Created in COMSOL Multiphysics 6.0

Stacking Sequence Optimization

Introduction

Composite laminates are synthetic structures and there is always a possibility to optimize its design it terms of number of layers, material of each layer, thickness of each layer, and stacking sequence for the specified loading conditions. Many times number of layers, layer materials, and layer thicknesses are also governed by other factors however there is always a possibility to find the optimum stacking sequence which gives lesser stresses in the structure for the specified loading conditions.

This example illustrates the optimization of stacking sequence in a composite laminate. The composite laminate considered for the analysis has six layers with symmetric layup. The Carbon-Epoxy material having orthotropic material properties is used as a lamina material. The optimization analysis is performed to find the optimum fiber orientation in each layer under specified loading conditions with an objective of minimizing maximum stress value in the laminate. The layup is assumed symmetric thus three ply angles are the control variables and BOBYQA method is applied to find the optimum stacking sequence.

Model Definition

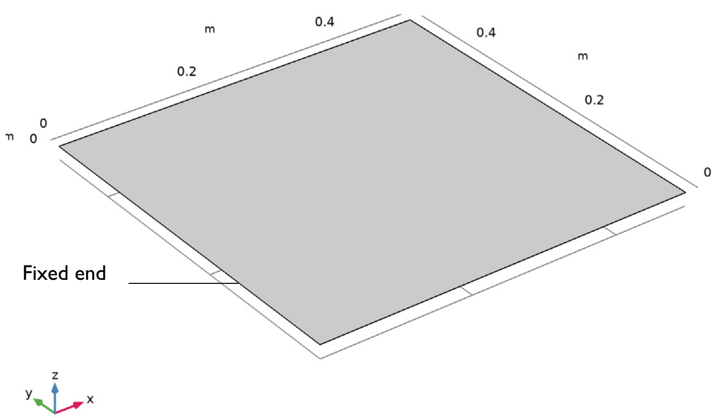

Figure 1: Model geometry of a composite laminate.

GEOMETRY AND BOUNDARY CONDITIONS

The model geometry of a composite laminate with a side length of 0.5 m is shown in Figure 1. Boundary conditions and loading are:

- Left end of the composite laminate is fixed.
- A total load of 1 kN is applied to one of the corners of the right end of the laminate in the form of a line load as shown in Figure 2.

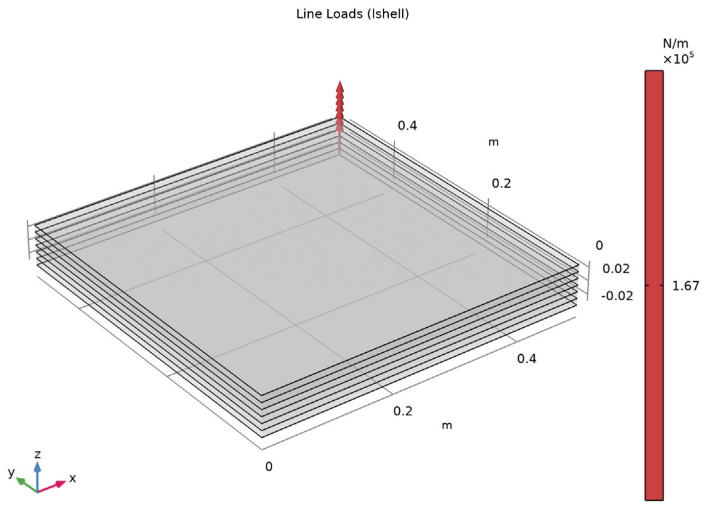

Figure 2: The 3D representation of model geometry together with the applied line load. Note that geometry is scaled by a factor of 10 in the thickness direction for the visualization purpose.

STACKING SEQUENCE

The laminate considered for the analysis has 6 layers with symmetric layup. The original ply angles are assumed to be zero and are optimized for minimizing the maximum stress value in the laminate under given loading. The through-thickness view and the original layup of the laminate can be seen in Figure 3 and Figure 4.

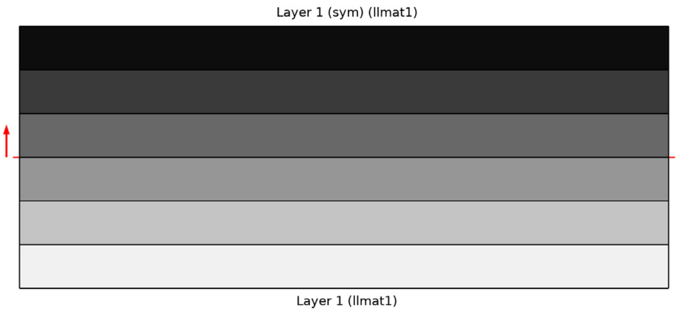

Figure 3: Through-thickness view of the laminated material with six layers.

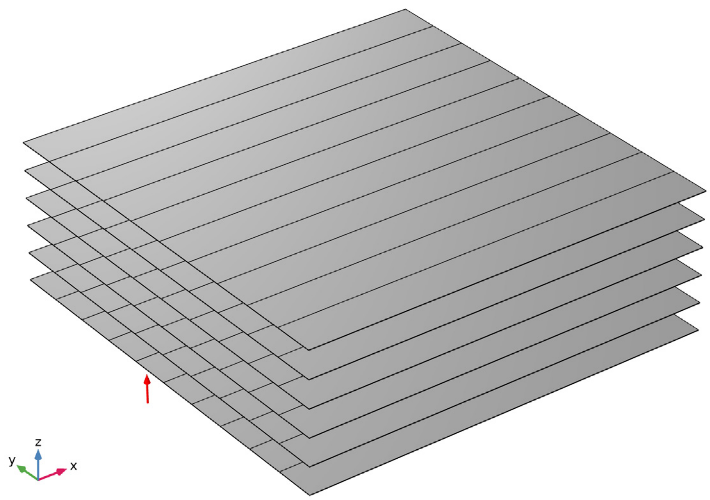

Figure 4: Stacking sequence $[0]_6$ of the original layup of the composite laminate.

MATERIAL PROPERTIES

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

TABLE 1: MATERIAL PROPERTIES OF A PLY.

Material property	Value
$\{E_1, E_2, E_3\}$	{134, 9.2, 9.2} GPa
$\{G_{12}, G_{23}, G_{13}\}$	{4.8, 4.8, 4.8} GPa
$\{\upsilon_{12}, \upsilon_{23}, \upsilon_{13}\}$	{0.28, 0.28, 0.28}

LAYUP OPTIMIZATION

In the original layup, all plies are assumed to be aligned with the laminate coordinate system axis or in other words they have zero ply angles. The objective is to optimize the ply angles in order to minimize the maximum stress values in the entire laminate.

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

Results and Discussion

The von Mises stress distribution in the composite laminate for original layup is shown in Figure 5. The layerwise distribution of von Mises stress at the middle of each layer can be seen in Figure 6. The corresponding plots for the optimized layup are shown in Figure 7 and Figure 8.

It can be seen here that the maximum stress values are reduced by 60% in the optimized layup case. If we look at the stress distribution, it is more even within the plies as well as across the plies in optimized layup case. This helps in distributing the high stresses generated around the constrained points closer to location of applied load.

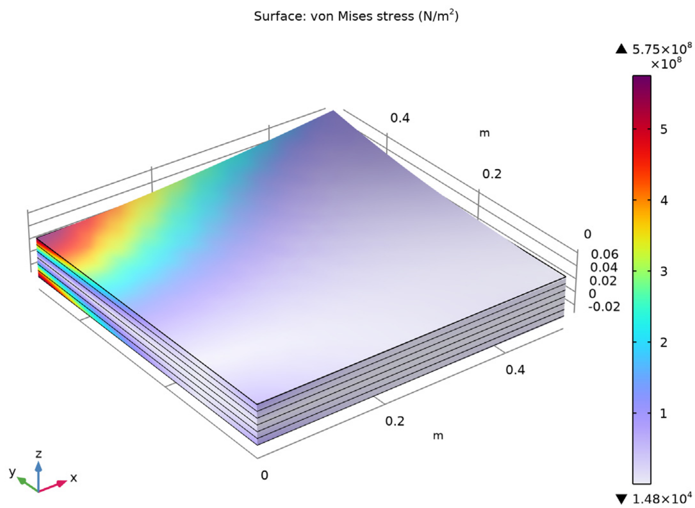

Figure 5: Von Mises stress distribution in the composite laminate for original layup.

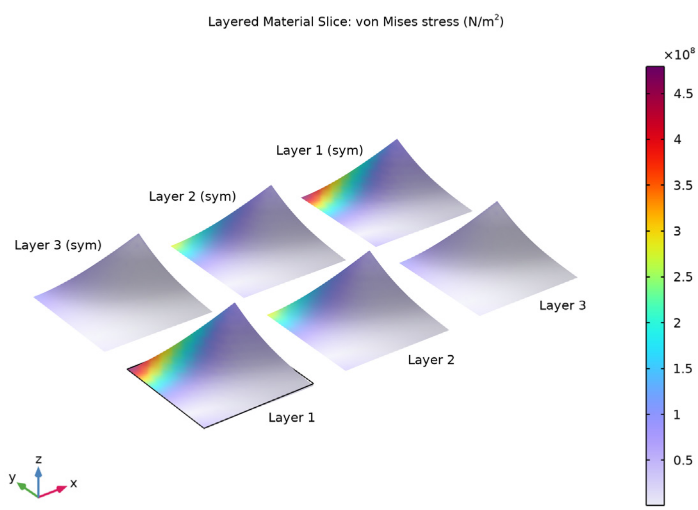

Figure 6: Layerwise von Mises stress distribution in the composite laminate for the original layup.

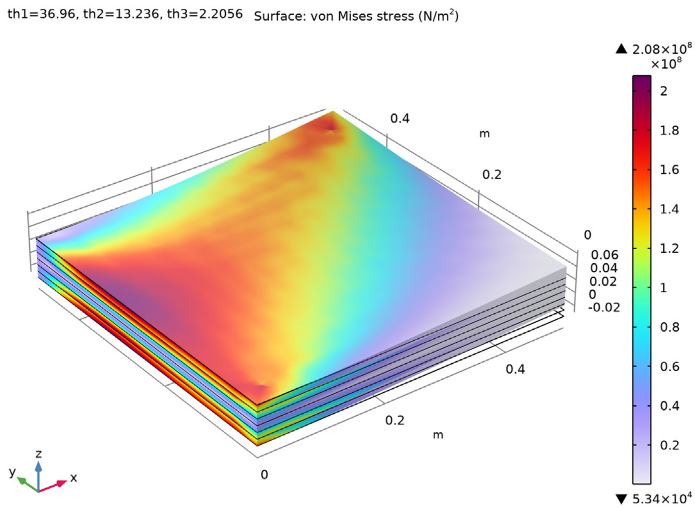

Figure 7: Von Mises stress distribution in the composite laminate for the optimized layup.

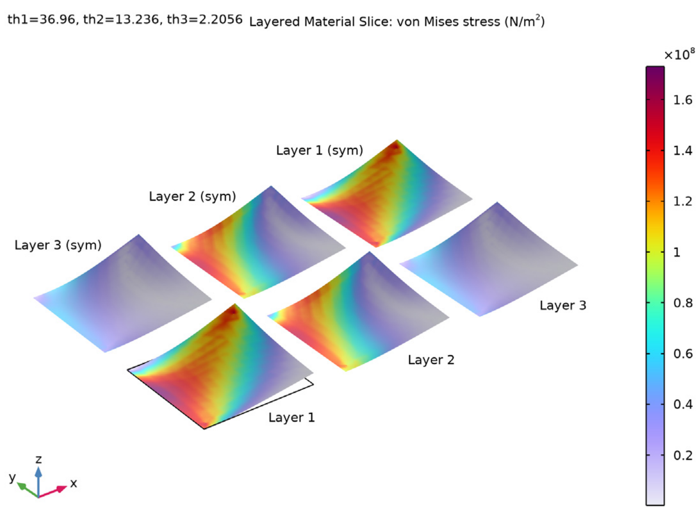

Figure 8: Layerwise von Mises stress distribution in the composite laminate for the optimized layup.

The original and optimized ply angles are shown in Figure 9. Similarly original and optimized principal material directions can be seen in Figure 10.

The original layup and optimized layup are as follows:

- Original layup: $[0/0/0]_s$
- Optimized layup: $[37/13/2]_s$ (after round-off)

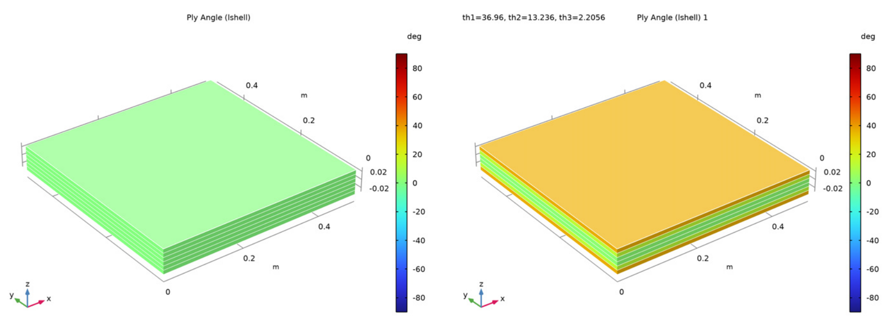

Figure 9: Original and optimized ply angles.

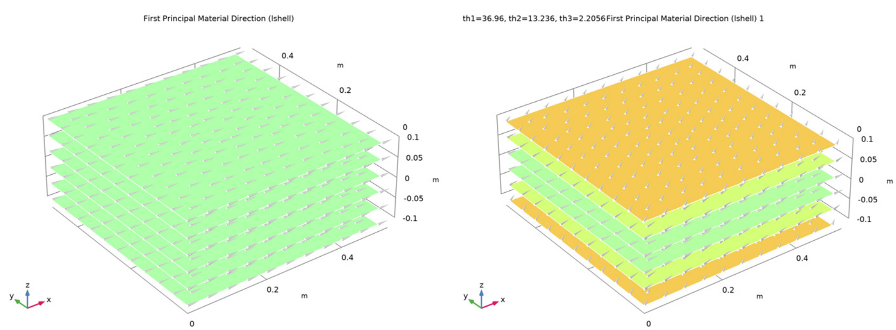

Figure 10: Original and optimized principal material directions.

Figure 11 shows the comparison of displacement and deformation profile of the laminate under given loading for original and optimized layup. Here original layup result is plotted in wireframe mode whereas optimized layup result is plotted as solid surface plot. It can be seen that the optimized layup has higher local stiffness at the loading point and predominantly goes in a bending mode compared to the original layup which goes in bending-twisting mode.

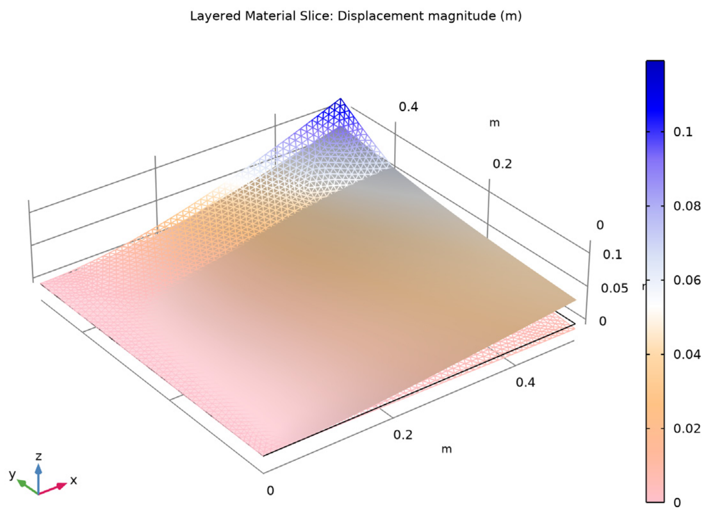

Figure 11: Displacement and deformation profile in a composite laminate for original layup (wireframe) and optimized layup (solid).

Notes About the COMSOL Implementation

The objective function for the optimization requires the computation of maximum stress values in the composite laminate. Under the given loading conditions, the bending would occur in the composite which would give maximum stresses in the outer layers. Hence, the maximum stress value in the outer interfaces of each layer is found out, and then the maximum value among all the layers is found out.

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`.

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 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	{Evector1, Evector2, Evector3}	`{E1, E2, E3}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{nu12, nu23, nu13}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{G12, G23, G13}`	N/m^2	Orthotropic
Density	rho	1	kg/m^3	Basic

LAYERED SHELL (LSHELL)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Layered Shell (lshell)** 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 $\mathbf{F}_{\mathrm{tot}}$ vector as

0	x
0	y
F	z

Define a global variable, corresponding to the maximum von mises stress 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**
`mises_max_l1`	`maxop1(lshell.atxd1(0,lshell.mises))`	N/m²	Maximum von Mises stress, layer 1
`mises_max_l2`	`maxop1(lshell.atxd1(d_layer,lshell.mises))`	N/m²	Maximum von Mises stress, layer 2
`mises_max_l3`	`maxop1(lshell.atxd1(2*d_layer, lshell.mises))`	N/m²	Maximum von Mises stress, layer 3
`mises_max`	`max(max(mises_max_l1, mises_max_l2), mises_max_l3)`	N/m²	Maximum von Mises stress

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

Global Evaluation 1

1 In the **Results** toolbar, click **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
`mises_max`	`N/m^2`	`Maximum von Mises stress`

4 Click **Evaluate**.

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

Layered Material 1

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Layered Material 1**.

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

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

Layered Material 2 (Shell Geometry)

1 In the **Model Builder** window, expand the **Results>Datasets>Layered Materials** node, then 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 `10`.

Layered Material 3 (Material Direction)

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

Stress (Original)

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

2 In the **Settings** window for **3D Plot Group**, type `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 **Stress (Original)** toolbar, click **Plot**.

Stress, Slice (Original)

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

Layered Material Slice 1

1. In the **Model Builder** window, expand the **Stress, Slice (Original)** node, then click **Layered Material Slice 1**.
2. In the **Settings** window for **Layered Material Slice**, locate the **Through-Thickness Location** section.
3. From the **Location definition** list, choose **Layer midplanes**.
4. Locate the **Layout** section. From the **Displacement** list, choose **Rectangular**.
5. In the **Relative x-separation** text field, type `0.15*2`.
6. In the **Relative y-separation** text field, type `0.15*2`.
7. Select the **Show descriptions** check box.
8. In the **Relative separation** text field, type `0.2*2`.
9. In the **Stress, Slice (Original)** toolbar, click **Plot**.
10. Click the **Zoom Extents** button in the **Graphics** toolbar.

Stress, Slice (Original)

1. In the **Model Builder** window, click **Stress, 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 **Stress, Slice (Original)** toolbar, click **Plot**.
5. Click the **Show Grid** button in the **Graphics** toolbar.

Stress, Through Thickness (Original)

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

Through Thickness 1

1. In the **Model Builder** window, expand the **Stress, Through Thickness (Original)** node, then click **Through Thickness 1**.

2 In the **Settings** window for **Through Thickness**, locate the **Selection** section.

3 Click **Clear Selection**.

4 Select Point 2 only.

5 Locate the **x-Axis Data** section. From the **Unit** list, choose **MPa**.

6 In the **Stress, Through Thickness (Original)** toolbar, click **Plot**.

Ply Angle (lshell)

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

2 In the **Model Builder** window, expand the **Results>Geometry and Layup (lshell)** node, then click **Ply Angle (lshell)**.

3 In the **Ply Angle (lshell)** toolbar, click **Plot**.

First Principal Material Direction (lshell)

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

2 In the **First Principal Material Direction (lshell)** toolbar, click **Plot**.

Geometry and Layup (Original)

1 In the **Model Builder** window, under **Results** click **Geometry and Layup (lshell)**.

2 In the **Settings** window for **Group**, type `Geometry and Layup (Original)` in the **Label** text field.

Applied Loads (Original)

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

2 In the **Settings** window for **Group**, type `Applied Loads (Original)` in the **Label** text field.

Line Loads (lshell)

1 In the **Model Builder** window, expand the **Applied Loads (Original)** node, then click **Line Loads (lshell)**.

2 In the **Line Loads (lshell)** toolbar, click **Plot**.

After solving the model for original layup, now 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.mises_max`	`Maximum von Mises stress`	Stationary

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

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

7 Click **Add**.

8 In the table, enter the following settings:

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

9 Click **Add**.

10 In the table, enter the following settings:

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

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

RESULTS

Layered Material 4

1 In the **Model Builder** window, under **Results>Datasets** click **Layered Material 4**.

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

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

Layered Material 5 (Shell Geometry)

1 In the **Model Builder** window, expand the **Results>Datasets>Layered Materials 1** node, then click **Layered Material 5 (Shell Geometry)**.

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

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

Layered Material 6 (Material Direction)

1 In the **Model Builder** window, click **Layered Material 6 (Material Direction)**.

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

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

Stress (Optimized)

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

2 In the **Settings** window for **3D Plot Group**, type `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 **Stress (Optimized)** toolbar, click **Plot**.

Stress, Slice (Optimized)

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

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

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Stress, Slice (Optimized)** node, then click **Layered Material Slice 1**.

2 In the **Settings** window for **Layered Material Slice**, locate the **Through-Thickness Location** section.

3 From the **Location definition** list, choose **Layer midplanes**.

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

5 In the **Relative x-separation** text field, type `0.15*2`.

6 In the **Relative y-separation** text field, type `0.15*2`.

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

8 In the **Relative separation** text field, type `0.2*2`.

Stress, Slice (Optimized)

1 In the **Model Builder** window, click **Stress, 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 **Stress, Slice (Optimized)** toolbar, click **Plot**.

Stress, Through Thickness (Optimized)

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

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

Stress, Through Thickness (Original)

1 In the **Model Builder** window, click **Stress, Through Thickness (Original)**.

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

Through Thickness 1

1 In the **Model Builder** window, expand the **Results>Stress, Through Thickness (Optimized)** node, then click **Through Thickness 1**.

2 In the **Settings** window for **Through Thickness**, locate the **Selection** section.

3 Click **Clear Selection**.

4 Select Point 2 only.

5 Locate the **x-Axis Data** section. From the **Unit** list, choose **MPa**.

6 In the **Stress, Through Thickness (Optimized)** toolbar, click **Plot**.

Geometry and Layup (Optimized)

1 In the **Model Builder** window, under **Results** click **Geometry and Layup (lshell)**.

2 In the **Settings** window for **Group**, type `Geometry and Layup (Optimized)` in the **Label** text field.

Ply Angle (lshell) 1

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

2 In the **Model Builder** window, expand the **Geometry and Layup (Optimized)** node, then click **Ply Angle (lshell) 1**.

3 In the **Ply Angle (lshell) 1** toolbar, click **Plot**.

First Principal Material Direction (lshell) 1

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

2 In the **First Principal Material Direction (lshell) 1** toolbar, click **Plot**.

Applied Loads (Optimized)

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

2 In the **Settings** window for **Group**, type `Applied Loads (Optimized)` in the **Label** text field.

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

Stress, Slice (Original) 1

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

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Stress, 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. From the **Color table** list, choose **Twilight**.

9 Select the **Wireframe** check box.

Layered Material Slice 2

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

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>Stress, 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.

4 In the associated text field, type `1`.

Displacement: Original and Optimized

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