Model created in COMSOL Multiphysics 6.4

Modeling of a Variable-Angle Tow Laminate

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 demonstrates the modeling of variable-angle tow laminates in which the fiber orientation varies linearly along the global x direction, with each ply assigned a different fiber angle. The results are compared against a full 3D solution obtained using the Solid Mechanics interface.

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

Model Definition

A two-layer square plate is considered. The plate lies in the global xy-plane, with its center located at the origin. Both layers are composed of the same fiber and matrix materials, however, the fiber orientations across the xy-plane differ between the plies.

The following law describes the angle with respect the global x-axis,

$$\phi(x) = \frac{2}{a}(\phi_e + \phi_c)\|x\| + \phi_c \tag{1}$$

where a is the side length, and ϕ_e and ϕ_c correspond to the angles on the edge and at the center of the plate, respectively. The values are given in the following table:

TABLE 1: ANGLES AT EDGE AND CENTER OF THE PLATE FOR BOTH LAYERS.

Angle	**Top layer**	**Bottom Layer**
Angle at the edge	60 deg	45 deg
Angle at the center	0 deg	90 deg

The fiber paths are shown in Figure 1.

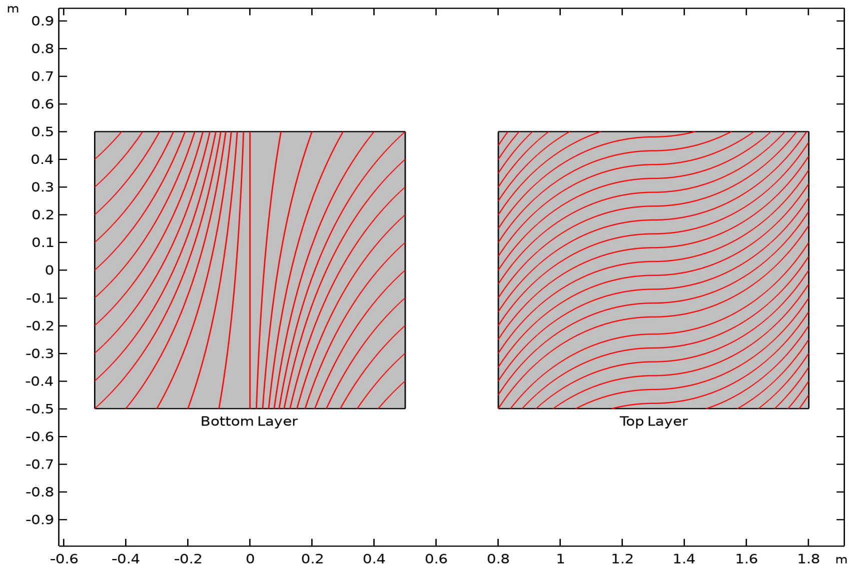

Figure 1: Fiber paths on the bottom and top layers.

MATERIAL PROPERTIES

The composite lamina is made of carbon fibers in an epoxy resin. The homogenized transversely isotropic material properties, Young's modulus, shear modulus, and Poisson's ratio, are given in Table 2.

TABLE 2: MATERIAL PROPERTIES OF A LAMINA.

Material property	**Value**
$\{E_{11}, E_{22}\}$	{40, 10} GPa
G_{12}	4.5 GPa
$\{\nu_{12}, \nu_{23}\}$	{0.28, 0.28}

Results and Discussion

The first principal material direction in the layers is shown in Figure 2.

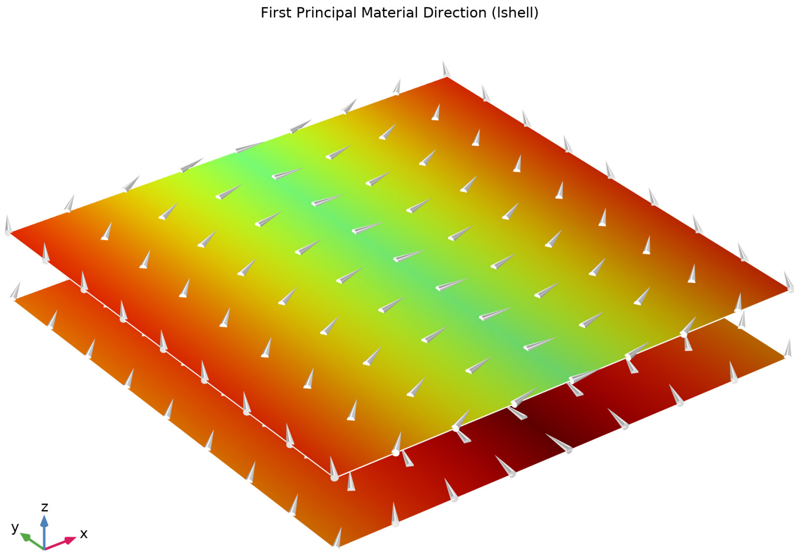

Figure 2: The first principal material direction in the layers. It coincides with the fiber direction.

The displacement and stress components across the thickness at a particular point are shown in Figure 3 and Figure 4. It can be seen that both in-plane and out-of-plane displacements are continuous across the interfaces. The same holds for the transverse shear and normal stress components, as shown in Figure 5 and Figure 6. This is due to the stress continuity in the normal direction

$$\sigma_{bot} \cdot \mathbf{n} = \sigma_{top} \cdot \mathbf{n} \tag{2}$$

The in-plane normal and shear stresses are discontinuous across layers with discontinuous material properties (Figure 7 and Figure 8). In the absence of delamination, the compatibility equation ensures continuity for the in-plane strains, which implies that the in-plane stresses are discontinuous when the laminae have different material properties.

The results obtained with the Layered Shell interface closely match the results obtained with the Solid Mechanics interface.

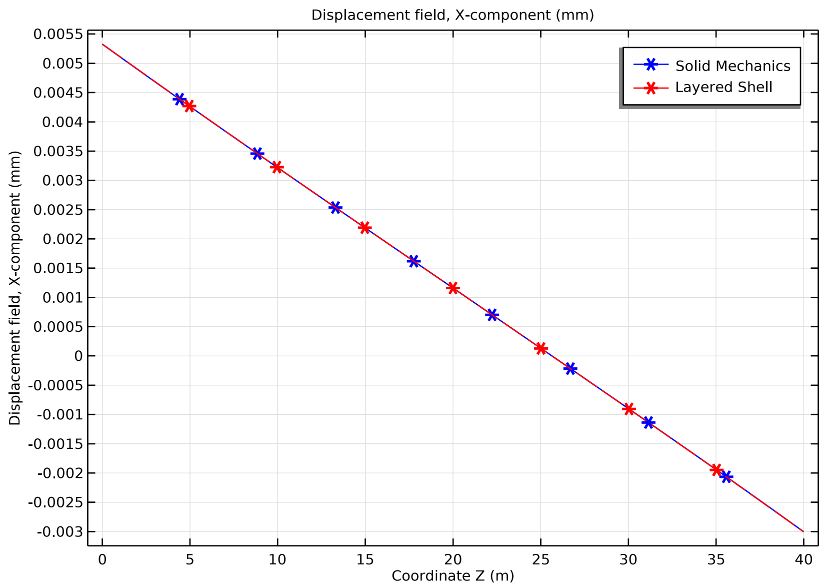

Figure 3: x-component of the displacement across the laminate thickness.

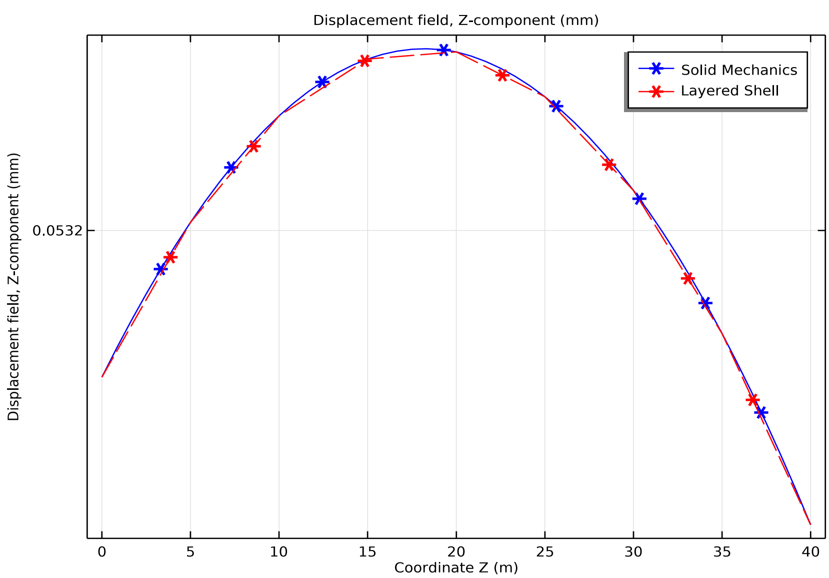

Figure 4: z-component of the displacement across the laminate thickness.

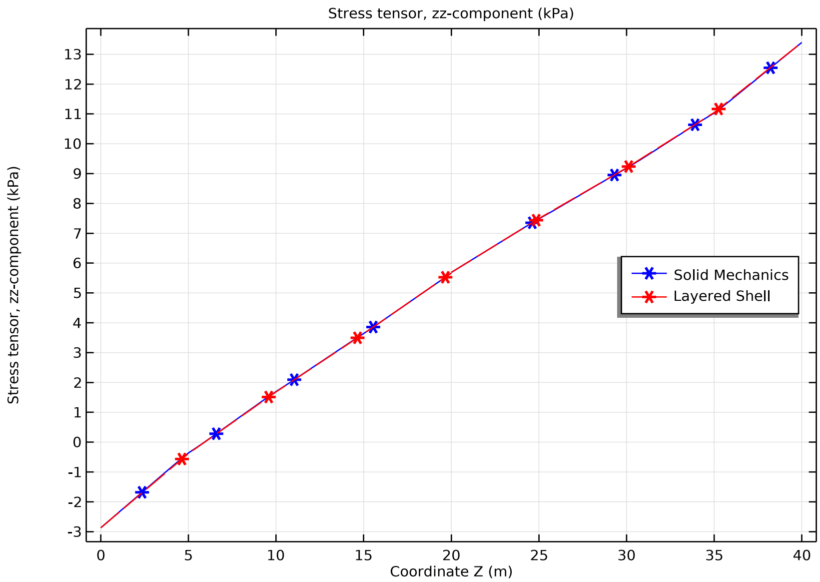

Figure 5: zz-component of the stress tensor across the laminate thickness.

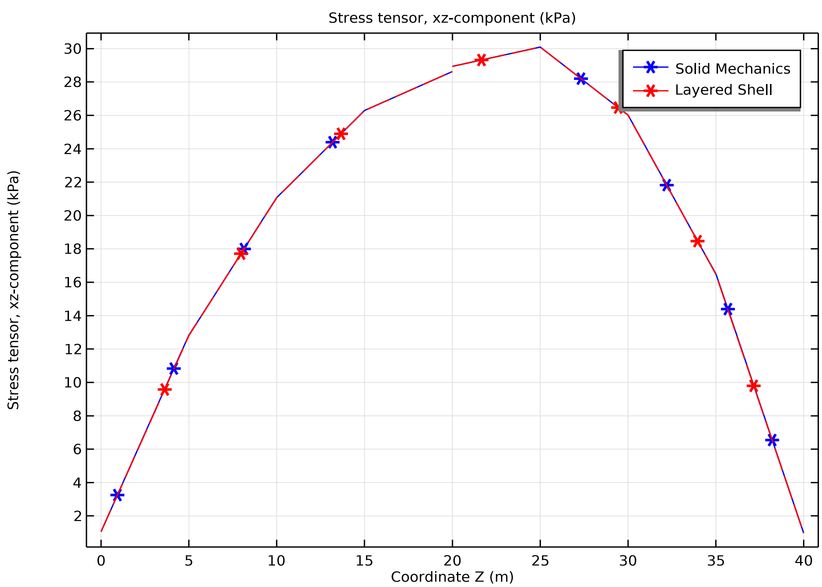

Figure 6: xz-component of the stress tensor across the laminate thickness.

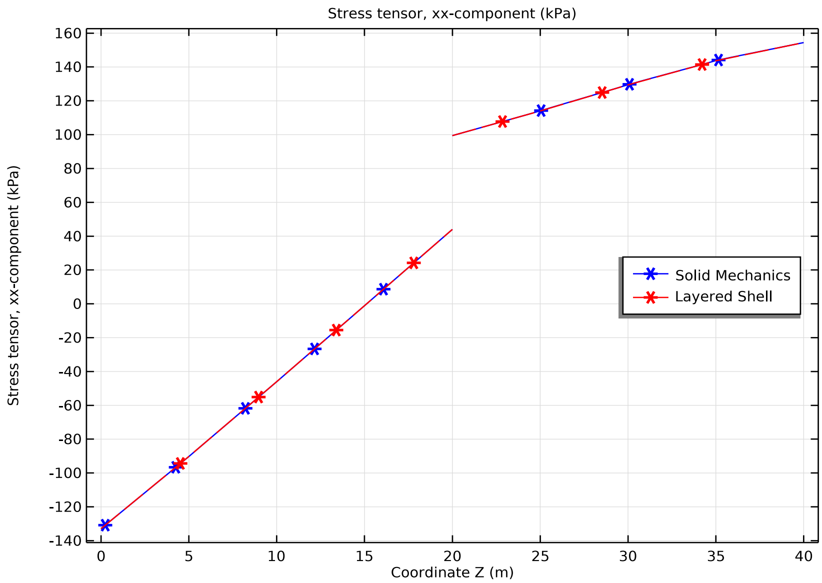

Figure 7: xx-component of the stress tensor across the laminate thickness.

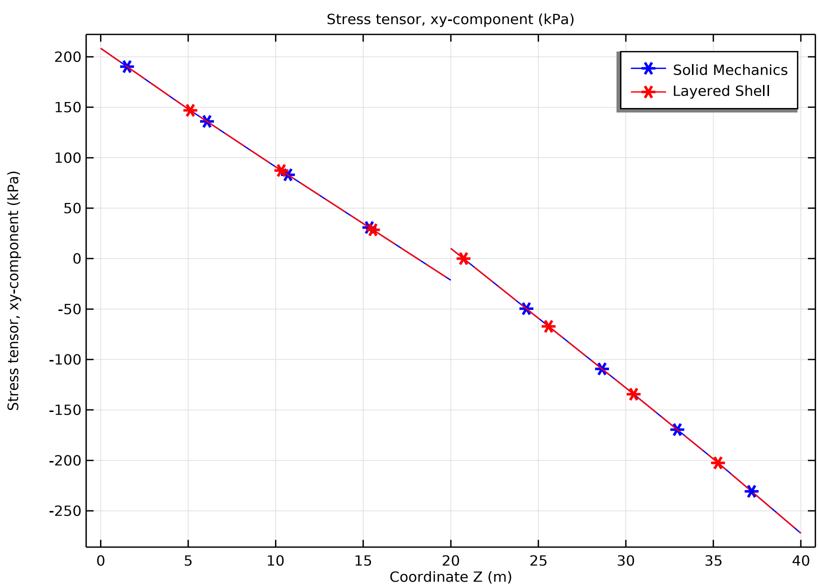

Figure 8: xy-component of the stress tensor across the laminate thickness.

Notes About the COMSOL Implementation

- Modeling a composite laminate requires a surface geometry, referred to as the base surface, and a **Layered Material** node, which adds an extra dimension in the normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other, each with a different thickness, material properties, and fiber orientation. You can optionally specify the interface materials 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 viewpoint, 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/variable_angle_tow_laminate`

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 In the **Select Physics** tree, select **Structural Mechanics > Solid Mechanics (solid)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies > Stationary**.
8 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 `variable_angle_tow_laminate_parameters.txt`.

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type `a`.
4 In the **Depth** text field, type `a`.
5 In the **Height** text field, type `2*th`.
6 Locate the **Position** section. From the **Base** list, choose **Center**.
7 In the **z** text field, type `th`.
8 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	th

9 Click **Build All Objects**.

GLOBAL DEFINITIONS

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 FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Composites** > **Laminae** > **Unidirectional fiber lamina: E-glass/epoxy [fiber volume fraction 60%]**.

3 Click the **Add to Global Materials** button in the window toolbar.

4 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

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: E-glass/epoxy [fiber volume fraction 60%] (mat1)	`0.0`	0 rad	`th`	`nelz`

4 Click **Add**.

Create the functions to describe the fiber angle variation in each layer. These functions are inserted in the **Rotation** field in the **Layered Material** node.

Fiber Angle (Bottom Layer)

1 In the **Home** toolbar, click **Functions** and choose **Global** > **Analytic**.

2 In the **Settings** window for **Analytic**, locate the **Definition** section.

3 In the **Expression** text field, type `2/a*(thetae1-thetac1)*abs(x) + thetac1`.

4 Locate the **Units** section. In the **Function** text field, type `rad`.

5 In the table, enter the following settings:

Argument	Unit
x	m

6 In the **Label** text field, type `Fiber Angle (Bottom Layer)`.

7 In the **Function name** text field, type `fiberAngleBot`.

Fiber Angle (Top Layer)

1 Right-click **Fiber Angle (Bottom Layer)** and choose **Duplicate**.

2 In the **Settings** window for **Analytic**, type `Fiber Angle (Top Layer)` in the **Label** text field.

3 In the **Function name** text field, type `fiberAngleTop`.

4 Locate the **Definition** section. In the **Expression** text field, type `2/a*(thetae2-thetac2)*abs(x) + thetac2`.

Layered Material 1 (lmat1)

1 In the **Model Builder** window, under **Global Definitions** > **Materials** click **Layered Material 1 (lmat1)**.

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: E-glass/epoxy [fiber volume fraction 60%] (mat1)	fiberAngleBot(X)	-	th	nelz
Layer 2	Unidirectional fiber lamina: E-glass/epoxy [fiber volume fraction 60%] (mat1)	fiberAngleTop(X)	-	th	nelz

The **Solid Mechanics** interface is used to verify the results. The spatially varying orthotropic material properties due to the presence of curvilinear fibers are obtained with the use of rotated coordinate systems.

DEFINITIONS

Rotated System (Solid, Bottom Layer)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

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

3 Find the **Euler angles** subsection. In the α text field, type `fiberAngleBot(x)`.

4 In the **Label** text field, type `Rotated System (Solid, Bottom Layer)`.

Rotated System (Solid, Top Layer)

1 Right-click **Rotated System (Solid, Bottom Layer)** and choose **Duplicate**.

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

3 Find the **Euler angles** subsection. In the α text field, type `fiberAngleTop(x)`.

4 In the **Label** text field, type `Rotated System (Solid, Top Layer)`.

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 From the **Position** list, choose **Top side on boundary**.

4 Locate the **Boundary Selection** section. Click **Clear Selection**.

5 Select Boundary 7 only.

Solid Material

1 Right-click **Materials** and choose **More Materials** > **Material Link**.

2 Click in the **Graphics** window and then press Ctrl+A to select both domains.

3 In the **Settings** window for **Material Link**, type `Solid Material` in the **Label** text field.

LAYERED SHELL (LSHELL)

Fixed Constraint 1

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

2 Select Edges 7, 8, 13, and 18 only.

Face Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Face Load**.

2 Select Boundary 7 only.

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

4 From the **Load type** list, choose **Pressure**.

5 In the p text field, type `p`.

6 Locate the **Interface Selection** section. From the **Apply to** list, choose **Top interface**.

SOLID MECHANICS (SOLID)

Use the same discretization used by the **Layered Shell** interface to have a more accurate comparison.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

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

Linear Elastic Material (Bottom Layer)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Linear Elastic Material 1**.

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

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

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System (Solid, Bottom Layer) (sys2)**.

5 In the **Label** text field, type `Linear Elastic Material (Bottom Layer)`.

Linear Elastic Material (Top Layer)

1 In the **Physics** toolbar, click **Domains** and choose **Linear Elastic Material**.

2 Select Domain 2 only.

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

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

5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System (Solid, Top Layer) (sys3)**.

6 In the **Label** text field, type `Linear Elastic Material (Top Layer)`.

Fixed Constraint 1

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

2 Select Boundaries 1, 2, 4, 5, and 8–11 only.

Boundary Load 1

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

2 Select Boundary 7 only.

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

4 From the **Load type** list, choose **Pressure**.

5 In the p text field, type `p`.

MESH 1

Mapped 1

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

2 Select Boundary 7 only.

Distribution 1

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

2 Select Edges 7 and 8 only.

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

4 In the **Number of elements** text field, type `8`.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

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

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

3 In the **Number of elements** text field, type `nelz`.

4 Click **Build All**.

STUDY 1

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, type `stres` in the text field.

5 In the tree, select **Solid Mechanics** > **Stress tensor (N/m^2)**.

6 Click **OK**.

7 In the **Settings** window for **Preferred Units**, locate the **Units** section.

8 In the table, enter the following settings:

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

9 Click **Add Physical Quantity**.

10 In the **Physical Quantity** dialog, type `disp` in the text field.

11 In the tree, select **General** > **Displacement (m)**.

12 Click **OK**.

13 In the **Settings** window for **Preferred Units**, locate the **Units** section.

14 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Displacement	m	mm

15 Select the **Apply conversions to expressions with the same dimensions** checkbox.

16 Click **Apply**.

Stress (lshell)

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

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

3 Select the **Show maximum and minimum values** checkbox.

4 Select the **Show units** checkbox.

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

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

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

Stress (solid)

1 In the **Model Builder** window, click **Stress (solid)**.

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

3 Select the **Show maximum and minimum values** checkbox.

4 Select the **Show units** checkbox.

Volume 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.

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

3 In the **Expression** text field, type `solid.slGp11`.

4 In the **Stress (solid)** toolbar, click **Plot**.

Use the **Cut Line 3D** datasets for through thickness plots.

Cut Line 3D (solid)

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

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

3 From the **Line entry method** list, choose **Point and direction**.

4 Find the **Point** subsection. In the **X** text field, type `-a/4`.

5 In the **Y** text field, type `-a/4`.

6 Find the **Direction** subsection. In the **X** text field, type `0`.

7 In the **Z** text field, type `1`.

8 In the **Label** text field, type `Cut Line 3D (solid)`.

Cut Line 3D (lshell)

1 Right-click **Cut Line 3D (solid)** and choose **Duplicate**.

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

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

4 In the **Label** text field, type `Cut Line 3D (lshell)`.

u along Cut Line

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

2 In the **Settings** window for **1D Plot Group**, type `u along Cut Line` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** checkbox. In the associated text field, type `Coordinate Z (m)`.

Line Graph 1

1 Right-click **u along Cut Line** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Cut Line 3D (solid)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `u2`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `Z`.

7 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Blue**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

9 From the **Positioning** list, choose **Interpolated**.

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

11 In the table, enter the following settings:

Legends
Solid Mechanics

12 Select the **Show legends** checkbox.

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Line 3D (lshell)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `u`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `lshell.Z`.

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

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

8 From the **Color** list, choose **Red**.

9 Find the **Line markers** subsection. Set the **Number** value to **7**.

10 Locate the **Legends** section. In the table, enter the following settings:

Legends
Layered Shell

11 In the **u along Cut Line** toolbar, click **Plot**.

Repeat the previous instructions for the other displacement components.

sxx along Cut Line

1 In the **Model Builder** window, right-click **u along Cut Line** and choose **Duplicate**.

2 In the **Model Builder** window, click **u along Cut Line 1**.

3 In the **Settings** window for **1D Plot Group**, type `sxx along Cut Line` in the **Label** text field.

4 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Line Graph 1

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

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

3 In the **Expression** text field, type `solid.sGpxx`.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

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

3 In the **Expression** text field, type `lshell.sGpxx`.

sxx along Cut Line

1 In the **Model Builder** window, click **sxx along Cut Line**.

2 In the **sxx along Cut Line** toolbar, click **Plot**.

Repeat the previous instructions for the other stress components: sGpxy, sGpxz, sGpyy, sGpyz, sGpzz, misesGp.

sxx along Cut Line, sxy along Cut Line, sxz along Cut Line, syy along Cut Line, syz along Cut Line, szz along Cut Line, u along Cut Line, v along Cut Line, von Mises along Cut Line, w along Cut Line

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **u along Cut Line**, **v along Cut Line**, **w along Cut Line**, **sxx along Cut Line**, **sxy along Cut Line**, **sxz along Cut Line**, **syy along Cut Line**, **syz along Cut Line**, **szz along Cut Line**, and **von Mises along Cut Line**.

2 Right-click and choose **Group**.

Through Thickness Plots

In the **Settings** window for **Group**, type `Through Thickness Plots` in the **Label** text field.

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/Solution 1 (sol1)** > **Layered Shell** > **Geometry and Layup (lshell)** > **First Principal Material Direction (lshell)**.

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

5 In the **Home** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

First Principal Material Direction (lshell)

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

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

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

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

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

5 In the **z-coordinate** text field, type `th/2`.

Cut Plane 2

1 Right-click **Cut Plane 1** and choose **Duplicate**.

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

3 In the **z-coordinate** text field, type `3*th/2`.

Fiber Path in Layers

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Fiber Path in Layers` in the **Label** text field.

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

4 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

Surface 1

1 Right-click **Fiber Path in Layers** and choose **Surface**.

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

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

5 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Streamline 1

1 In the **Model Builder** window, right-click **Fiber Path in Layers** and choose **Streamline**.

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

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

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

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

6 In the **x** text field, type `-a/2`.

7 In the **y** text field, type `range(-a/2,a/10,a/2)`.

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

9 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Streamline 2

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

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

3 In the **x** text field, type `a/2`.

Streamline 3

1 Right-click **Streamline 2** and choose **Duplicate**.

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

3 In the **x** text field, type `range(0,a/10,a/2-a/20)`.

4 In the **y** text field, type `a/2`.

Streamline 4

1 Right-click **Streamline 3** and choose **Duplicate**.

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

3 In the **x** text field, type `range(-a/2,a/10,0)`.

4 In the **y** text field, type `-a/2`.

Streamline 1, Streamline 2, Surface 1

1 In the **Model Builder** window, under **Results** > **Fiber Path in Layers**, Ctrl-click to select **Surface 1**, **Streamline 1**, and **Streamline 2**.

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

Surface 2

1 In the **Settings** window for **Surface**, locate the **Plot Array** section.

2 In the **Index** text field, type `1`.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Plane 2**.

Streamline 5

1 In the **Model Builder** window, click **Streamline 5**.

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

3 In the **Index** text field, type `1`.

4 Locate the **Data** section. From the **Dataset** list, choose **Cut Plane 2**.

5 Locate the **Streamline Positioning** section. In the **y** text field, type `range(-a/2,a/20, a/2)`.

Streamline 6

1 In the **Model Builder** window, click **Streamline 6**.

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

3 In the **Index** text field, type `1`.

4 Locate the **Data** section. From the **Dataset** list, choose **Cut Plane 2**.

5 Locate the **Streamline Positioning** section. In the **y** text field, type `range(-a/2,a/20, a/10)`.

Annotation 1

1 In the **Model Builder** window, right-click **Fiber Path in Layers** and choose **Annotation**.

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

3 From the **Dataset** list, choose **Cut Plane 1**.

4 Locate the **Annotation** section. In the **Text** text field, type `Bottom Layer`.

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 From the **Anchor point** list, choose **Center**.

7 Locate the **Position** section. In the **y** text field, type `-a/2-a/20`.

Annotation 2

1 Right-click **Annotation 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Plane 2**.

4 Locate the **Annotation** section. In the **Text** text field, type `Top Layer`.

5 In the **Fiber Path in Layers** toolbar, click **Plot**.