Micromechanics and Stress Analysis of a Composite Cylinder
Introduction

The use of fiber composites in the manufacturing industry is increasing. Compared to traditional metallic engineering materials, fiber composites are lighter and more corrosion resistant, and properties like strength, stiffness and toughness can often be tailored to a specific application. A fiber composite consists of load carrying fibers embedded in a polymer resin. The composite material is typically a laminate of individual layers, where the fibers in each layer are uni-directional. In this model, we perform a stress analysis of a laminated composite cylinder.

Modeling individual fibers in every layer in the laminate is unfeasible. A simplified micromechanics model of a single carbon fiber in epoxy is instead used to estimate the elastic properties of a single layer. These properties are then used in the homogenized model of the laminated composite cylinder. Two approaches are used to model the laminate, namely the Layerwise (LW) theory, and the Equivalent Single Layer (ESL) theory.

Model Definition

This model performs different types of analyses of a laminated composite cylinder. The model is divided into three parts:

- Micromechanics analysis
- Stress analysis using the Layerwise theory
- Stress analysis using the Equivalent Single Layer theory

Eigenfrequencies and mode shapes are computed and compared using both theories.

Micromechanics Analysis

A micromechanics analysis of a single layer is performed in order to obtain its homogenized material properties. The composite layer is assumed to be made of carbon fibers unidirectionally embedded in epoxy resin. A representative unit cell having a cylindrical fiber located at the middle of resin is shown in Figure 1. The fiber radius is computed assuming a fiber volume fraction of 0.6.
Figure 1: Geometry of the unit cell with a carbon fiber in an epoxy resin.

**Fiber and Resin Properties**

The layers of the laminate are made of T300 carbon fiber and 914C epoxy. The carbon fiber is assumed transversely isotropic (modeled as orthotropic), and the epoxy resin is assumed isotropic. The material properties of fiber and resin are given in Table 1 and Table 2, respectively.

**TABLE 1: CARBON FIBER MATERIAL PROPERTIES**

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( {E_1, E_2, E_3} )</td>
<td>( {230, 15, 15} ) (GPa)</td>
</tr>
<tr>
<td>( {G_{12}, G_{23}, G_{13}} )</td>
<td>( {15, 7, 15} ) (GPa)</td>
</tr>
<tr>
<td>( {\nu_{12}, \nu_{23}, \nu_{13}} )</td>
<td>( {0.2, 0.07, 0.2} )</td>
</tr>
<tr>
<td>( \rho )</td>
<td>1800 (kg/m³)</td>
</tr>
</tbody>
</table>

**TABLE 2: EPOXY RESIN MATERIAL PROPERTIES**

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E )</td>
<td>4 (GPa)</td>
</tr>
<tr>
<td>( \nu )</td>
<td>0.35</td>
</tr>
<tr>
<td>( \rho )</td>
<td>1100 (kg/m³)</td>
</tr>
</tbody>
</table>
Cell Periodicity
In order to perform a micromechanics analysis, the Cell Periodicity node in the Solid Mechanics interface is used. The Cell Periodicity node is used to apply periodic boundary conditions to the three pairs of faces of the unit cell.

In order to extract the homogenized elasticity matrix for a layer, the unit cell needs to be analyzed for six different load cases. This is automatically done by the Cell Periodicity node as it adds a stationary study with six different load groups.

With the Average Strain periodicity type chosen, you can obtain a Global Material with an elasticity matrix corresponding to that of the homogenized material. This global material can be used to define the properties of individual layers in a composite laminate.

STRESS ANALYSIS USING THE LAYERWISE THEORY

Layerwise (LW) Theory
In the Layerwise theory, the degrees of freedom are the displacements \((u, v, w)\) available on the reference surface (or modeled surface) as well as in the through-thickness direction. From a constitutive equation point of view, this theory is similar to 3D solid elasticity. The layerwise theory is useful for detailed modeling of thick composite laminates because it can capture inter-laminar shear stresses. It could therefore be used to also study delamination.

Geometry and Boundary Conditions
The model geometry of a composite cylinder with length of 0.5 m and radius of 0.1 m is shown in Figure 2. Boundary conditions and loading are:

- One end of the cylinder is fixed.
- The other end has a roller support.
- A load of 1 kN is applied to a quarter of the cylinder outer surface.
Figure 2: Geometry of the cylinder showing boundary conditions and loading.

Figure 3: Through-thickness view of the laminated material with five layers.
Stacking Sequence and Material Properties

The laminate consists of five layers of 1 mm thickness, as shown in Figure 3. The orientations of the layers are different. The orientations, starting from the bottom of the laminate, are taken as 0, 45, 90, -45, 0 degrees, as shown in Figure 4. The orientation of a layer is specified with respect to the laminate coordinate system as shown in Figure 5. The material properties for each layer are given by the homogenized material computed in the micromechanics analysis.

Figure 4: Stacking sequence [0/45/90/-45/0] for the laminate showing the fiber orientation of each layer, from bottom to top.
Figure 5: The laminate coordinate system showing the first principal direction along the cylinder axis.

STRESS ANALYSIS USING THE ESL THEORY

Equivalent Single Layer (ESL) Theory

In the equivalent single layer (ESL) theory, the degrees of freedom are the displacements and rotations on the midplane of the laminate. From a constitutive equation point of view, this theory is similar to 3D shell elasticity. Through-thickness homogenized material properties of the laminate are used. It is therefore computationally less expensive than the layerwise theory. It can be used for the modeling of thin to moderately thick laminates with good accuracy. An aim of this analysis is to compare the results to the results obtained from the Layerwise theory.

The model set-up including geometry, boundary conditions, material properties etc. is the same as described in the previous section.

Results and Discussion

In the micromechanics analysis, six load cases are used to evaluate the elasticity matrix. The distribution of effective (von Mises) stress for four of the load cases is shown in Figure 6.
The von Mises stress distribution in the composite cylinder obtained from the two theories is presented in Figure 7. Both theories give similar results.

The through-thickness variation of the axial second Piola-Kirchhoff stress at a particular point on the cylinder is shown in Figure 8. We see that the stress variation is discontinuous between layers, but that the two theories predict similar distributions.
Figure 7: Von Mises stress distribution in the composite cylinder obtained using the Layerwise and ESL theories.

Figure 8: Through-thickness stress variation in the axial direction at a particular point on the cylinder.
Figure 9: Von Mises stress distribution in the five layers of the laminate.

Figure 9 shows the distribution of von Mises stress in each layer of the laminate using the Layered Material Slice plot. The distributions of stress in the individual layers are markedly different. The middle layer, with fibers perpendicular to the first principal laminate direction, shows the lowest stresses.

The first six eigenfrequencies of the constrained cylinder are shown in Table 3, and the corresponding mode shapes are shown in Figure 10. The eigenfrequencies obtained using the LW and ESL theories match closely.

TABLE 3: COMPARISON OF EIGENFREQUENCIES

<table>
<thead>
<tr>
<th>Eigenfrequencies from layerwise theory (Hz)</th>
<th>Eigenfrequencies from ESL theory (Hz)</th>
<th>Percentage difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>485.98</td>
<td>485.31</td>
<td>0.138</td>
</tr>
<tr>
<td>572.7</td>
<td>572.05</td>
<td>0.113</td>
</tr>
<tr>
<td>983.85</td>
<td>983.54</td>
<td>0.032</td>
</tr>
<tr>
<td>998.96</td>
<td>998.4</td>
<td>0.056</td>
</tr>
<tr>
<td>1213.2</td>
<td>1210.6</td>
<td>0.214</td>
</tr>
<tr>
<td>1275.4</td>
<td>1273.4</td>
<td>0.157</td>
</tr>
</tbody>
</table>
Notes About the COMSOL Implementation

- The micromechanics analysis of a single fiber in a resin can be performed using the **Cell Periodicity** node available in the **Solid Mechanics** interface. Using this functionality, the elasticity matrix of the homogenized material can be computed for given fiber and resin properties, and the fiber volume fraction.
• Modeling a composite laminated shell requires a surface geometry (2D), in general called a base surface, and a Layered Material node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. Using the Layered Material functionality, you can model several layers of different thickness, material properties, and fiber orientations. You can optionally specify the interface materials between the layers and control mesh elements in each layer.

• You can either use the Layerwise (LW) theory based Layered Shell interface or the Equivalent Single Layer (ESL) theory based Layered Linear Elastic Material node in Shell interface. Use one of these to apply loads and constraints on different layers of a composite shell and solve for stresses and other relevant variables in each individual layer.

• To analyze the results in a composite shell, you can either create a slice plot using the Layered Material Slice plot for in-plane variation of a quantity, or you can create a Through Thickness plot for out-of-plane variation of a quantity. To visualize the results as a 3D solid object, you can use the Layered Material dataset which creates a virtual 3D solid object combining the surface geometry (2D) and the extra dimension (1D).

Application Library path: Composite_Materials_Module/Tutorials/composite_cylinder_micromechanics_and_stress_analysis

Modeling Instructions (Micromechanics)

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>Solid Mechanics (solid).
3 Click Add.
4 Click Done.

GLOBAL DEFINITIONS
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 `composite_cylinder_micromechanics_and_stress_analysis_parameters.txt`.

**GEOMETRY I**

*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 1.

4 In the **Depth** text field, type 1.

5 In the **Height** text field, type 1.

6 Click **Build Selected**.

*Cylinder 1 (cyl1)*

1 In the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type $r_f$.

4 In the **Height** text field, type 1.

5 Locate the **Position** section. In the **y** text field, type 1/2.

6 In the **z** text field, type 1/2.

7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

8 Click **Build Selected**.

*Form Union (fin)*

1 In the **Model Builder** window, under **Component 1 (compl)>Geometry 1** click **Form Union (fin)**.

2 Click **Build Selected**.

**SOLID MECHANICS (SOLID)**

*Linear Elastic Material 2*

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 **Solid model** list, choose **Orthotropic**.

**Cell Periodicity 1**

1 In the **Physics** toolbar, click **Domains** and choose **Cell Periodicity**.
2 In the **Settings** window for **Cell Periodicity**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **All domains**.
4 Locate the **Periodicity Type** section. From the list, choose **Average strain**.
5 From the **Calculate average properties** list, choose **Elasticity matrix, Standard (XX, YY, ZZ, XY, YZ, XZ)**.

**Boundary Pair 1**

1 Right-click **Cell Periodicity 1** and choose **Boundary Pair**.
2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3 Click **Clear Selection**.
4 Select Boundaries 1, 5, 11, and 12 only.

**Boundary Pair 2**

1 Right-click **Component 1 (comp1)>Solid Mechanics (solid)>Cell Periodicity 1>Boundary Pair 1** and choose **Duplicate**.
2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3 Click **Clear Selection**.
4 Select Boundaries 2 and 10 only.

**Boundary Pair 3**

1 Right-click **Component 1 (comp1)>Solid Mechanics (solid)>Cell Periodicity 1>Boundary Pair 2** and choose **Duplicate**.
2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3 Click **Clear Selection**.
4 Select Boundaries 3 and 4 only.

**Cell Periodicity 1**

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Cell Periodicity 1**.
2 In the **Settings** window for **Cell Periodicity**, locate the **Load Group, Material and Study Generation** section.
3 Click **Create**.
**Materials**

**Material 1 (mat1)**

1. In the *Model Builder* window, under *Component 1 (comp1)* right-click *Materials* and choose *Blank Material*.
2. In the *Settings* window for *Material*, type *Material 1: Epoxy Resin* in the *Label* text field.
3. Select Domain 1 only.
4. Locate the *Material Contents* section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>E</td>
<td>E_r</td>
<td>Pa</td>
<td>Basic</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>nu</td>
<td>nu_r</td>
<td>1</td>
<td>Basic</td>
</tr>
<tr>
<td>Density</td>
<td>rho</td>
<td>rho_r</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
</tbody>
</table>

**Material 2 (mat2)**

1. Right-click *Materials* and choose *Blank Material*.
2. In the *Settings* window for *Material*, type *Material 2: Carbon Fiber* in the *Label* text field.
3. Locate the *Geometric Entity Selection* section. Click *Paste Selection*.
4. In the *Paste Selection* dialog box, type 2 in the *Selection* text field.
5. Click *OK*.
6. In the *Settings* window for *Material*, locate the *Material Contents* section.
7. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>{Evector1, Evector2, Evector3}</td>
<td>{E1_f, E2_f, E2_f}</td>
<td>Pa</td>
<td>Orthotropic</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>{nuvector1, nuvector2, nuvector3}</td>
<td>{nu12_f, nu23_f, nu12_f}</td>
<td>1</td>
<td>Orthotropic</td>
</tr>
<tr>
<td>Shear modulus</td>
<td>{Gvector1, Gvector2, Gvector3}</td>
<td>{G12_f, G23_f, G12_f}</td>
<td>N/m²</td>
<td>Orthotropic</td>
</tr>
<tr>
<td>Density</td>
<td>rho</td>
<td>rho_f</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
</tbody>
</table>
**Mesh 1**

*Free Triangular 1*

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2. Select Boundaries 1 and 5 only.

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

*Swept 1*

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2. Click **Build Selected**.

**Cell Periodicity Study**

In the **Home** toolbar, click **Compute**.

**Results**

*Stress (solid)*

Use the following instructions to plot the von Mises stress in the unit cell as shown in Figure 6.

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

2. In the **Settings** window for **3D Plot Group**, type **Stress, Unit Cell** in the **Label** text field.

3. In the **Stress, Unit Cell** toolbar, click **Plot**.

*Global Matrix Evaluation 1*

1. In the **Results** toolbar, click **More Derived Values** and choose **Other>Global Matrix Evaluation**.

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

3. From the **Parameter selection (Load case)** list, choose **Last**.

4. Locate the **Expression** section. Click **solid.cp1.D - Elasticity matrix - Pa** in the upper-right corner of the section. Click **Evaluate**.

**Modeling Instructions (Stress Analysis using the Layerwise (LW) Theory)**

This section describes how to model a laminated composite cylinder using the layerwise theory based **Layered Shell** interface.
ROOT
In the Home toolbar, click Component and choose Add Component>3D.

GEOMETRY 2
In the Model Builder window, under Component 2 (comp2) click Geometry 2.

Cylinder 1 (cyl1)
1 In the Geometry toolbar, click Cylinder.
2 In the Settings window for Cylinder, locate the Object Type section.
3 From the Type list, choose Surface.
4 Locate the Size and Shape section. In the Radius text field, type rc.
5 In the Height text field, type hc.
6 Locate the Axis section. From the Axis type list, choose x-axis.
7 Click Build Selected.

DEFINITIONS (COMP2)

Boundary System 2 (sys2)
1 In the Model Builder window, expand the Component 2 (comp2)>Definitions node, then click Boundary System 2 (sys2).
2 In the Settings window for Boundary System, locate the Settings section.
3 Find the Coordinate names subsection. From the Axis list, choose x.

ADD PHYSICS
1 In the Home toolbar, click Add Physics to open the Add Physics window.
2 Go to the Add Physics window.
3 In the tree, select Structural Mechanics>Layered Shell (lshell).
4 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Cell periodicity study.
5 Click Add to Component in the window toolbar.
6 In the Home toolbar, click Add Physics to close the Add Physics 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**, type **Layered Material: [0/45/90/-45/0]** in the **Label** text field.

3 Locate the **Layer Definition** section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Layer</th>
<th>Material</th>
<th>Rotation (deg)</th>
<th>Thickness (m)</th>
<th>Mesh elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 1</td>
<td>Homogeneous Material 1 (cp1mat)</td>
<td>0</td>
<td>th</td>
<td>1</td>
</tr>
</tbody>
</table>

4 Click **Add**.

5 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Layer</th>
<th>Material</th>
<th>Rotation (deg)</th>
<th>Thickness (m)</th>
<th>Mesh elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 2</td>
<td>cp1mat</td>
<td>45</td>
<td>th</td>
<td>1</td>
</tr>
</tbody>
</table>

6 Click **Add**.

7 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Layer</th>
<th>Material</th>
<th>Rotation (deg)</th>
<th>Thickness (m)</th>
<th>Mesh elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 3</td>
<td>cp1mat</td>
<td>90</td>
<td>th</td>
<td>1</td>
</tr>
</tbody>
</table>

8 Click **Add**.

9 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Layer</th>
<th>Material</th>
<th>Rotation (deg)</th>
<th>Thickness (m)</th>
<th>Mesh elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 4</td>
<td>cp1mat</td>
<td>-45</td>
<td>th</td>
<td>1</td>
</tr>
</tbody>
</table>

10 Click **Add**.

11 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Layer</th>
<th>Material</th>
<th>Rotation (deg)</th>
<th>Thickness (m)</th>
<th>Mesh elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 5</td>
<td>cp1mat</td>
<td>0</td>
<td>th</td>
<td>1</td>
</tr>
</tbody>
</table>

12 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type **0.4**.

13 Locate the **Layer definition** section. Click **Layer Cross Section Preview** in the upper-right corner of the section.

14 Click **Layer Stack Preview** in the upper-right corner of the **Layer definition** section.
MATERIALS
In the Model Builder window, under Component 2 (comp2) right-click Materials and choose Layers>Layered Material Link.

GLOBAL DEFINITIONS

Homogeneous Material 1 (cp1mat)
1 In the Settings window for Material, locate the Material Contents section.
2 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>rho</td>
<td>rho_1</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
</tbody>
</table>

LAYERED SHELL (LSHELL)

Linear Elastic Material 1
1 In the Model Builder window, under Component 2 (comp2)>Layered Shell (lshell) click Linear Elastic Material 1.
2 In the Settings window for Linear Elastic Material, locate the Linear Elastic Material section.
3 From the Solid model list, choose Anisotropic.

Fixed Constraint 1
1 In the Physics toolbar, click Edges and choose Fixed Constraint.
2 Select Edges 1, 2, 4, and 6 only.

Roller 1
1 In the Physics toolbar, click Edges and choose Roller.
2 Select Edges 9–12 only.

Body Load 1
1 In the Physics toolbar, click Boundaries and choose Body Load.
2 Select Boundary 2 only.
3 In the Settings window for Body Load, locate the Force section.
4 From the Load type list, choose Total force.
5 Specify the \( F_{\text{tot}} \) vector as

\[
\begin{bmatrix}
0 \\
x
\end{bmatrix}
\]
MESH 2

Mapped 1
1 In the Model Builder window, under Component 2 (comp2) right-click Mesh 2 and choose More Operations>Mapped.
2 In the Settings window for Mapped, locate the Boundary Selection section.
3 From the Selection list, choose All boundaries.

Distribution 1
1 Right-click Component 2 (comp2)>Mesh 2>Mapped 1 and choose Distribution.
2 In the Settings window for Distribution, locate the Edge Selection section.
3 From the Selection list, choose All edges.
4 Locate the Distribution section. In the Number of elements text field, type 20.
5 Click Build All.

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 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Solid Mechanics (solid).
5 Click Add Study in the window toolbar.
6 In the Home toolbar, click Add Study to close the Add Study window.

STUDY 1
1 In the Settings window for Study, type Study 1: Stationary (Layerwise Theory) in the Label text field.
2 In the Home toolbar, click Compute.

RESULTS

Layered Material 1
1 In the Model Builder window, expand the Results>Data Sets 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 5.

**Cut Point 3D 1**
1 In the **Results** toolbar, click **Cut Point 3D**.
2 In the **Settings** window for **Cut Point 3D**, locate the **Data** section.
3 From the **Data set** list, choose **Study 1: Stationary (Layerwise Theory)/ Solution 1a (3) (sol1)**.
4 Locate the **Point Data** section. In the **X** text field, type $hc/2$.
5 In the **Y** text field, type $rc$.
6 In the **Z** text field, type $rc$.
7 Select the **Snap to closest boundary** check box.

Use the following instructions to plot the von Mises stress in the cylinder as shown in **Figure 7**.

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

**Annotation 1**
1 Right-click **Results>Stress (mises)** and choose **Annotation**.
2 In the **Settings** window for **Annotation**, locate the **Annotation** section.
3 In the **Text** text field, type **Layerwise Theory**.
4 Locate the **Position** section. In the **x** text field, type $-2*hc/5$.
5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Lower middle**.
6 In the **Stress (mises)** toolbar, click **Plot**.

Use the following instructions to plot the axial through-thickness stress variation at a particular point on the cylinder as shown in **Figure 8**.

**Stress, Through Thickness (lshell)**
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 (S1m11)** in the **Label** text field.
3 Locate the **Plot Settings** section. Select the **x-axis label** check box.
4 In the associated text field, type **S1m11 (MPa)**.
Through Thickness 1

1 In the Model Builder window, expand the Results>Stress, Through-Thickness (Slm11) node, then click Through Thickness 1.

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

3 From the Data set list, choose Cut Point 3D 1.

4 Locate the x-Axis Data section. In the Expression text field, type lshell.Slm11.

5 From the Unit list, choose MPa.

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

7 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Legends</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layerwise Theory</td>
</tr>
</tbody>
</table>

8 In the Stress, Through-Thickness (Slm11) toolbar, click Plot.

Use the following instructions to plot the von Mises stress in different layers of the cylinder as shown in Figure 9.

Stress, Slice (Ishell)

1 In the Model Builder window, expand the Results>Stress, Slice (Ishell) node, then click Stress, Slice (Ishell).

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

3 Locate the Plot Settings section. Clear the Plot data set edges check box.

4 Click the Zoom Extents button in the Graphics toolbar.

5 From the View list, choose New view.

6 In the Stress, Slice (mises) toolbar, click Plot.

Layered Material Slice 1

1 In the Model Builder window, under Results>Stress, Slice (mises) click Layered Material Slice 1.

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

3 In the Local z-coordinate [-1,1] text field, type -0.8.

4 Click to expand the Range section. Select the Manual color range check box.

5 In the Minimum text field, type 0.

6 In the Maximum text field, type 5e6.
7 Click to expand the Quality section. From the Resolution list, choose No refinement.

Deformation 1
1 In the Model Builder window, expand the Layered Material Slice 1 node, then click Deformation 1.
2 In the Settings window for Deformation, locate the Expression section.
3 In the X component text field, type 0.
4 In the Y component text field, type 0.
5 In the Z component text field, type 0.
6 Locate the Scale section. Select the Scale factor check box.
7 In the associated text field, type 1.
8 In the Stress, Slice (mises) toolbar, click Plot.

Layered Material Slice 2
1 In the Model Builder window, under Results>Stress, Slice (mises) right-click Layered Material Slice 1 and choose Duplicate.
2 In the Settings window for Layered Material Slice, locate the Through-Thickness Location section.
3 In the Local z-coordinate [-1,1] text field, type -0.4.
4 Click to expand the Title section. From the Title type list, choose None.
5 Click to expand the Inherit Style section. From the Plot list, choose Layered Material Slice 1.

Deformation 1
1 In the Model Builder window, expand the Layered Material Slice 2 node, then click Deformation 1.
2 In the Settings window for Deformation, locate the Expression section.
3 In the Z component text field, type 3.5*rc.
4 In the Stress, Slice (mises) toolbar, click Plot.

Layered Material Slice 3
1 In the Model Builder window, under Results>Stress, Slice (mises) right-click Layered Material Slice 2 and choose Duplicate.
2 In the Settings window for Layered Material Slice, locate the Through-Thickness Location section.
3 In the Local z-coordinate [-1,1] text field, type 0.
Deformation 1
1 In the Model Builder window, expand the Layered Material Slice 3 node, then click Deformation 1.
2 In the Settings window for Deformation, locate the Expression section.
3 In the Z component text field, type $7r_c$.
4 In the Stress, Slice (mises) toolbar, click Plot.

Layered Material Slice 4
1 In the Model Builder window, under Results>Stress, Slice (mises) right-click Layered Material Slice 3 and choose Duplicate.
2 In the Settings window for Layered Material Slice, locate the Through-Thickness Location section.
3 In the Local z-coordinate [-1,1] text field, type 0.4.

Deformation 1
1 In the Model Builder window, expand the Layered Material Slice 4 node, then click Deformation 1.
2 In the Settings window for Deformation, locate the Expression section.
3 In the X component text field, type $1.3h_c$.
4 In the Z component text field, type 0.
5 In the Stress, Slice (mises) toolbar, click Plot.

Layered Material Slice 5
1 In the Model Builder window, under Results>Stress, Slice (mises) right-click Layered Material Slice 4 and choose Duplicate.
2 In the Settings window for Layered Material Slice, locate the Through-Thickness Location section.
3 In the Local z-coordinate [-1,1] text field, type 0.8.

Deformation 1
1 In the Model Builder window, expand the Layered Material Slice 5 node, then click Deformation 1.
2 In the Settings window for Deformation, locate the Expression section.
3 In the Z component text field, type $3.5r_c$.
4 In the Stress, Slice (mises) toolbar, click Plot.
Annotation 1
1 In the Model Builder window, under Results right-click Stress, Slice (mises) and choose Annotation.
2 In the Settings window for Annotation, locate the Annotation section.
3 In the Text text field, type Layer 1.
4 Locate the Position section. In the X text field, type -hc/4.
5 Locate the Coloring and Style section. From the Anchor point list, choose Lower right.

Annotation 2
1 Right-click Results>Stress, Slice (mises)>Annotation 1 and choose Duplicate.
2 In the Settings window for Annotation, locate the Annotation section.
3 In the Text text field, type Layer 2.
4 Locate the Position section. In the Z text field, type 3.5*rc.

Annotation 3
1 Right-click Results>Stress, Slice (mises)>Annotation 2 and choose Duplicate.
2 In the Settings window for Annotation, locate the Annotation section.
3 In the Text text field, type Layer 3.
4 Locate the Position section. In the Z text field, type 7*rc.

Annotation 4
1 Right-click Results>Stress, Slice (mises)>Annotation 3 and choose Duplicate.
2 In the Settings window for Annotation, locate the Annotation section.
3 In the Text text field, type Layer 4.
4 Locate the Position section. In the X text field, type 2.5*hc.
5 In the Z text field, type 0.
6 Locate the Coloring and Style section. From the Anchor point list, choose Upper left.

Annotation 5
1 Right-click Results>Stress, Slice (mises)>Annotation 4 and choose Duplicate.
2 In the Settings window for Annotation, locate the Annotation section.
3 In the Text text field, type Layer 5.
4 Locate the Position section. In the Z text field, type 3.5*rc.
5 In the Stress, Slice (mises) toolbar, click Plot.

Use the following instructions to plot the laminate coordinate system of the cylinder.
3D Plot Group 5
1 In the Home toolbar, click Add Plot Group and choose 3D Plot Group.
2 In the Settings window for 3D Plot Group, type Laminate Coordinate System in the Label text field.
3 Locate the Data section. From the Data set list, choose Study 1: Stationary (Layerwise Theory)/Solution 1a (so1).

Laminate Coordinate System
In the Model Builder window, expand the Results>Laminate Coordinate System node.

Surface 1
1 Right-click Laminate Coordinate System and choose Surface.
2 In the Settings window for Surface, locate the Expression section.
3 In the Expression text field, type dom.
4 Click to expand the Title section. From the Title type list, choose None.
5 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
6 From the Color list, choose Gray.

Laminate Coordinate System
In the Model Builder window, under Results click Laminate Coordinate System.

Coordinate System Surface 1
1 In the Laminate Coordinate System toolbar, click More Plots and choose Coordinate System Surface.
2 In the Settings window for Coordinate System Surface, locate the Coordinate System section.
3 From the Coordinate system list, choose Boundary System 2 (sys2).
4 Locate the Coloring and Style section. In the Number of arrows text field, type 50.
5 Click the Zoom Extents button in the Graphics toolbar.
6 In the Laminate Coordinate System toolbar, click Plot.

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> Eigenfrequency.
4 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Solid Mechanics (solid).

5 Click Add Study in the window toolbar.

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

STUDY 2
In the Settings window for Study, type Study 2: Eigenfrequency (Layerwise Theory) in the Label text field.

STUDY 2: EIGENFREQUENCY (LAYERWISE THEORY)

Step 1: Eigenfrequency
1 In the Model Builder window, under Study 2: Eigenfrequency (Layerwise Theory) click Step 1: Eigenfrequency.
2 In the Settings window for Eigenfrequency, locate the Study Settings section.
3 Select the Desired number of eigenfrequencies check box.
4 In the associated text field, type 12.
5 In the Home toolbar, click Compute.

Use the following instructions to plot mode shapes and eigenfrequencies as shown in Figure 10.

RESULTS

Mode Shape (Ishell)
1 In the Model Builder window, under Results click Mode Shape (Ishell).
2 In the Settings window for 3D Plot Group, type Mode Shape (Layerwise Theory) in the Label text field.
3 Locate the Plot Settings section. From the View list, choose New view.
4 Click the Zoom Extents button in the Graphics toolbar.
5 In the Mode Shape (Layerwise Theory) toolbar, click Plot.

Modeling Instructions (Stress Analysis using the Equivalent Single Layer (ESL) Theory)

This section describes how to model a laminated composite cylinder using the equivalent single layer (ESL) theory based Shell interface.
GEOMETRY 2

Move 1 (mov1)

1. In the Geometry toolbar, click Transforms and choose Move.
2. Select the object cyl1 only.
3. In the Settings window for Move, locate the Input section.
4. Select the Keep input objects check box.
5. Locate the Displacement section. In the y text field, type hc.
6. Click Build Selected.

LAYERED SHELL (LSHELL)

1. In the Model Builder window, under Component 2 (comp2) click Layered Shell (lshell).
2. In the Settings window for Layered Shell, locate the Boundary Selection section.
3. In the list, choose 5, 6, 7, and 8.
4. Click Remove from Selection.
5. Select Boundaries 1–4 only.

ADD PHYSICS

1. In the Home toolbar, click Add Physics to open the Add Physics window.
2. Go to the Add Physics window.
3. In the tree, select Structural Mechanics>Shell (shell).
4. Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Cell periodicity study, Study 1: Stationary (Layerwise Theory) and Study 2: Eigenfrequency (Layerwise Theory).
5. Click Add to Component in the window toolbar.
6. In the Home toolbar, click Add Physics to close the Add Physics window.

SHELL (SHELL)

1. In the Model Builder window’s toolbar, click the Show button and select Advanced Physics Options in the menu.
2. In the Settings window for Shell, locate the Boundary Selection section.
3. In the list, choose 1, 2, 3, and 4.
4. Click Remove from Selection.
5. Select Boundaries 5–8 only.
6 Click to expand the Advanced Settings section. Clear the Use MITC interpolation check box.

Layered Linear Elastic Material
1 In the Physics toolbar, click Boundaries and choose Layered Linear Elastic Material.
2 In the Settings window for Layered Linear Elastic Material, locate the Boundary Selection section.
3 From the Selection list, choose All boundaries.
4 Locate the Linear Elastic Material section. From the Solid model list, choose Anisotropic.

Fixed Constraint
1 In the Physics toolbar, click Edges and choose Fixed Constraint.
2 Select Edges 9, 10, 12, and 14 only.

Symmetry
1 In the Physics toolbar, click Edges and choose Symmetry.
2 Select Edges 21–24 only.

Face Load
1 In the Physics toolbar, click Boundaries and choose Face Load.
2 Click the Go to Default View button in the Graphics toolbar.
3 Select Boundary 6 only.
4 In the Settings window for Face Load, locate the Force section.
5 From the Load type list, choose Total force.
6 Specify the $F_{tot}$ vector as

<table>
<thead>
<tr>
<th>0</th>
<th>x</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>y</td>
</tr>
<tr>
<td>$F_{tot}$</td>
<td>z</td>
</tr>
</tbody>
</table>

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 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Solid Mechanics (solid) and Layered Shell (lshell).
5 Click Add Study in the window toolbar.
In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

**STUDY 3**

1. In the **Settings** window for **Study**, type **Study 3: Stationary (ESL Theory)** in the **Label** text field.
2. Locate the **Study Settings** section. Clear the **Generate default plots** check box.
3. In the **Home** toolbar, click **Compute**.

**RESULTS**

**Layered Material 3**

1. In the **Model Builder** window, under **Results>Data Sets** right-click **Layered Material 1** and choose **Duplicate**.
2. In the **Settings** window for **Layered Material**, locate the **Data** section.
3. From the **Data set** list, choose **Study 3: Stationary (ESL Theory)/Solution 3 (7) (sol3)**.

**Cut Point 3D 2**

1. In the **Model Builder** window, under **Results>Data Sets** right-click **Cut Point 3D 1** and choose **Duplicate**.
2. In the **Settings** window for **Cut Point 3D**, locate the **Point Data** section.
3. In the **Y** text field, type **hc+rc**.
4. Locate the **Data** section. From the **Data set** list, choose **Study 3: Stationary (ESL Theory)/Solution 3 (7) (sol3)**.

Use the following instructions to plot the von Mises stress obtained using the ESL theory as shown in Figure 7.

**Stress (mises)**

1. In the **Model Builder** window, under **Results** click **Stress (mises)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Data set** list, choose **Layered Material 3**.

**Surface 1**

1. In the **Model Builder** window, under **Results>Stress (mises)** click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Data** section.
3. From the **Data set** list, choose **Layered Material 1**.

**Surface 2**

1. Right-click **Results>Stress (mises)>Surface 1** and choose **Duplicate**.
2 In the Settings window for Surface, locate the Expression section.

3 In the Expression text field, type shell.mises.

4 Locate the Data section. From the Data set list, choose Layered Material 3.

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

6 Click to expand the Inherit Style section. From the Plot list, choose Surface 1.

Deformation 1
1 In the Model Builder window, expand the Surface 2 node, then click Deformation 1.

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

3 In the x component text field, type u3.

4 In the y component text field, type v3.

5 In the z component text field, type w3.

Annotation 2
1 In the Model Builder window, under Results>Stress (mises) right-click Annotation 1 and choose Duplicate.

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

3 In the Text text field, type ESL Theory.

4 Locate the Position section. In the y text field, type hc.

5 In the Stress (mises) toolbar, click Plot.

Use the following instructions to plot the through-thickness stress variation as shown in Figure 8.

Through Thickness 2
1 In the Model Builder window, under Results>Stress, Through-Thickness (S11) right-click Through Thickness 1 and choose Duplicate.

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

3 From the Data set list, choose Cut Point 3D 2.

4 Locate the x-Axis Data section. In the Expression text field, type shell.S11.

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

6 Click to expand the Coloring and Style section. Find the Line style subsection. From the Line list, choose Dashed.
7 Locate the Legends section. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Legends</th>
</tr>
</thead>
<tbody>
<tr>
<td>ESL Theory</td>
</tr>
</tbody>
</table>

8 In the Stress, Through-Thickness (Slm11) toolbar, click Plot.

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> Eigenfrequency.
4 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Solid Mechanics (solid) and Layered Shell (lshell).
5 Click Add Study in the window toolbar.
6 In the Home toolbar, click Add Study to close the Add Study window.

STUDY 4
1 In the Settings window for Study, type Study 4: Eigenfrequency (ESL Theory) in the Label text field.
2 Locate the Study Settings section. Clear the Generate default plots check box.

STUDY 4: EIGENFREQUENCY (ESL THEORY)

Step 1: Eigenfrequency
1 In the Model Builder window, under Study 4: Eigenfrequency (ESL Theory) click Step 1: Eigenfrequency.
2 In the Settings window for Eigenfrequency, locate the Study Settings section.
3 Select the Desired number of eigenfrequencies check box.
4 In the associated text field, type 12.
5 In the Home toolbar, click Compute.

RESULTS

Layered Material 4
1 In the Model Builder window, under Results>Data Sets right-click Layered Material 2 and choose Duplicate.
2 In the Settings window for Layered Material, locate the Data section.
3 From the Data set list, choose Study 4: Eigenfrequency (ESL Theory)/Solution 4 (9) (sol4).

4 Right-click Results>Data Sets>Layered Material 4 and choose Selection.

Selection
1 In the Model Builder window, under Results>Data Sets>Layered Material 4 click Selection.
2 In the Settings window for Selection, locate the Geometric Entity Selection section.
3 From the Geometric entity level list, choose Boundary.
4 Select Boundaries 5–8 only.

Use the following instructions to plot mode shapes and eigenfrequencies obtained using the ESL theory.

Mode Shape (Layerwise Theory)
1 In the Model Builder window, under Results right-click Mode Shape (Layerwise Theory) and choose Duplicate.
2 In the Settings window for 3D Plot Group, type Mode Shape (ESL Theory) in the Label text field.
3 Locate the Data section. From the Data set list, choose None.

Surface
1 In the Model Builder window, expand the Results>Mode Shape (ESL Theory) node, then click Surface 1.
2 In the Settings window for Surface, locate the Expression section.
3 In the Expression text field, type shell.disp.

Deformation
1 In the Model Builder window, expand the Surface 1 node, then click Deformation 1.
2 In the Settings window for Deformation, locate the Expression section.
3 In the x component text field, type u3.
4 In the y component text field, type v3.
5 In the z component text field, type w3.

Mode Shape (ESL Theory)
1 In the Model Builder window, under Results click Mode Shape (ESL Theory).
2 In the Settings window for 3D Plot Group, locate the Data section.
3 From the Data set list, choose Layered Material 4.
4 From the Eigenfrequency (Hz) list, choose 1273.8.
5 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

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

7 In the **Mode Shape (ESL Theory)** toolbar, click **Plot**.