Model created in COMSOL Multiphysics 6.4

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.

	Read more about the Composite Materials Module in the COMSOL blog, Introduction to the Composite Materials Module.

Model Definition

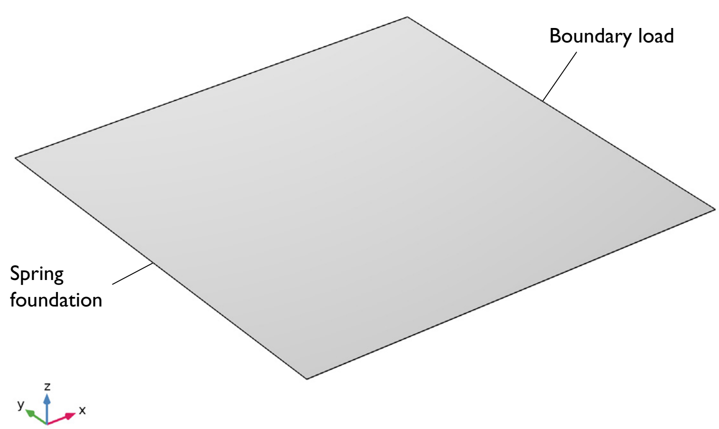

Figure 1: Model geometry of a composite laminate.

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:

- A fixed, fully constrained, boundary condition contains the assumption that the analyzed structure is attached to an infinitely stiff support. In many cases, this is a useful approximation, but cases like optimization you may need to consider the flexibility of the supporting structure in your model. Hence, the spring foundation is used to constraint the left side of the composite laminate.
- A total load of 12 kN, varying linearly with position, is applied to the right side of the laminate as a boundary load as shown in Figure 2.

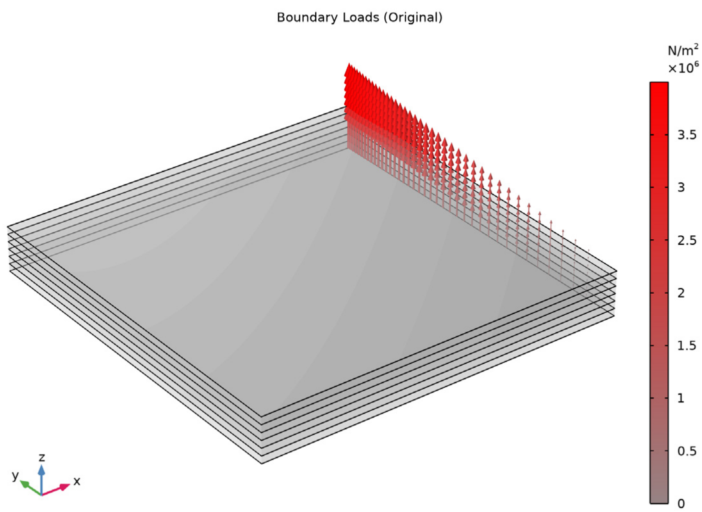

Figure 2: A 3D representation of the composite geometry together with the applied boundary load. Note that the geometry is scaled by a factor of 10 in the thickness direction for visualization purposes.

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.

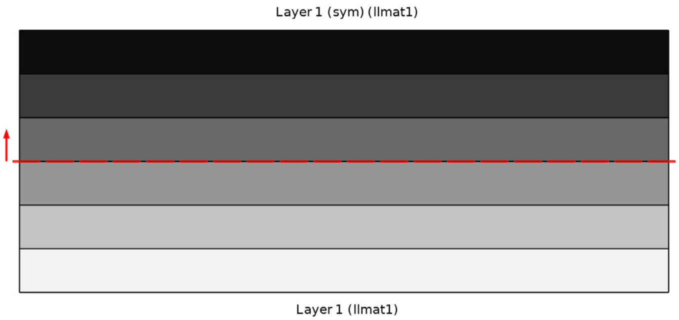

Figure 3: Through-thickness view of the laminate.

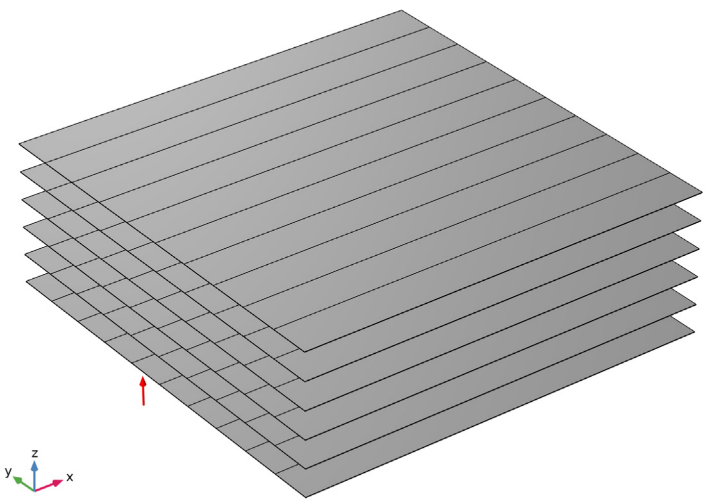

Figure 4: Stacking sequence $[0]_6$ *of the original layup of the laminate.*

MATERIAL PROPERTIES

Each ply is made up of AS4/APC carbon–thermoplastic composite material. The AS4/APC carbon–thermoplastic is a built-in material in the **Composites** material library. The transversely isotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given in Table 1. The material strengths are given in Table 2.

TABLE 1: MATERIAL PROPERTIES OF A PLY.

Material property	Value
$\{E_{11}, E_{22}\}$	{138, 8.7} GPa
G_{12}	5 GPa
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.28, 0.45}

TABLE 2: MATERIAL STRENGTHS OF A PLY.

Material property	Value
$\{\sigma_{T1}, \sigma_{T2}, \sigma_{T3}\}$	{2060, 78, 78} MPa
$\{\sigma_{C1}, \sigma_{C2}, \sigma_{C3}\}$	{1590, 200, 200} MPa
$\{\sigma_{S12}, \sigma_{S23}, \sigma_{S13}\}$	{157, 157, 157} 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.

Expressing the objective in terms of a maximum operator can cause numerical issues and therefore it is often better to use an approximate maximum. In this model a p-norm ($P = 10$) is used, that is

$$\max(x) \approx \left(\int_\Omega x^P d\Omega / \int_\Omega d\Omega\right)^{1/P}$$

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 near to one, indicating possibility of composite failure by different failure modes at the given load. The unidirectional composites have maximum load carrying capacity in the fiber direction. For this reason, the stress in the fiber direction in each lamina 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 above one in the top 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 stress in fiber direction (Figure 7) and the failure index (Figure 8) are reduced considerably. The maximum stress and maximum failure index values are reduced substantially for the optimized layup. The reduction of the failure index well below 1 indicates that the structure is safe for the given loading conditions.

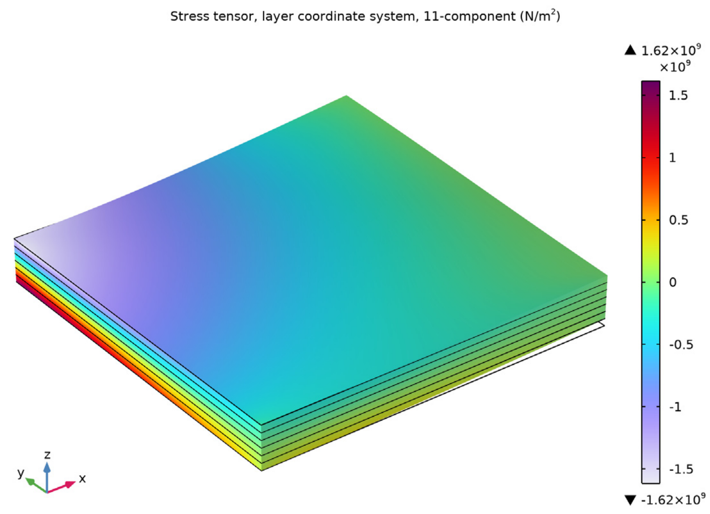

Figure 5: von Mises stress distribution in the laminate with the original layup.

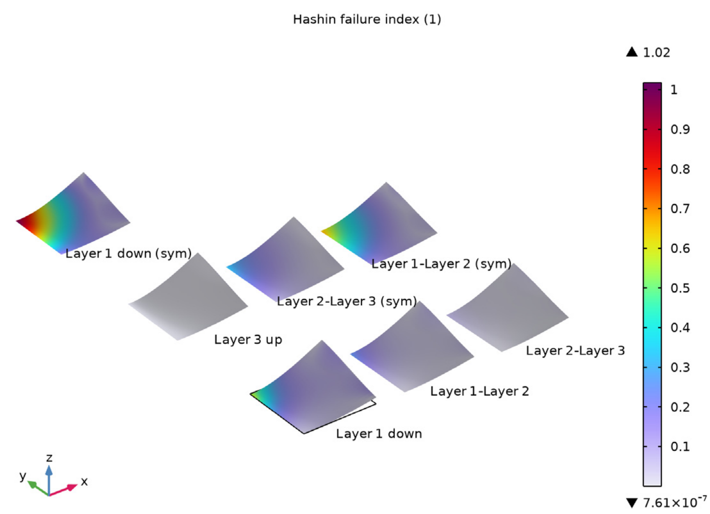

Figure 6: Distribution of the Hashin failure index in each interface of the laminate for the original layup.

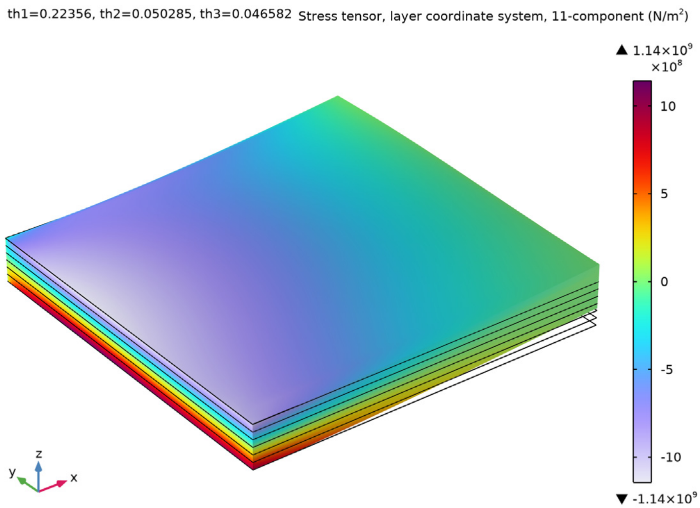

Figure 7: von Mises stress distribution in the laminate for the optimized layup.

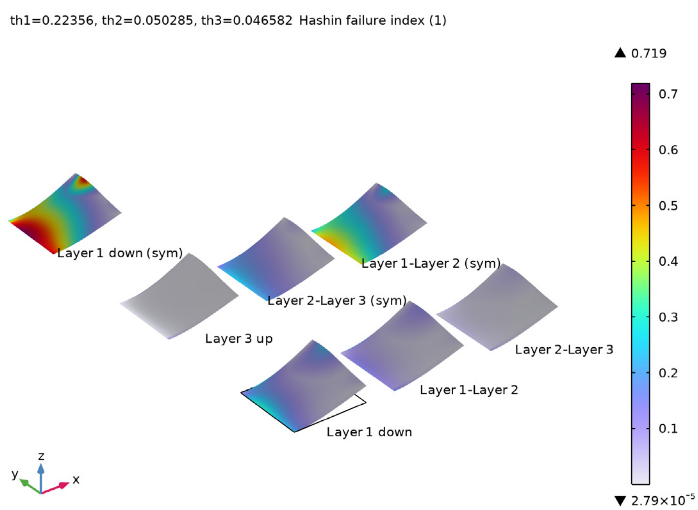

Figure 8: Distribution of the Hashin failure index in each interface of the laminate for the optimized layup.

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

- Original layup: $[0/0/0]_s$
- Optimized layup: $[12.81/2.88/2.67]_s$ (after round-off)

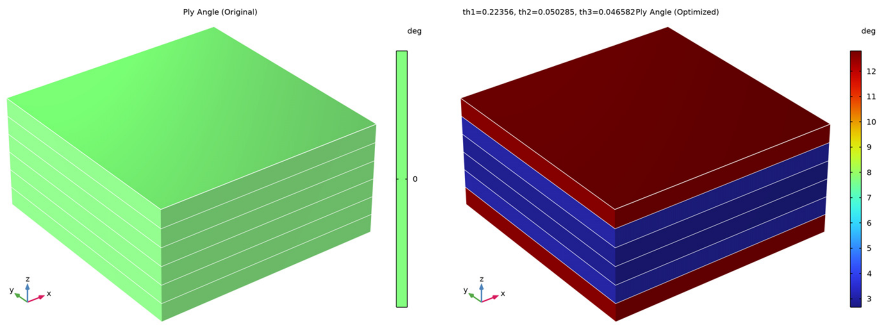

Figure 9: Original and optimized ply angles.

Figure 10 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.

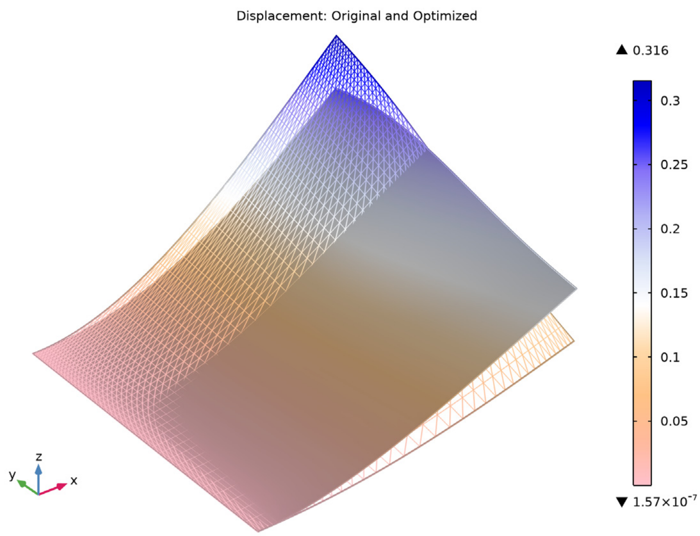

Figure 10: Displacement magnitude plotted on the deformed configuration for the original (wireframe) and optimized (solid) layups.

Notes About the COMSOL Implementation

- Modeling a composite laminate as a layered shell requires a surface geometry, in general referred to as a base surface, and a **Layered Material** node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other having different thicknesses, material properties, and fiber orientations. You can optionally specify the interface materials between the layers, and control the number of through-thickness mesh elements for each layer.
- The third direction for the selected coordinate system in the **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack** represents the normal direction of the **Layered Shell** or **Shell** physics. This is also the direction in which the layer stacking is interpreted from bottom to top, and therefore, it is crucial to know it during modeling. There are two ways to achieve this:
  - Using physics symbols: Go to the physics settings, find the **Physics Symbols** section, and select the **Enable physics symbols** checkbox. Then go to the material feature, for

instance, **Linear Elastic Material**, to see the normal direction represented by green arrows in the geometry.

- Using result templates: When a solution dataset is available, use the result template **Thickness and Orientation** to plot the normal direction.

- From a constitutive model point of view, you can either use the *Layerwise (LW)* theory based **Layered Shell** interface, or the *Equivalent Single Layer (ESL)* theory based **Linear Elastic Material, Layered** node in the **Shell** interface. The laminated composite presented in the current model is modeled using the **Layered Shell** interface.
- The built-in **Composites** material library contains data for fiber and matrix constituents as well as for unidirectional and bidirectional laminae.

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`.

COMSOL Multiphysics is equipped with built-in material properties for a number of lamina materials. Select the needed materials from the **Composites** material folder in the built-in material library.

ADD MATERIAL

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

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

3 In the tree, select **Composites** > **Laminae** > **Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%]**.

4 Right-click and choose **Add to Global Materials**.

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

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

GLOBAL DEFINITIONS

Layered Material: [th1/th2/th3]

1 In the **Model Builder** window, under **Global Definitions** 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	Value	Thickness	Mesh elements
Layer 1	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat1)	th1	0 rad	d_layer	1

4 Click **Add** two times.

5 In the table, enter the following settings:

Layer	**Material**	**Rotation**	**Value**	**Thickness**	**Mesh elements**
`Layer 2`	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat1)	`th2`	0 rad	`d_layer`	1
`Layer 3`	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat1)	`th3`	0 rad	`d_layer`	1

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

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 `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 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**.

LAYERED SHELL (LSHELL)

Linear Elastic Material 1

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

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**.

Spring Foundation 1

1 In the **Physics** toolbar, click **Edges** and choose **Spring Foundation**.

2 Select Edge 1 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the **Spring type** list, choose **Total spring constant**.

5 From the list, choose **Diagonal**.

6 Specify the $\mathbf{k}_{\mathrm{tot}}$ matrix as

1e12	0	0
0	1e12	0
0	0	1e9

Boundary Load 1

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

2 Select Edge 4 only.

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

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

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

F*(Y/a)	z

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

DEFINITIONS (COMP1)

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

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

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 Click **Load from File**.

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

The maximum stress concentration occurs at the constrained edge. To obtain a sufficiently accurate solution, refine the mesh in this region.

MESH 1

Mapped 1

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

2 Select Boundary 1 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 1 and 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `40`.

6 In the **Element ratio** text field, type `5`.

7 From the **Growth rate** list, choose **Exponential**.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 2 and 3 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `40`.

6 In the **Element ratio** text field, type `5`.

7 From the **Growth rate** list, choose **Exponential**.

8 Select the **Symmetric distribution** checkbox.

9 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 **Study** toolbar, click **Compute**.

RESULTS

Failure Indices and Fiber Orientations (Original)

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

2 In the **Settings** window for **Evaluation Group**, type `Failure Indices and Fiber Orientations (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** checkbox.

5 Click to expand the **Format** section. From the **Include parameters** list, choose **Off**.

Volume Maximum 1

1 Right-click **Failure Indices and Fiber Orientations (Original)** and choose **Maximum** > **Volume Maximum**.

2 In the **Settings** window for **Volume Maximum**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
th1	deg	Fiber orientation, layer 1
th2	deg	Fiber orientation, layer 2
th3	deg	Fiber orientation, layer 3
lshell.lemm1.sf1.f_ifT	1	Hashin fiber tensile failure index
lshell.lemm1.sf1.f_ifC	1	Hashin fiber compressive failure index
lshell.lemm1.sf1.f_imT	1	Hashin matrix tensile failure index
lshell.lemm1.sf1.f_imC	1	Hashin matrix compressive failure index
lshell.lemm1.sf1.f_iiT	1	Hashin interlaminar tensile failure index
lshell.lemm1.sf1.f_iiC	1	Hashin interlaminar compressive failure index

4 In the **Failure Indices and Fiber Orientations (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.

Stress, Layer Coordinate System (Original)

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

2 In the **Settings** window for **3D Plot Group**, type Stress, Layer Coordinate System (Original) in the **Label** text field.

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

4 In the **Stress, Layer Coordinate System (Original)** toolbar, click **Plot**.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

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

4 Click the **Add Result Template** button 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** checkbox.

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** checkbox.

8 In the **Relative separation** text field, type `0.7`.

9 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

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

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

12 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.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study 1: Original Layup/Solution 1 (sol1)** > **Layered Shell** > **Geometry and Layup (lshell)** > **Ply Angle (lshell)**.

3 Click the **Add Result Template** button in the window toolbar.

RESULTS

Ply Angle (lshell)

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

2 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`.

Surface 1

1 In the **Model Builder** window, expand the **Results** > **Ply Angle (lshell)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Clear the **Manual color range** checkbox.

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**.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study 1: Original Layup/Solution 1 (sol1)** > **Layered Shell** > **Applied Loads (lshell)** > **Boundary Loads (lshell)**.

3 Click the **Add Result Template** button in the window toolbar.

RESULTS

Boundary Loads (Original)

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

2 In the **Model Builder** window, expand the **Boundary Loads (Original)** node.

Boundary Load 1

1 In the **Model Builder** window, expand the **Results** > **Boundary Loads (Original)** > **Boundary Load 1** node, then click **Boundary Load 1**.

2 In the **Settings** window for **Arrow Line**, locate the **Coloring and Style** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `15E-9`.

4 In the **Boundary Loads (Original)** toolbar, click **Plot**.

After solving the model for the original layup, add an optimization analysis for optimizing the layup for given loading and boundary 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 the **Add Study** button in the window toolbar.

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

STUDY 2: LAYUP OPTIMIZATION

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

General Optimization

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

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

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

4 In the **Optimality tolerance** text field, type `0.001`.

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

Expression	Description	Evaluate for
`comp1.FI_max`	`Measure of maximum failure index`	Stationary

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

7 In the table, enter the following settings:

Parameter	Initial value	Scale	Lower bound	Upper bound	Unit
th1 (Fiber orientation, layer 1)	`0[deg]`	1	`-90[deg]`	`90[deg]`	rad
th2 (Fiber orientation, layer 2)	`0[deg]`	1	`-90[deg]`	`90[deg]`	rad
th3 (Fiber orientation, layer 3)	`0[deg]`	1	`-90[deg]`	`90[deg]`	rad

8 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`.

Stress, Layer Coordinate System (Optimized)

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

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

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

4 In the **Stress, Layer Coordinate System (Optimized)** toolbar, click **Plot**.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

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

3 Click the **Add Result Template** button 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** checkbox.

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** checkbox.

8 In the **Relative separation** text field, type `0.7`.

9 Locate the **Quality** section. From the **Resolution** list, choose **No refinement**.

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

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**.

RESULT TEMPLATES

1 Go to the **Result Templates** 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 the **Add Result Template** button in the window toolbar.

4 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Ply Angle (Ishell)

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

2 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`.

Surface 1

1 In the **Model Builder** window, expand the **Results** > **Ply Angle (Ishell)** node, then click **Surface 1**.

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

3 Clear the **Manual color range** checkbox.

Ply Angle (Optimized)

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

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**.

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

Failure Index, Slice (Original) 1

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

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

3 Clear the **Plot dataset edges** checkbox.

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** checkbox.

8 Locate the **Coloring and Style** section. From the **Color table** list, choose **Twilight**.

9 Select the **Wireframe** checkbox.

Layered Material Slice 2

Right-click **Results** > **Failure Index, Slice (Original) 1** > **Layered Material Slice 1** and choose **Duplicate**.

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** checkbox. In the associated text field, type `1`.

Transparency 1

1 In the **Model Builder** window, right-click **Layered Material Slice 1** and choose **Transparency**.

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

3 Find the **Transparency** subsection. In the **Transparency** text field, type `0.2`.

Layered Material Slice 2

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

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

Line 1

1 In the **Model Builder** window, right-click **Failure Index, Slice (Original) 1** and choose **Line**.

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

3 From the **Dataset** list, choose **Layered Material**.

4 Locate the **Expression** section. In the **Expression** text field, type `lshell.atxd1(lshell.d/2,mean(lshell.disp))`.

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

6 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

7 Select the **Radius scale factor** checkbox. In the associated text field, type `0.001`.

8 From the **Color table** list, choose **Twilight**.

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

Deformation 1

1 Right-click **Line 1** and choose **Deformation**.

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

3 In the **x-component** text field, type `lshell.umX`.

4 In the **y-component** text field, type `lshell.umY`.

5 In the **z-component** text field, type `lshell.umZ`.

Line 1

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

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

3 From the **Dataset** list, choose **From parent**.

Line 2

1 Right-click **Results** > **Failure Index, Slice (Original) 1** > **Line 1** and choose **Duplicate**.

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

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

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.

Failure Indices and Fiber Orientations (Optimized)

1 In the **Model Builder** window, right-click **Failure Indices and Fiber Orientations (Original)** and choose **Duplicate**.
2 In the **Settings** window for **Evaluation Group**, type `Failure Indices and Fiber Orientations (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 Indices and Fiber Orientations (Optimized)** toolbar, click = **Evaluate**.