that considers six failure modes: fiber failure in tension, fiber failure in compression, matrix failure in tension, matrix failure in compression, interlaminar failure in tension, and interlaminar failure in compression. The overall failure criterion of the composite is evaluated as the most critical of the underlying failure modes.

In the original ply design, the failure indices for several failure modes are greater than one, indicating composite failure by different failure modes at the given load. Therefore, there is no simple choice of stress component that can be used to assess the severity of the load. For this reason, the von Mises stress measure is used to the visualize the loading intensity in Figure 5; note, however, that the stacking optimization is based on the failure index.

The distribution of the maximum failure index per interface in the laminate can be seen in Figure 6. The failure index is maximal and above one in the first ply, which indicates that the structure in unsafe for the given loading conditions.

When the laminate stacking sequence is optimized with the objective of reducing the maximum failure index, both the von Mises stress (Figure 7) and the failure index (Figure 8) are reduced considerably. The maximum stress values are reduced by 50% in the optimized layup, while the maximum failure index is reduced by 70%. If we look at the stress distribution, it is now more evenly distributed within the plies as well as across the plies. The reduction of the failure index well below 1 indicates that the structure is safe for the given loading conditions.

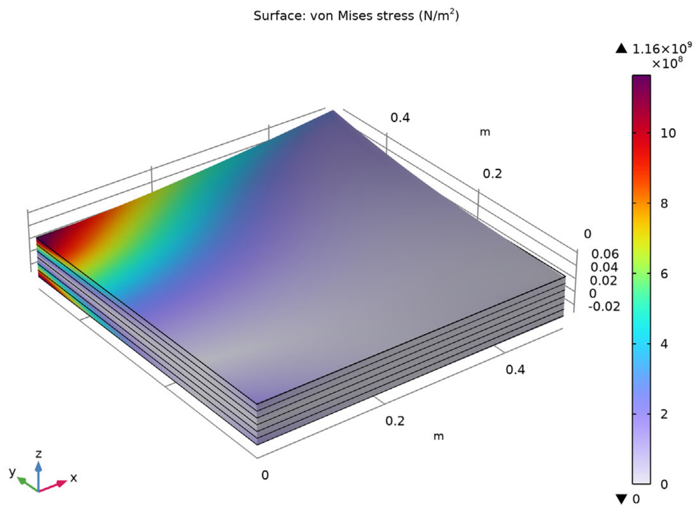

Figure 5: von Mises stress distribution in the laminate with the original layup.

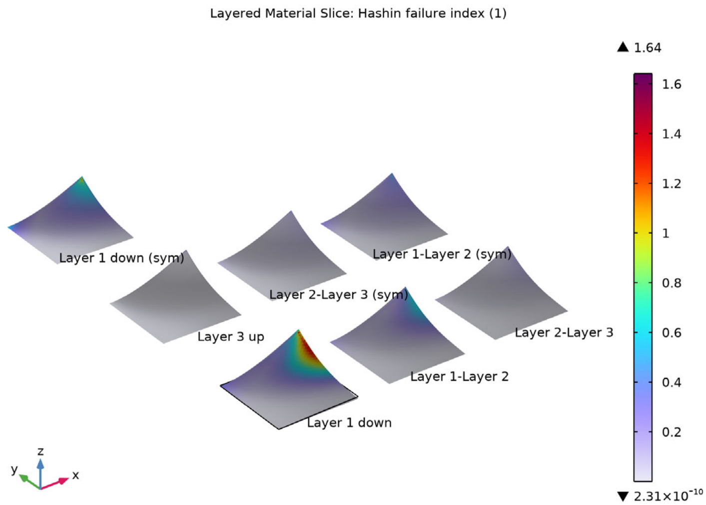

Figure 6: Distribution of the Hashin failure index in each interface of the laminate for the original layup.

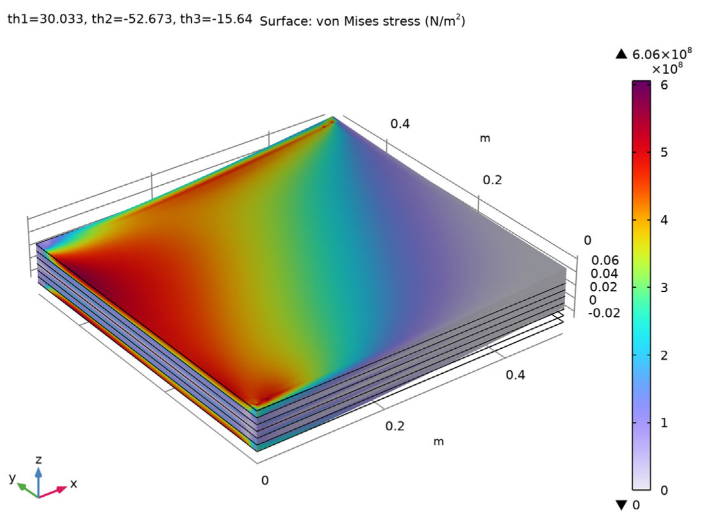

Figure 7: von Mises stress distribution in the laminate for the optimized layup.

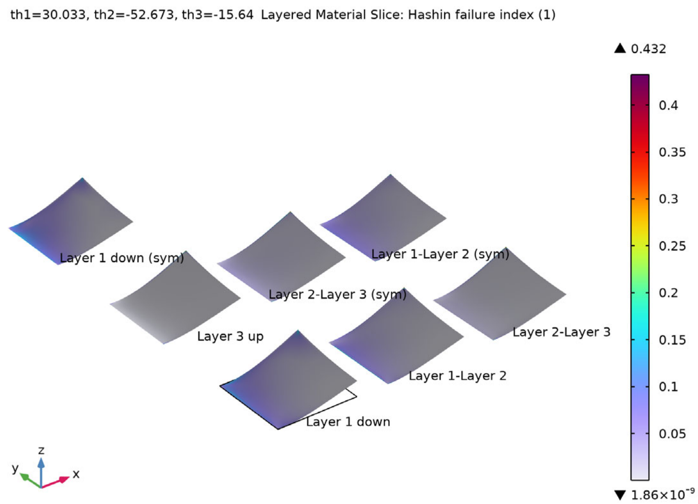

Figure 8: Distribution of the Hashin failure index in each interface of the laminate for the optimized layup.

The ply angles in the original and optimized layups are provided below and visualized in Figure 9 and Figure 10.

- Original layup: $[0/0/0]_s$
- Optimized layup: $[30/-52/-15]_s$ (after round-off)

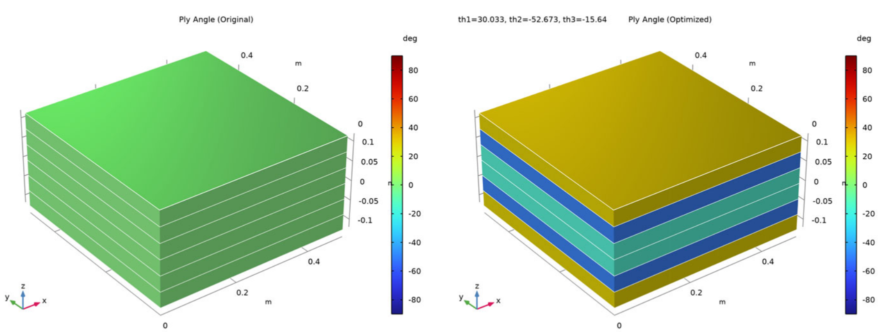

Figure 9: Original and optimized ply angles.

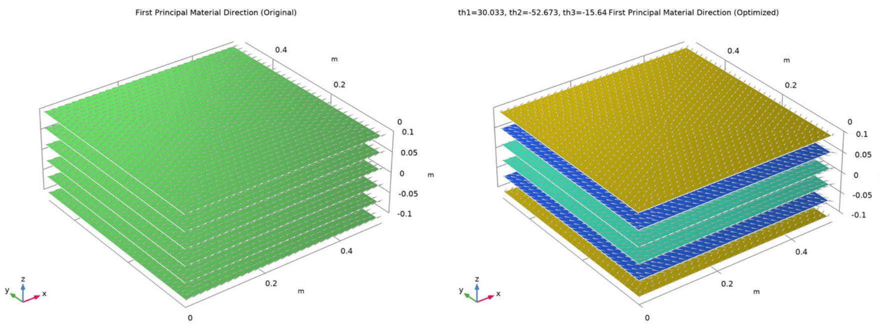

Figure 10: Original and optimized first principal material directions.

Figure 11 compares the displacement magnitude on the deformed configuration of the laminate between the original and optimized layups. Here, the original layup result is plotted as a wireframe whereas the optimized layup result is plotted as a solid surface. The optimized layup is stiffer at the loading point and predominantly goes into a bending mode compared to the original layup which goes into a mixed bending–twisting mode.

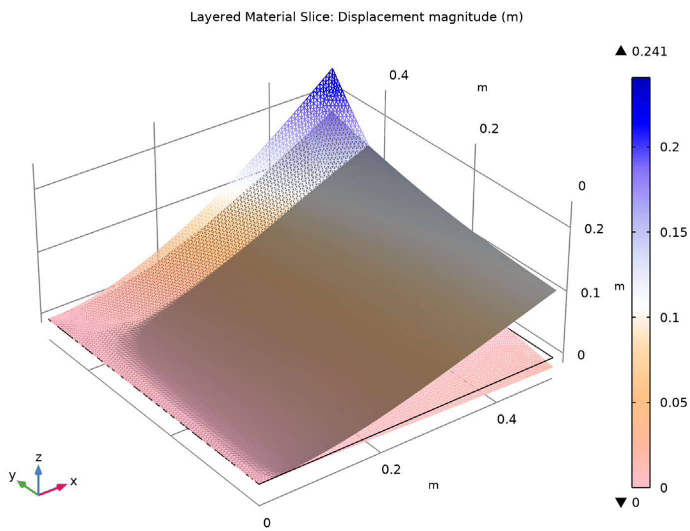

Figure 11: Displacement magnitude plotted on the deformed configuration for the original (wireframe) and optimized (solid) layups.

Application Library path: `Composite_Materials_Module/Tutorials/stacking_sequence_optimization`

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 (lshell)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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 `stacking_sequence_optimization_parameters.txt`.

LAYERED SHELL (LSHELL)

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** check box.

Safety 1

1 In the **Physics** toolbar, click **Attributes** and choose **Safety**.

2 In the **Settings** window for **Safety**, locate the **Failure Model** section.

3 From the **Failure criterion** list, choose **Hashin**.

GLOBAL DEFINITIONS

Material: Carbon–Epoxy

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

2 In the **Settings** window for **Material**, type `Material: Carbon-Epoxy` in the **Label** text field.

Define a layered material with ply rotations or stacking sequence as parameters to be optimized.

Layered Material: [th1/th2/th3]

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: Carbon-Epoxy (mat1)	`th1`	`d_layer`	1

4 Click **Add**.

5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 2`	Material: Carbon-Epoxy (mat1)	`th2`	`d_layer`	1

6 Click **Add**.

7 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 3`	Material: Carbon-Epoxy (mat1)	`th3`	`d_layer`	1

8 In the **Label** text field, type `Layered Material: [th1/th2/th3]`.

GEOMETRY 1

Work Plane 1 (wp1)

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

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>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 `a`.
4 Click the **Go to Default View** button in the **Graphics** toolbar.
5 In the **Home** toolbar, click **Build All**.
6 Click the **Show Grid** button in the **Graphics** toolbar.
7 Click the **Show Grid** button in the **Graphics** toolbar.
8 In the **Model Builder** window, collapse the **Geometry 1** node.

MATERIALS

Layered Material Link 1 (llmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Layers>Layered Material Link**.
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 `0.4`.
5 Click **Section_bar** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Layer Cross-Section Preview**.
6 Click **Section_bar** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Layer Stack Preview**.

GLOBAL DEFINITIONS

Material: Carbon–Epoxy (mat1)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material: Carbon–Epoxy (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, nu12}	1	Transversely isotropic
Shear modulus	Gvect1	G12	N/m²	Transversely isotropic
Density	rho	1	kg/m³	Basic
Tensile strengths	{sigmats1, sigmats2, sigmats3}	{sigmaT1, sigmaT2, sigmaT2}	Pa	Orthotropic strength parameters, Voigt notation
Compressive strengths	{sigmacs1, sigmacs2, sigmacs3}	{sigmaC1, sigmaC2, sigmaC2}	Pa	Orthotropic strength parameters, Voigt notation
Shear strengths	{sigmass1, sigmass2, sigmass3}	{sigmaS, sigmaS, sigmaS}	Pa	Orthotropic strength parameters, Voigt notation

LAYERED SHELL (LSHELL)

Fixed Constraint 1

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

2 Select Edge 1 only.

Line Load 1

1 In the **Physics** toolbar, click **Points** and choose **Line Load**.

2 Select Point 4 only.

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

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

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

0	x
0	y
F	z

Define a global variable, corresponding to the maximum failure index in the laminate, in order to use as the objective function in the optimization analysis.

DEFINITIONS (COMP1)

Maximum 1 (maxop1)

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

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
FI_max_l1	maxop1(lshell.atxd1(0, lshell.lemm1.sf1.f_i))		Maximum failure index, layer 1
FI_max_l2	maxop1(lshell.atxd1(d_layer, lshell.lemm1.sf1.f_i))		Maximum failure index, layer 2
FI_max_l3	maxop1(lshell.atxd1(2*d_layer, lshell.lemm1.sf1.f_i))		Maximum failure index, layer 3
FI_max_l4	maxop1(lshell.atxd1(3*d_layer, lshell.lemm1.sf1.f_i))		Maximum failure index, layer 4
FI_max_l5	maxop1(lshell.atxd1(4*d_layer, lshell.lemm1.sf1.f_i))		Maximum failure index, layer 5
FI_max_l6	maxop1(lshell.atxd1(5*d_layer, lshell.lemm1.sf1.f_i))		Maximum failure index, layer 6
FI_max	max(max(max(max(max(FI_max_l1, FI_max_l2),FI_max_l3), FI_max_l4),FI_max_l5), FI_max_l6)		Maximum failure index

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
3. From the **Element size** list, choose **Extra fine**.
4. Click **Build All**.

STUDY 1: ORIGINAL LAYUP

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type `Study 1: Original Layup` in the **Label** text field.
3. In the **Home** toolbar, click **Compute**.

RESULTS

Maximum Failure Index (Original)

1. In the **Results** toolbar, click **Evaluation Group**.
2. In the **Settings** window for **Evaluation Group**, type `Maximum Failure Index (Original)` in the **Label** text field.

Global Evaluation 1

1. Right-click **Maximum Failure Index (Original)** and choose **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
`FI_max`	1	`Maximum failure index`

4. In the **Maximum Failure Index (Original)** toolbar, click **Evaluate**.

Failure Index of Different Failure Modes (Original)

1. In the **Results** toolbar, click **Evaluation Group**.
2. In the **Settings** window for **Evaluation Group**, type `Failure Index of Different Failure Modes (Original)` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Layered Material**.
4. Locate the **Transformation** section. Select the **Transpose** check box.

Volume Maximum 1

1. Right-click **Failure Index of Different Failure Modes (Original)** and choose **Maximum>Volume Maximum**.
2. In the **Settings** window for **Volume Maximum**, click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Layered Shell>Safety>Hashin>lshell.lemm1.sf1.f_ifT - Hashin fiber tensile failure index - 1**.
3. Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Layered Shell>Safety>Hashin>lshell.lemm1.sf1.f_ifC - Hashin fiber compressive failure index - 1**.
4. Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Layered Shell>Safety>Hashin>lshell.lemm1.sf1.f_imT - Hashin matrix tensile failure index - 1**.
5. Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Layered Shell>Safety>Hashin>lshell.lemm1.sf1.f_imC - Hashin matrix compressive failure index - 1**.
6. Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Layered Shell>Safety>Hashin>lshell.lemm1.sf1.f_iiT - Hashin interlaminar tensile failure index - 1**.
7. Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Layered Shell>Safety>Hashin>lshell.lemm1.sf1.f_iiC - Hashin interlaminar compressive failure index - 1**.
8. In the **Failure Index of Different Failure Modes (Original)** toolbar, click **Evaluate**.

Increase the through thickness scale factor in various layered material datasets for better visualization.

Layered Material

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

von Mises Stress (Original)

1. In the **Model Builder** window, under **Results** click **Stress (lshell)**.
2. In the **Settings** window for **3D Plot Group**, type `von Mises Stress (Original)` in the **Label** text field.

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

4 In the **von Mises Stress (Original)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1: Original Layup/Solution 1 (sol1)>Layered Shell>Stress, Slice (lshell)**.

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

RESULTS

Failure Index, Slice (Original)

1 In the **Settings** window for **3D Plot Group**, type `Failure Index, Slice (Original)` in the **Label** text field.

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

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Failure Index, Slice (Original)** node, then click **Layered Material Slice 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)> Layered Shell>Safety>Hashin>lshell.lemm1.sf1.f_i - Hashin failure index - 1**.

3 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Interfaces**.

4 Locate the **Layout** section. From the **Displacement** list, choose **Rectangular**.

5 In the **Relative x-separation** text field, type `0.4`.

6 In the **Relative y-separation** text field, type `0.4`.

7 Select the **Show descriptions** check box.

8 In the **Relative separation** text field, type `0.5`.

9 In the **Failure Index, Slice (Original)** toolbar, click **Plot**.

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

Failure Index, Slice (Original)

1 In the **Model Builder** window, click **Failure Index, Slice (Original)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.
3 From the **View** list, choose **New view**.
4 In the **Failure Index, Slice (Original)** toolbar, click **Plot**.
5 Click the **Show Grid** button in the **Graphics** toolbar.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.
2 In the tree, select **Study 1: Original Layup/Solution 1 (sol1)>Layered Shell>Geometry and Layup (lshell)>Ply Angle (lshell)**.
3 Click **Add Plot** in the window toolbar.
4 In the tree, select **Study 1: Original Layup/Solution 1 (sol1)>Layered Shell>Geometry and Layup (lshell)>First Principal Material Direction (lshell)**.
5 Click **Add Plot** in the window toolbar.

RESULTS

Ply Angle (lshell)

1 In the **Model Builder** window, under **Results** click **Ply Angle (lshell)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 Click **Go to Source**.

Layered Material 2 (Shell Geometry)

1 In the **Model Builder** window, under **Results>Datasets** click **Layered Material 2 (Shell Geometry)**.
2 In the **Settings** window for **Layered Material**, locate the **Layers** section.
3 In the **Scale** text field, type `40`.

Ply Angle (Original)

1 In the **Model Builder** window, under **Results** click **Ply Angle (lshell)**.
2 In the **Settings** window for **3D Plot Group**, type `Ply Angle (Original)` in the **Label** text field.
3 Click the **Go to Default View** button in the **Graphics** toolbar.
4 In the **Ply Angle (Original)** toolbar, click **Plot**.

First Principal Material Direction (lshell)

1 In the **Model Builder** window, click **First Principal Material Direction (lshell)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 Click **Go to Source**.

Layered Material 2 (Material Direction)

1 In the **Model Builder** window, under **Results>Datasets** click **Layered Material 2 (Material Direction)**.
2 In the **Settings** window for **Layered Material**, locate the **Layers** section.
3 In the **Scale** text field, type `40`.

First Principal Material Direction (Original)

1 In the **Model Builder** window, under **Results** click **First Principal Material Direction (lshell)**.
2 In the **Settings** window for **3D Plot Group**, type `First Principal Material Direction (Original)` in the **Label** text field.
3 In the **First Principal Material Direction (Original)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.
2 In the tree, select **Study 1: Original Layup/Solution 1 (sol1)>Layered Shell> Applied Loads (lshell)>Line Loads (lshell)**.
3 Click **Add Plot** in the window toolbar.

RESULTS

Line Loads (Original)

1 In the **Settings** window for **3D Plot Group**, type `Line Loads (Original)` in the **Label** text field.
2 In the **Line Loads (Original)** toolbar, click **Plot**.

After solving the model for the original layup, add an optimization analysis for optimizing the layup for given loading conditions.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.
4 Click **Add Study** in the window toolbar.

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

STUDY 2: LAYUP OPTIMIZATION

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

2 In the **Settings** window for **Study**, type `Study 2: Layup Optimization` in the **Label** text field.

Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Optimization**.

2 In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.

3 From the **Method** list, choose **BOBYQA**.

4 Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description	Evaluate for
`comp1.FI_max`	`Maximum failure index`	Stationary

5 Locate the **Control Variables and Parameters** section. Click **Add** three times.

6 In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
th1 (Fiber orientation, layer 1)	0	45	-90	90
th2 (Fiber orientation, layer 2)	0	45	-90	90
th3 (Fiber orientation, layer 3)	0	45	-90	90

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

RESULTS

Layered Material 2

1 In the **Model Builder** window, under **Results>Datasets** click **Layered Material 2**.

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

3 In the **Scale** text field, type `10`.

von Mises Stress (Optimized)

1 In the **Model Builder** window, under **Results** click **Stress (lshell)**.

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

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

4 In the **von Mises Stress (Optimized)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 2: Layup Optimization/Parametric Solutions 1 (sol3)> Layered Shell>Stress, Slice (lshell)**.

3 Click **Add Plot** in the window toolbar.

RESULTS

Failure Index, Slice (Optimized)

1 In the **Settings** window for **3D Plot Group**, type `Failure Index, Slice (Optimized)` in the **Label** text field.

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

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Failure Index, Slice (Optimized)** node, then click **Layered Material Slice 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)> Layered Shell>Safety>Hashin>lshell.lemm1.sf1.f_i - Hashin failure index - 1**.

3 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Interfaces**.

4 Locate the **Layout** section. From the **Displacement** list, choose **Rectangular**.

5 In the **Relative x-separation** text field, type `0.4`.

6 In the **Relative y-separation** text field, type `0.4`.

7 Select the **Show descriptions** check box.

8 In the **Relative separation** text field, type `0.5`.

Failure Index, Slice (Optimized)

1 In the **Model Builder** window, click **Failure Index, Slice (Optimized)**.

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

3 From the **View** list, choose **View 3D 4**.

4 In the **Failure Index, Slice (Optimized)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 2: Layup Optimization/Parametric Solutions 1 (sol3)>Layered Shell>Geometry and Layup (lshell)>Ply Angle (lshell)**.

3 Click **Add Plot** in the window toolbar.

4 In the tree, select **Study 2: Layup Optimization/Parametric Solutions 1 (sol3)>Layered Shell>Geometry and Layup (lshell)>First Principal Material Direction (lshell)**.

5 Click **Add Plot** in the window toolbar.

6 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Ply Angle (lshell)

1 In the **Model Builder** window, under **Results** click **Ply Angle (lshell)**.

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

3 Click **Go to Source**.

Layered Material 3 (Shell Geometry)

1 In the **Model Builder** window, under **Results>Datasets** click **Layered Material 3 (Shell Geometry)**.

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

3 In the **Scale** text field, type `40`.

Ply Angle (Optimized)

1 In the **Model Builder** window, under **Results** click **Ply Angle (lshell)**.

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

3 Click the **Go to Default View** button in the **Graphics** toolbar.

4 In the **Ply Angle (Optimized)** toolbar, click **Plot**.

First Principal Material Direction (lshell)

1 In the **Model Builder** window, click **First Principal Material Direction (lshell)**.

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

3 Click **Go to Source**.

Layered Material 3 (Material Direction)

1 In the **Model Builder** window, under **Results>Datasets** click **Layered Material 3 (Material Direction)**.
2 In the **Settings** window for **Layered Material**, locate the **Layers** section.
3 In the **Scale** text field, type `40`.

First Principal Material Direction (Optimized)

1 In the **Model Builder** window, under **Results** click **First Principal Material Direction (lshell)**.
2 In the **Settings** window for **3D Plot Group**, type `First Principal Material Direction (Optimized)` in the **Label** text field.
3 In the **First Principal Material Direction (Optimized)** toolbar, click **Plot**.

Create a plot to compare the deformation and displacement profile of the laminate for original and optimized layup.

Failure Index, Slice (Original)

In the **Model Builder** window, right-click **Failure Index, Slice (Original)** and choose **Duplicate**.

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Failure Index, Slice (Original) 1** node, then click **Layered Material Slice 1**.
2 In the **Settings** window for **Layered Material Slice**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 1: Original Layup/Solution 1 (sol1)**.
4 Locate the **Expression** section. In the **Expression** text field, type `lshell.disp`.
5 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Reference surface**.
6 Locate the **Layout** section. From the **Displacement** list, choose **None**.
7 Clear the **Show descriptions** check box.
8 Locate the **Coloring and Style** section. Click **Change Color Table**.
9 In the **Color Table** dialog box, select **Aurora>Twilight** in the tree.
10 Click **OK**.
11 In the **Settings** window for **Layered Material Slice**, locate the **Coloring and Style** section.
12 Select the **Wireframe** check box.
13 Right-click **Layered Material Slice 1** and choose **Duplicate**.

Layered Material Slice 2

1 In the **Model Builder** window, click **Layered Material Slice 2**.

2 In the **Settings** window for **Layered Material Slice**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2: Layup Optimization/Parametric Solutions 1 (sol3)**.

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

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

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

Deformation

1 In the **Model Builder** window, expand the **Results>Failure Index, Slice (Original) 1>Layered Material Slice 1** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box. In the associated text field, type `1`.

Displacement: Original and Optimized

1 In the **Model Builder** window, under **Results** click **Failure Index, Slice (Original) 1**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement: Original and Optimized` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **Automatic**.

4 In the **Displacement: Original and Optimized** toolbar, click **Plot**.

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

Maximum Failure Index (Original)

In the **Model Builder** window, right-click **Maximum Failure Index (Original)** and choose **Duplicate**.

Maximum Failure Index (Optimized)

1 In the **Model Builder** window, under **Results** click **Maximum Failure Index (Original) 1**.

2 In the **Settings** window for **Evaluation Group**, type `Maximum Failure Index (Optimized)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Layup Optimization/Solution 2 (sol2)**.

4 In the **Maximum Failure Index (Optimized)** toolbar, click **Evaluate**.

Failure Index of Different Failure Modes (Original)

In the **Model Builder** window, right-click **Failure Index of Different Failure Modes (Original)** and choose **Duplicate**.

Failure Index of Different Failure Modes (Optimized)

1. In the **Model Builder** window, under **Results** click **Failure Index of Different Failure Modes (Original) 1**.
2. In the **Settings** window for **Evaluation Group**, type `Failure Index of Different Failure Modes (Optimized)` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Layered Material 2**.
4. From the **Parameter selection (th1, th2, th3)** list, choose **Last**.
5. In the **Failure Index of Different Failure Modes (Optimized)** toolbar, click **Evaluate**.