Model created in COMSOL Multiphysics 6.4

Optimization of a Variable-Stiffness Composite Plate with a Central Hole

Introduction

In traditional composite laminates, fibers within each ply are straight, and the global stiffness response is tailored through the selection of an appropriate stacking sequence. In contrast, variable-angle tow (VAT) laminates employ curvilinear fibers within each layer. By allowing the fiber orientation to vary spatially, VAT laminates significantly expand the design space and deliver improved structural performance. This enhanced capability can potentially reduce the required number of plies and the overall weight of the structure.

This example shows how to set up an optimization study to maximize the stiffness of a composite plate using the fiber orientation as the control variable.

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

Model Definition

A one-layer square plate with a hole at its center is considered. The plate is subject to uniaxial traction. Only a quarter of the plate is modeled due to the symmetry of the boundary conditions and loads.

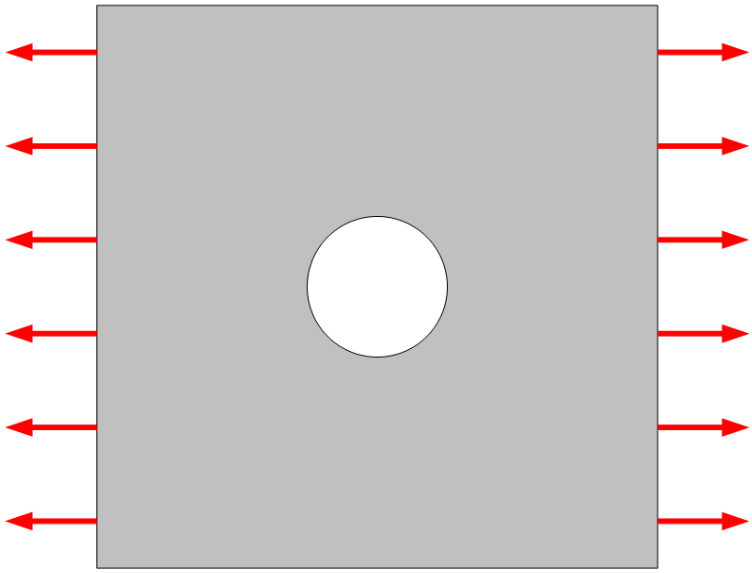

Figure 1: Geometry and applied loads.

To maximize the stiffness, the total elastic strain energy is chosen as the objective function to minimize, and the fiber angle with respect to the load direction is chosen as the control variable.

MATERIAL PROPERTIES

The composite lamina is made of carbon reinforced thermoplastic. The homogenized transversely isotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given in Table 1.

TABLE 1: MATERIAL PROPERTIES OF A LAMINA.

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

Results and Discussion

The optimized solution (Figure 2) is compared to a plate with straight fibers. Figure 3 and Figure 4 show the value of the total strain energy and the maximum displacement for different ply angles using a constant fiber angle. The VAT provides a stiffer solution (lower maximum displacement) with lower level of elastic strain energy. The plate with curvilinear fibers provide a lower maximum stress when comparing the stress fields for the optimal solutions for straight and curvilinear fibers (Figure 5, Figure 6).

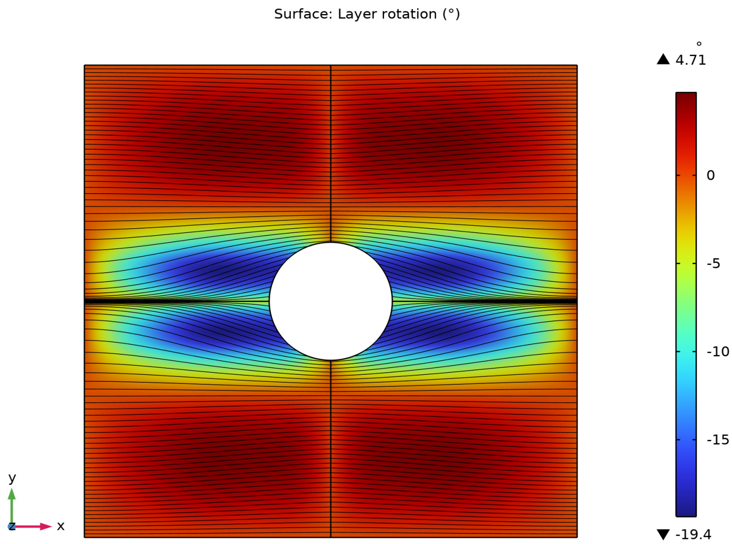

Figure 2: Fiber orientation that maximize the plate stiffness for the given boundary conditions.

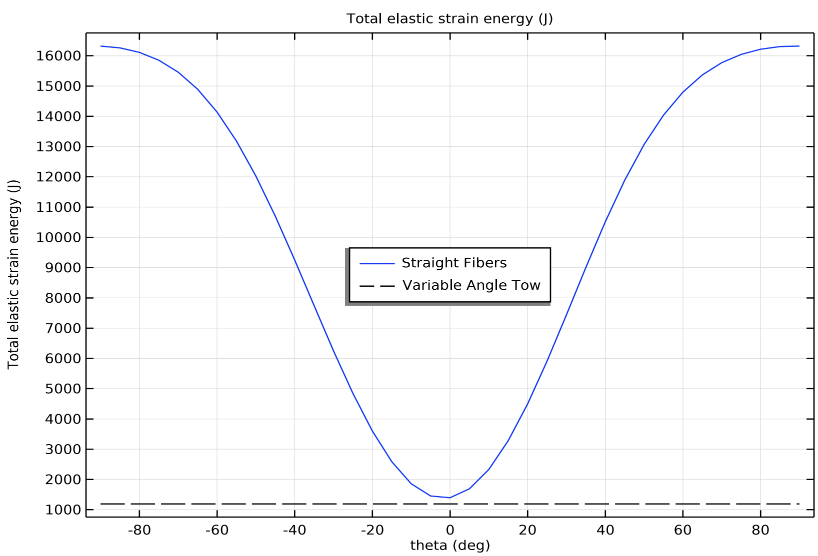

Figure 3: Strain energy values for composites with straight fibers. The horizontal dashed line indicates the optimum value using curvilinear fibers.

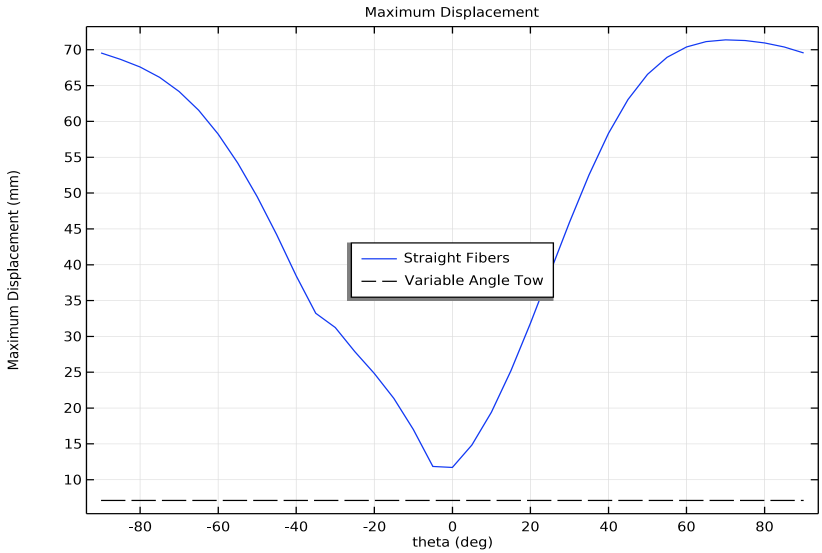

Figure 4: Maximum displacement for composites with straight fibers. The horizontal dashed line indicates the optimum value using curvilinear fibers.

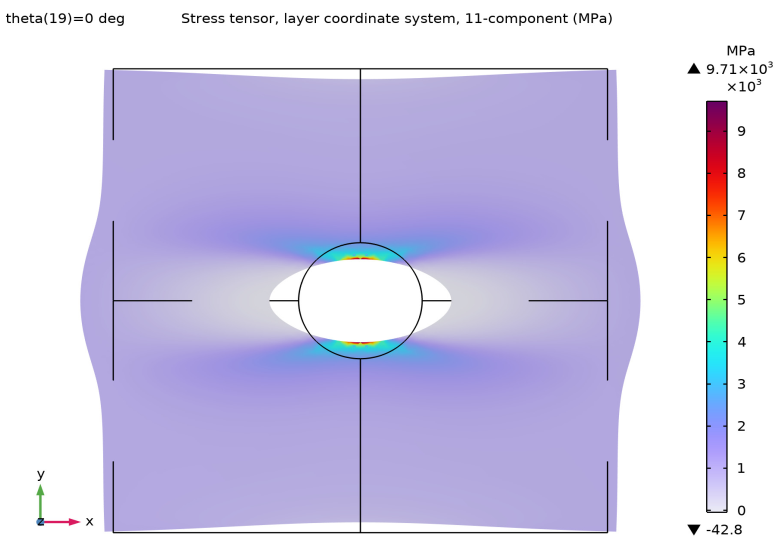

Figure 5: Stress field for the composite plate with straight fibers oriented at 0 deg with respect to the x-axis. The deformation is amplified.

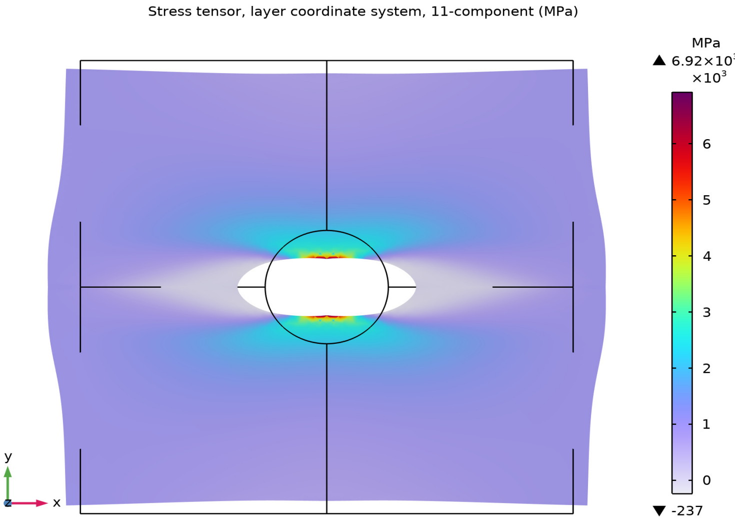

Figure 6: Stress field for the optimum solution with curvilinear fibers. The deformation is amplified.

Notes About the COMSOL Implementation

- Modeling a composite laminate as a layered shell requires a surface geometry, referred to as a the base surface, and a **Layered Material** node, which adds an extra dimension (1D) in the normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other, each having a different thickness, material properties, and fiber orientation. You can optionally specify the interface material between the layers and control the number of through-thickness mesh elements for each layer. To model a curvilinear fiber, add an expression as a function of the local coordinates in the **Rotation** field.
- The third direction for the selected coordinate system in the **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack** represents the normal direction. This is also the direction in which the layer stacking is interpreted from bottom to top, and therefore, it is crucial to visualize 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.

- 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 available in the Layered Shell interface, or the *Equivalent Single Layer* (ESL) theory available in the **Linear Elastic Material, Layered** node in the Shell interface. The laminated composite presented in this example uses 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/fiber_angle_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

Load the parameters from a file.

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

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

3 Click Load from File.

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

Variables (Angle Ply for Straight Fibers)

1 In the **Model Builder** window, right-click **Global 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**
plyAngle	theta	rad	

4 In the **Label** text field, type `Variables (Angle Ply for Straight Fibers)`.

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

Name	**Expression**	**Unit**	**Description**
plyAngle	theta	rad	Fiber orientation with respect X-axis

COMSOL Multiphysics is equipped with built-in material properties for a number of composite materials. Select the needed material 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 1 (lmat1)

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)	`plyAngle`	-	`th`	1

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`.

Work Plane 1 (wp1) > Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r0`.

Work Plane 1 (wp1) > Difference 1 (dif1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **sq1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **c1** only.

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

7 In the **Home** toolbar, click **Build All**.

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 **Orientation and Position** section.

3 Click **Go to Source** for **Coordinate system**.

DEFINITIONS (COMP1)

Boundary System 1 (sys1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** click **Boundary System 1 (sys1)**.

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

3 Find the **Coordinate names** subsection. From the **Axis** list, choose **x**.

LAYERED SHELL (LSHELL)

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

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

3 From the **Displacement field** list, choose **Quadratic serendipity**.

Symmetry 1

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

2 Select Edges 1 and 4 only.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement**.

2 Select Edge 5 only.

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

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

Boundary Load 1

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

2 Select Edge 5 only.

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

4 Specify the $\mathbf{f}_A$ vector as

sigma0	x

MESH 1

Free Triangular 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Free Triangular**, click **Build All**.

Size

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

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

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.

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.

STUDY 1: ORIGINAL ORIENTATION

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

2 In the **Settings** window for **Study**, type `Study 1: Original Orientation` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta (Fiber orientation)	range(-90,5,90)	deg

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1. In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.
2. In the **Settings** window for **Preferred Units**, locate the **Units** section.
3. Click **Add Physical Quantity**.
4. In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.
5. Click **OK**.
6. In the **Settings** window for **Preferred Units**, locate the **Units** section.
7. In the table, enter the following settings:

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

8. Click **Add Physical Quantity**.
9. In the **Physical Quantity** dialog, select **General** > **Displacement (m)** in the tree.
10. Click **OK**.
11. In the **Settings** window for **Preferred Units**, locate the **Units** section.
12. In the table, enter the following settings:

Quantity	Unit	Preferred unit
Displacement	m	mm

13. Select the **Apply conversions to expressions with the same dimensions** checkbox.
14. Click **Apply**.

Layered Material (Original Orientation)

1. In the **Model Builder** window, expand the **Results** > **Datasets** node, then click **Layered Material**.
2. In the **Settings** window for **Layered Material**, type `Layered Material (Original Orientation)` in the **Label** text field.

Use the **Mirror 3D** datasets for full model visualization.

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

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

3 From the **Dataset** list, choose **Layered Material (Original Orientation)**.

Mirror 3D (Original Orientation)

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, type `Mirror 3D (Original Orientation)` in the **Label** text field.

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

4 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

Stress (Original Orientation)

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D (Original Orientation)**.

4 From the **Parameter value (theta (deg))** list, choose **0**.

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

6 Select the **Show units** checkbox.

7 In the **Stress (Original Orientation)** toolbar, click **Plot**.

Now, setup the model to perform an optimization analysis using the fiber angle as control variable and the elastic strain energy as objective function.

Use a **Control Function** to optimize the fiber angle. Set the angle to be bounded between -90 and +90 degree and insert an initial guess.

DEFINITIONS (COMP1)

Control Function 1 (cfunc1)

1 In the **Definitions** toolbar, click **Control Variables** and choose **Control Function**.

2 In the **Settings** window for **Control Function**, locate the **Output** section.

3 In the f_{min} text field, type `-pi/2`.

4 In the f_{max} text field, type `pi/2`.

5 In the c_0 text field, type `sqrt(pi/4)*cos(cfunc1.x*2*pi)`.

6 Locate the **Control Variable Discretization** section. From the **Control type** list, choose **Piecewise Bernstein polynomial**.

7 Locate the **Units** section. Click **Select Input Quantity**.

8 In the **Physical Quantity** dialog, type dim in the text field.

9 In the tree, select **General** > **Dimensionless (1)**.

10 Click **OK**.

Control Function 2 (cfunc2)

1 Right-click **Control Function 1 (cfunc1)** and choose **Duplicate**.

2 In the **Settings** window for **Control Function**, locate the **Output** section.

3 In the c_0 text field, type sqrt(pi/4)*sin(cfunc2.x*2*pi).

GLOBAL DEFINITIONS

Use the control functions to describe the distribution of the angle to be optimized by the solver.

Variables (Angle Ply for Curvilinear Fibers)

1 In the **Model Builder** window, right-click **Global 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
plyAngle	(comp1.cfunc1(X/a)*comp1.cfunc2(Y/a))[rad]	rad	Fiber orientation with respect X-axis

4 In the **Label** text field, type Variables (Angle Ply for Curvilinear Fibers).

STUDY 1: ORIGINAL ORIENTATION

Step 1: Stationary

Disable newly added variables node in the study step of **Study 1: Original Orientation**.

1 In the **Model Builder** window, under **Study 1: Original Orientation** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Global Definitions** > **Variables (Angle Ply for Curvilinear Fibers)**.

5 Click **Disable**.

ADD STUDY

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

STUDY 2: ORIENTATION OPTIMIZATION

In the **Settings** window for **Study**, type `Study 2: Orientation 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 **GCMMA**.

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

Expression	Description	Evaluate for
`comp1.lshell.Ws_tot`	`Total elastic strain energy`	Stationary

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Global Definitions** > **Variables (Angle Ply for Straight Fibers)**.

5 Click **Disable**.

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

RESULTS

Layered Material (Optimization)

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

2 In the **Settings** window for **Layered Material**, type `Layered Material (Optimization)` in the **Label** text field.

Mirror 3D 3

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Mirror 3D 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Layered Material (Optimization)**.

Mirror 3D (Optimized Orientation)

1 In the **Model Builder** window, right-click **Mirror 3D (Original Orientation)** and choose **Duplicate**.

2 In the **Settings** window for **Mirror 3D**, type `Mirror 3D (Optimized Orientation)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 3**.

3D Plot Group 3

In the **Results** toolbar, click **3D Plot Group**.

Stress (Optimized Orientation)

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D (Optimized Orientation)**.

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

5 Select the **Show units** checkbox.

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

Optimized Fiber Orientation

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

2 In the **Settings** window for **3D Plot Group**, type `Optimized Fiber Orientation` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D (Optimized Orientation)**.

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

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

6 In the **Title** text area, type `Surface: Layer rotation (°)`.

7 Locate the **Color Legend** section. Select the **Show units** checkbox.

Surface 1

1 Right-click **Optimized Fiber Orientation** and choose **Surface**.

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

3 In the **Expression** text field, type `llmat1.rot`.

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

5 Click the **Go to XY View** button in the **Graphics** toolbar.

Optimized Fiber Orientation

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

Streamline 1

1 In the **Settings** window for **Streamline**, locate the **Expression** section.

2 In the **x-component** text field, type `lshell.tm11`.

3 In the **y-component** text field, type `lshell.tm12`.

4 In the **z-component** text field, type `lshell.tm13`.

5 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.

6 In the **x** text field, type `r0`.

7 In the **y** text field, type `range(-a,a/50,a)`.

8 In the **z** text field, type `0`.

9 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **Black**.

10 Find the **Line style** subsection. From the **Type** list, choose **Tube**.

11 Select the **Radius scale factor** checkbox.

12 In the **Tube radius expression** text field, type `1e-3`.

Streamline 2

1 Right-click **Results** > **Optimized Fiber Orientation** > **Streamline 1** and choose **Duplicate**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 In the **x** text field, type `-r0`.

Arrow Line 1

1 In the **Model Builder** window, right-click **Optimized Fiber Orientation** and choose **Arrow Line**.

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

3 In the **x-component** text field, type `lshell.bndl1.fax`.

4 In the **y-component** text field, type `lshell.bndl1.fay`.

5 In the **z-component** text field, type `lshell.bndl1.faz`.

6 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh vertices**.

7 Locate the **Coloring and Style** section.

8 Select the **Scale factor** checkbox. In the associated text field, type `8e-11`.

9 In the **Optimized Fiber Orientation** toolbar, click **Plot**.

Total Elastic Strain Energy

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

2 In the **Settings** window for **1D Plot Group**, type `Total Elastic Strain Energy` in the **Label** text field.

3 Locate the **Legend** section. From the **Position** list, choose **Center**.

Global 1

1 Right-click **Total Elastic Strain Energy** and choose **Global**.

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

3 From the **Dataset** list, choose **Study 1: Original Orientation/Solution 1 (sol1)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`lshell.Ws_tot`	`J`	`Total elastic strain energy`

5 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`Straight Fibers`

Line Segments 1

1 In the **Model Builder** window, right-click **Total Elastic Strain Energy** and choose **Line Segments**.

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

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

4 Locate the **x-Coordinates** section. In the table, enter the following settings:

Expression	Unit	Description
-90	1	
90	1	

5 Locate the **y-Coordinates** section. In the table, enter the following settings:

Expression	Unit	Description
`lshell.Ws_tot`	`J`	`Total elastic strain energy`
`lshell.Ws_tot`	`J`	`Total elastic strain energy`

6 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 From the **Color** list, choose **From theme**.

8 Click to expand the **Legends** section. Select the **Show legends** checkbox.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Variable Angle Tow`

11 In the **Total Elastic Strain Energy** toolbar, click **Plot**.

Maximum Displacement

1 Right-click **Total Elastic Strain Energy** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Maximum Displacement` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Maximum Displacement (mm)`.

Global 1

1 In the **Model Builder** window, expand the **Maximum Displacement** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
maxop1(lshell.atxd1(0,lshell.disp))	mm	Maximum 1

Line Segments 1

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

2 In the **Settings** window for **Line Segments**, locate the **y-Coordinates** section.

3 In the table, enter the following settings:

Expression	Unit	Description
maxop1(lshell.atxd1(0,lshell.disp))	mm	Maximum 1
maxop1(lshell.atxd1(0,lshell.disp))	mm	Maximum 1

4 In the **Maximum Displacement** toolbar, click **Plot**.