Thermal Expansion of a Laminated Composite Shell
Introduction

Composite materials are often used in structural applications, where the ability to tailor properties such as stiffness and strength make them attractive compared to traditional engineering materials. In addition to structural applications, composites are also used in applications where both thermal and structural properties are important. An example is silicon wafers used in the electronics industry. Consequently, coupled thermal-structural analyses of thin structures is becoming increasingly important from a simulation standpoint.

In this example, a laminated composite shell subjected to a deposited beam power heat source is analyzed from thermal and structural points of view. The layerwise theory based approach is used to model the structural part of the shell.

The effect of the position of a heat source on the stress and deformation profiles is studied. The example also demonstrates the computation of homogenized thermal expansion coefficients of individual laminae.

In COMSOL Multiphysics, a structural analysis of a layered material can be carried out using the Layered Shell interface available in the Composite Materials Module. The thermal analysis of a layered material can be carried out using the Heat Transfer in Shells interface available in the Heat Transfer Module.

Model Definition

The geometry of the laminated composite shell consists of six H shaped flat layers stacked on top of each other. The section height is 25 cm, the web thickness is 15 cm, the flange width is 25 cm, and the flange thickness is 5 cm. The geometry of the laminate is shown in Figure 1.

Stacking Sequence

Each layer of the composite shell has a thickness of 0.125 mm as shown in Figure 3. The laminate has a [30/-45/75/-75/45/-30] stacking sequence as shown in Figure 2. This stacking sequence is antisymmetric with respect to the mid-plane of the laminate.
Figure 1: Geometry of the laminated composite shell.

Figure 2: Stacking sequence [30/-45/75/-75/45/30], showing the fiber orientation in each layer from bottom to top.
Figure 3: Through thickness view of the laminated composite shell showing the thickness (0.125 mm) of each layer.

**MATERIAL PROPERTIES**

All the layers (laminae) of the laminated composite shell are made of carbon fibers in an epoxy resin.

The homogenized orthotropic elastic material properties (the elasticity matrix) are given in Table 1. Note that only non-zero elements of the elasticity matrix are presented.

<table>
<thead>
<tr>
<th>Elasticity Matrix</th>
<th>Value (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>{D_{11}, D_{12}, D_{13}, D_{22}, D_{23}, D_{33}, D_{44}, D_{55}, D_{66}}</td>
<td>{141.34, 3.35, 3.35, 10.25, 2.83, 10.25, 4.52, 2.95, 4.52}</td>
</tr>
</tbody>
</table>

The homogenized orthotropic thermal properties of a lamina are given in Table 2.

<table>
<thead>
<tr>
<th>Thermal Conductivity</th>
<th>Value (W/(m-K))</th>
</tr>
</thead>
<tbody>
<tr>
<td>{k_{11}, k_{22}, k_{33}}</td>
<td>{6.2, 0.5, 0.5}</td>
</tr>
</tbody>
</table>

As the analysis is stationary, the values of density and heat capacity at constant pressure for a lamina do not affect the results, and are set to unity.
All elastic and thermal material properties are given in the lamina coordinate system (local material directions of a layer), where the first axis is aligned with the fiber orientation.

**Coefficient of Thermal Expansion**

The homogenized value of the coefficient of thermal expansion of a lamina for given fiber and matrix material properties is computed using a rule of mixture. The constituent material properties needed to determine the lamina thermal expansion coefficient are listed in Table 3.

**TABLE 3: Material Properties of Fiber and Matrix**

<table>
<thead>
<tr>
<th>Material Properties</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( V_f )</td>
<td>0.6</td>
<td>Fiber volume fraction</td>
</tr>
<tr>
<td>( V_m )</td>
<td>0.4</td>
<td>Matrix volume fraction</td>
</tr>
<tr>
<td>( E_{1f} )</td>
<td>230 [GPa]</td>
<td>Fiber Young's modulus in fiber direction</td>
</tr>
<tr>
<td>( E_m )</td>
<td>4 [GPa]</td>
<td>Matrix Young's modulus</td>
</tr>
<tr>
<td>( \nu_{12f} )</td>
<td>0.2</td>
<td>Fiber Poisson's ratio</td>
</tr>
<tr>
<td>( \nu_m )</td>
<td>0.35</td>
<td>Matrix Poisson's ratio</td>
</tr>
<tr>
<td>( \alpha_{1f} )</td>
<td>( -0.6 \times 10^{-6} [1/K] )</td>
<td>Fiber thermal expansion coefficient in fiber direction</td>
</tr>
<tr>
<td>( \alpha_{2f} )</td>
<td>( 8.5 \times 10^{-6} [1/K] )</td>
<td>Fiber thermal expansion coefficient perpendicular to fiber direction</td>
</tr>
<tr>
<td>( \alpha_m )</td>
<td>( 55 \times 10^{-6} [1/K] )</td>
<td>Matrix thermal expansion coefficient</td>
</tr>
</tbody>
</table>

Based on the material properties given in Table 3, the coefficients of thermal expansion for a lamina in the fiber direction as well as perpendicular to the fiber direction are calculated from the rule of mixture as below (Ref. 1):

\[
\alpha_{11} = \frac{V_f \alpha_{1f} E_{1f} + V_m \alpha_m E_m}{V_f E_{1f} + V_m E_m} \quad (1)
\]

\[
\nu_{12} = \nu_{12f} V_f + \nu_m V_m \quad (2)
\]

\[
\alpha_{22} = \alpha_{33} = (1 + \nu_m) V_m \alpha_m + \left( 1 + \nu_{12f} \frac{\alpha_{1f}}{\alpha_{2f}} \right) V_f \alpha_{2f} - \nu_{12} \alpha_{11} \quad (3)
\]
The values of the lamina thermal expansion coefficients computed using these expressions are given in Table 4. Note that the coefficient of thermal expansion in the fiber direction is three orders of magnitude smaller than the one perpendicular to the fiber direction. This is because the carbon fibers have a negative coefficient of thermal expansion in the fiber direction.

**TABLE 4: LAMINA THERMAL EXPANSION COEFFICIENTS**

<table>
<thead>
<tr>
<th>Thermal Expansion Coefficient</th>
<th>Value (1/K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>({\alpha_{11}, \alpha_{22}, \alpha_{33}})</td>
<td>({3.72E-8, 3.47E-5, 3.47E-5})</td>
</tr>
</tbody>
</table>

**BOUNDARY CONDITIONS AND LOADS**

The following boundary conditions and loads are applied to the model:

- **Structural boundary conditions:** The edges at \(X=0\) and \(X=25\) cm are fixed.

- **Thermal boundary conditions:** The temperature is set to room temperature at the edges at \(X=0\) and \(X=25\) cm. A convective heat flux with a heat transfer coefficient of 20 W/(m\(^2\)K) is applied on the bottom surface of the laminate (an exterior interface of the bottom layer).

- **Thermal loads:** A deposited beam power of 10 W is applied on the top surface of the laminate (exterior interface of the top layer). The \(x\) and \(z\)-position of the beam source is fixed in space at 12.5 cm and 25 cm whereas the \(y\)-position of the beam is varied from 0 to 25 cm. The standard deviation of the beam is taken as 1/10 of its height (or \(z\)-position) which is 2.5 cm.

**Results and Discussion**

The temperature profile in the composite shell when the beam power heat source is above its center is shown in Figure 4. The maximum temperature is observed just at the center of the shell and it is distributed along all the directions away from the center. The temperature distribution can also be seen by creating line plots along \(X\) and \(Y\) directions as shown in the model.
Figure 4: Temperature profile at yp = 12.5 cm.

The effects of the material orthotropy and layer orientations are evident in the thermal stresses and deformations pattern as shown in Figure 5. The overall thermal stress pattern is similar to the temperature profile shown in Figure 4, as the shell is subjected only to thermal loads. An interesting deformation pattern caused by the orthotropy and layer orientations can however be observed.

To see the effect of layer orientation on the von Mises stress distribution, a Layered Material Slice plot is generated at the mid-plane of the laminated composite shell, as shown in Figure 6. It can be seen that it has different stress distribution as well as magnitude when compared to the Figure 5 in which stress distribution is shown for the top layer.
Figure 5: von Mises stress distribution in laminated composite shell at $y_p = 0.125$ m.

Figure 6: von Mises stress distribution at the mid-plane of laminated composite shell at $y_p = 0.125$ m.
Figure 7 shows the through-thickness variation of the von Mises stress at four different locations in the shell. The discontinuity of the stress across the layers can be seen in the plot. Also note that there is a rotational symmetry of stresses between the points which are 180° apart when seen from the center.

Figure 7: Through-thickness variation of von Mises stress at selected points when yp = 0.125 m.

Figure 8, Figure 9, Figure 10 and Figure 11 show the distribution of von Mises stress and different components of the stress tensor in the laminate coordinate system. The stresses are plotted at the mid-plane of each layer. The effect of the antisymmetric layup is can be seen in Figure 8, Figure 9, and Figure 10. For example, the stress patterns in Layer 1 and Layer 6 are similar but antisymmetric about the mid-plane of the laminate.

Figure 11 shows the shear stress distribution and also has the antisymmetric pattern among the top and bottom layer. Also, the sign of the stress is reversed when compared between top and bottom layers because of the antisymmetry.
Figure 8: von Mises stress in laminate coordinate system at the mid-plane of each layer when \( y_p = 0.125 \text{ m} \).

Figure 9: Stress component 11 (fiber direction) at the mid-plane of each layer when \( y_p = 0.125 \text{ m} \).
Figure 10: Stress component $22$ (transverse to fiber direction) at the mid-plane of each layer when $y_p = 0.125$ m.

Figure 11: Stress component $12$ (in-plane shear) at the mid-plane of each layer when $y_p = 0.125$ m.
Notes About the COMSOL Implementation

- Modeling a composite laminated shell requires a surface geometry (2D), 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 many layers stacked on top of each other, having different thickness, material properties, and fiber orientations. You can also optionally specify the interface materials between the layers and control mesh elements through each layer.

- From a structural analysis point of view, 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 the Shell interface for modeling layered shells.

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

Reference


Application Library path: Composite_Materials_Module/Multiphysics/thermal_expansion_of_a_laminated_composite_shell

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 Heat Transfer>Thin Structures>
   Heat Transfer in Shells (htlsh).
3 Click Add.
4 In the Select Physics tree, select Structural Mechanics>Layered Shell (lshell).
5 Click Add.
6 Click Study.
7 In the Select Study tree, select General Studies>Stationary.
8 Click Done.

GLOBAL DEFINITIONS
Load the material properties and general parameters from a file.
1 In the Model Builder window, under Global Definitions click Parameters 1.
2 In the Settings window for Parameters, type Parameters: General in the Label text field.
3 Locate the Parameters section. Click Load from File.
4 Browse to the model’s Application Libraries folder and double-click the file
   thermal_expansion_of_a_laminatedCompositeShell_parameters_general.txt.

Load the thermal expansion parameters from a file in a separate Parameters node.

Parameters 2
1 In the Home toolbar, click Parameters and choose Add>Parameters.
2 In the Settings window for Parameters, type Parameters: Thermal Expansion in the Label text field.
3 Locate the Parameters section. Click Load from File.
4 Browse to the model’s Application Libraries folder and double-click the file
   thermal_expansion_of_a_laminatedCompositeShell_parameters_thermal_expansion.txt.

Material 1 (mat1)
1 In the Model Builder window, under Global Definitions right-click Materials and choose Blank Material.
2 In the Settings window for Material, type Material: Carbon-Epoxy in the Label text field.

Now add a Layered Material node and load the thickness and rotation angles of each layer from a file.

Layered Material 1 (lmat1)
1 Right-click Materials and choose Layered Material.
2 In the Settings window for Layered Material, type Layered Material: [30/-45/75] as in the Label text field.
3 Locate the Layer definition section. Click Load Layers from File in the upper-right corner of the section. Browse to model’s Application Libraries folder and double-click the file thermal_expansion_of_a_laminated_composite_shell_layers.txt.
4 Click Layer Cross Section Preview in the upper-right corner of the Layer definition section. Click to expand the Preview Plot Settings section. In the Distance between the orientation lines text field, type 0.15.
5 In the Thickness-to-width ratio text field, type 0.6.
6 Click the Show Grid button in the Graphics toolbar.
7 Locate the Layer definition section. Click Layer Cross Section Preview in the upper-right corner of the section. Click Layer Stack Preview in the upper-right corner of the Layer definition section.

GEOMETRY 1

Work Plane 1 (wp1)
1 In the Geometry toolbar, click Work Plane.
2 Browse to the model’s Application Libraries folder and double-click the file thermal_expansion_of_a_laminated_composite_shell_layers.txt.

Work Plane 1 (wp1)>Square 1 (sq1)
1 In the Work Plane toolbar, click Primitives and choose 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)>Rectangle 1 (rl)
1 In the Work Plane toolbar, click Primitives and choose Rectangle.
2 In the Settings window for Rectangle, locate the Size and Shape section.
3 In the Width text field, type 0.2*a.
4 In the **Height** text field, type 0.6*a.
5 Locate the **Position** section. In the **yw** text field, type 0.2*a.

**Work Plane 1 (wp1)>Copy 1 (copy1)**
1 In the **Work Plane** toolbar, click **Transforms** and choose **Copy**.
2 Select the object **r1** only.
3 In the **Settings** window for **Copy**, locate the **Displacement** section.
4 In the **xw** text field, type 0.8*a.

**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 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
5 Select the objects **copy1** and **r1** only.
6 In the **Work Plane** 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**.
   The geometry is in X-Y plane in which fibers are oriented with respect to X direction. Hence set the first axis of the laminate coordinate system in X direction.
2 In the **Settings** window for **Layered Material Link**, locate the **Orientation and Position** section.
3 Click **Go to Source**.

**DEFINITIONS (COMPI)**

**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)

Linear Elastic Material 1

1. In the Model Builder window, under Component 1 (comp1)>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.

MULTIPHYSICS

Thermal Expansion, Layered Shell 1 (tel1)

1. In the Physics toolbar, click Multiphysics Couplings and choose Boundary> Thermal Expansion, Layered Shell.

2. In the Settings window for Thermal Expansion, Layered Shell, locate the Heat Sources section.

3. Clear the Thermoelastic damping check box.

GLOBAL DEFINITIONS

Material: Carbon-Epoxy (mat1)

1. In the Model Builder window, under Global Definitions>Materials click Material: Carbon-Epoxy (mat1).

2. In the Settings window for Material, locate the Material Contents section.

3. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal conductivity</td>
<td>{k11, k22, k33}</td>
<td>{k1, k2}</td>
<td>W/(m·K)</td>
<td>Basic</td>
</tr>
<tr>
<td>Density</td>
<td>rho</td>
<td>1</td>
<td>kg/m³</td>
<td>Basic</td>
</tr>
<tr>
<td>Heat capacity at constant pressure</td>
<td>Cp</td>
<td>1</td>
<td>J/(kg·K)</td>
<td>Basic</td>
</tr>
</tbody>
</table>
The default discretization in **Layered Shell** interface is quadratic, which makes the discretization of strains linear. The temperature field from heat transfer interface contributes to the thermal strains, so set the discretization of **Heat Transfer in Shells** interface to linear for consistency.

### **HEAT TRANSFER IN SHELLS (HTLSh)**

In the **Physics** toolbar, click **Layered Shell (Ishell)** and choose **Heat Transfer in Shells (htlsh)**.

1. In the **Model Builder** window, under **Component 1 (compl)** click **Heat Transfer in Shells (htlsh)**.
2. In the **Settings** window for **Heat Transfer in Shells**, locate the **Discretization** section.
3. From the **Temperature** list, choose **Linear**.

Use deposited beam as a heat source through a **Deposited Beam Power, Interface** feature. Select beam origin point and orientation appropriately.

**Deposited Beam Power, Interface 1**

1. In the **Physics** toolbar, click **Boundaries** and choose **Deposited Beam Power, Interface**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Deposited Beam Power, Interface**, locate the **Interface Selection** section.
4. From the **Layered material** list, choose **Layered Material Link 1 (Ilmat1)-[Imat1]**.
5 Specify the Selection vector as

Layer 1 down

6 Specify the e vector as

0  x
0  y
-1  z

7 Locate the Beam Profile section. In the $P_0$ text field, type $P_0$.

8 Specify the O vector as

  a/2   x
  yp   y
  a   z

9 In the σ text field, type a/10.

Heat Flux, Interface 1
1 In the Physics toolbar, click Boundaries and choose Heat Flux, Interface.
2 Select Boundary 1 only.
3 In the Settings window for Heat Flux, Interface, locate the Interface Selection section.
4 From the Layered material list, choose Layered Material Link 1 (Imat1)-[Imat1].
5 Specify the Selection vector as

Layer 6 up

6 Locate the Heat Flux section. Click the Convective heat flux button.
7 In the h text field, type ht.

Temperature 1
1 In the Physics toolbar, click Edges and choose Temperature.
2 Select Edges 1, 4, 11, and 12 only.

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

Fixed Constraint 1
1 In the Physics toolbar, click Edges and choose Fixed Constraint.
2 Select Edges 1, 4, 11, and 12 only.

3 In the Model Builder window, click Layered Shell (lshell).

4 In the Settings window for Layered Shell, click to expand the Default Through-Thickness Result Location section.

5 In the z text field, type 0.

MESH 1

1 In the Model Builder window, under Component 1 (comp1) click Mesh 1.

2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.

3 From the Element size list, choose Extra fine.

STUDY 1

Step 1: Stationary

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

2 In the Settings window for Stationary, click to expand the Study Extensions section.

3 Select the Auxiliary sweep check box.

4 Click Add.

5 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Parameter value list</th>
<th>Parameter unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>yp (y-position of beam)</td>
<td>range(0,0.1*a,a)</td>
<td>m</td>
</tr>
</tbody>
</table>

6 In the Home toolbar, click Compute.

Increase the thickness scale in Layered Material data sets to 10 times in order to have better visualization.

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 10.

Layered Material 2

1 In the Model Builder window, under Results>Data Sets click Layered Material 2.
2 In the **Settings** window for **Layered Material**, locate the **Layers** section.

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

**Temperature (htlsh)**

1 In the **Model Builder** window, under **Results** click **Temperature (htlsh)**.

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

3 From the **Parameter value (yp (m))** list, choose **0.125**.

4 In the **Temperature (htlsh)** toolbar, click **Plot**.

**Stress (Ishell)**

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

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

3 From the **Parameter value (yp (m))** list, choose **0.125**.

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

**Stress, Slice (Ishell)**

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

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

3 From the **Parameter value (yp (m))** list, choose **0.125**.

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

**Stress, Through Thickness (Ishell)**

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

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

3 From the **Parameter selection (yp)** list, choose **From list**.

4 In the **Parameter values (yp (m))** list, select **0.125**.

**Through Thickness 1**

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

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

3 Click **Clear Selection**.

4 Select Points 5–8 only.

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.
6 In the Stress, Through Thickness (Ishell) toolbar, click Plot.

Cut Line 3D 1
1 In the Results toolbar, click Cut Line 3D.
2 In the Settings window for Cut Line 3D, locate the Line Data section.
3 In row Point 1, set Y to a/2.
4 In row Point 2, set X to a.
5 In row Point 2, set Y to a/2.

Cut Line 3D 2
1 Right-click Cut Line 3D 1 and choose Duplicate.
2 In the Settings window for Cut Line 3D, locate the Line Data section.
3 In row Point 1, set X to a/2.
4 In row Point 1, set Y to 0.
5 In row Point 2, set X to a/2.
6 In row Point 2, set Y to a.

1D Plot Group 5
1 In the Results toolbar, click 1D Plot Group.
2 In the Settings window for 1D Plot Group, type Temperature Distribution along X Axis in the Label text field.
3 Click to expand the Title section. From the Title type list, choose Manual.
4 In the Title text area, type Line Graph: Temperature Distribution for Different Beam Location.

Line Graph 1
1 Right-click Temperature Distribution along X Axis and choose Line Graph.
2 In the Settings window for Line Graph, locate the Data section.
3 From the Data set list, choose Cut Line 3D 1.
4 Locate the x-Axis Data section. From the Parameter list, choose Expression.
5 In the Expression text field, type X.
6 Click to expand the Coloring and Style section. Find the Line style subsection. From the Line list, choose Cycle.
7 Click to expand the Legends section. Select the Show legends check box.
8 In the Temperature Distribution along X Axis toolbar, click Plot.
Temperature Distribution along X Axis

1 In the Model Builder window, under Results right-click Temperature Distribution along X Axis and choose Duplicate.

2 In the Settings window for 1D Plot Group, type Temperature Distribution along Y Axis in the Label text field.

Line Graph

1 In the Model Builder window, expand the Results>Temperature Distribution along Y Axis node, then click Line Graph 1.

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

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

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

5 In the Temperature Distribution along Y Axis toolbar, click Plot.

3D Plot Group

1 In the Home toolbar, click Add Plot Group and choose 3D Plot Group.

2 In the Settings window for 3D Plot Group, type Stress: von Mises in the Label text field.

3 Locate the Data section. From the Parameter value (yp (m)) list, choose 0.125.

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

5 In the Title text area, type Layered Material Slice: von Mises stress (MPa).

6 Locate the Plot Settings section. From the View list, choose View 3D 4.

Layered Material Slice

1 In the Stress: von Mises toolbar, click More Plots and choose Layered Material Slice.

2 In the Settings window for Layered Material Slice, locate the Expression section.

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

4 Locate the Through-Thickness Location section. From the Location definition list, choose Physical.

5 Locate the Expression section. From the Unit list, choose MPa.

6 Locate the Through-Thickness Location section. In the Local z-coordinate text field, type 0.5*th.

7 Locate the Coloring and Style section. From the Color table list, choose RainbowLight.

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

9 In the Maximum text field, type 25.
Deformation 1
1 Right-click Layered Material Slice 1 and choose Deformation.
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.

Layered Material Slice 2
1 In the Model Builder window, under Results>Stress: von 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 text field, type 1.5*th.
4 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 X component text field, type 1.2*a.

Layered Material Slice 3
1 In the Model Builder window, under Results>Stress: von 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 text field, type 2.5*th.

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 X component text field, type 2.4*a.
Layered Material Slice 4
1 In the Model Builder window, under Results>Stress: von 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 text field, type 3.5*th.

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 0.
4 In the Y component text field, type 1.2*a.

Layered Material Slice 5
1 In the Model Builder window, under Results>Stress: von 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 text field, type 4.5*th.

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 X component text field, type 1.2*a.

Layered Material Slice 6
1 In the Model Builder window, under Results>Stress: von Mises right-click Layered Material Slice 5 and choose Duplicate.
2 In the Settings window for Layered Material Slice, locate the Through-Thickness Location section.
3 In the Local z-coordinate text field, type 5.5*th.

Deformation 1
1 In the Model Builder window, expand the Layered Material Slice 6 node, then click Deformation 1.
2 In the Settings window for Deformation, locate the Expression section.
3 In the X component text field, type $2.4*a$.

Annotation 1
1 In the Model Builder window, under Results right-click Stress: von 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 $0.5*a$.
5 In the Y text field, type $-0.3*a$.

Annotation 2
1 Right-click Results>Stress: von 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 X text field, type $0.5*a+1.2*a$.

Annotation 3
1 Right-click Results>Stress: von 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 X text field, type $0.5*a+2.4*a$.

Annotation 4
1 Right-click Results>Stress: von 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 $0.5*a$.
5 In the Y text field, type $2.7*a$.

Annotation 5
1 Right-click Results>Stress: von 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 X text field, type $0.5*a+1.2*a$.

Annotation 6
1 Right-click Results>Stress: von Mises>Annotation 5 and choose Duplicate.
2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type **Layer 6**.

4 Locate the **Position** section. In the **X** text field, type 0.5*a+2.4*a.

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

6 In the **Stress: von Mises** toolbar, click **Plot**.

   In order to plot different normal and shear components of stress tensor in the laminate coordinate system at the mid-plane of each layer, duplicate the previous plot and change the plot expressions to `lshell.Sml11`, `lshell.Sml22`, and `lshell.Sml12` respectively.

   Animation can be created in **Export** node to visualize the temperature and stress profiles as the deposited beam heat source moves in Y direction.