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.

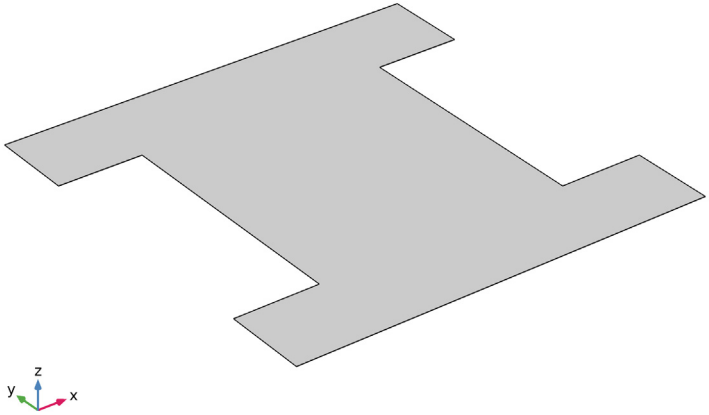

Figure 1: Geometry of the laminated composite shell.

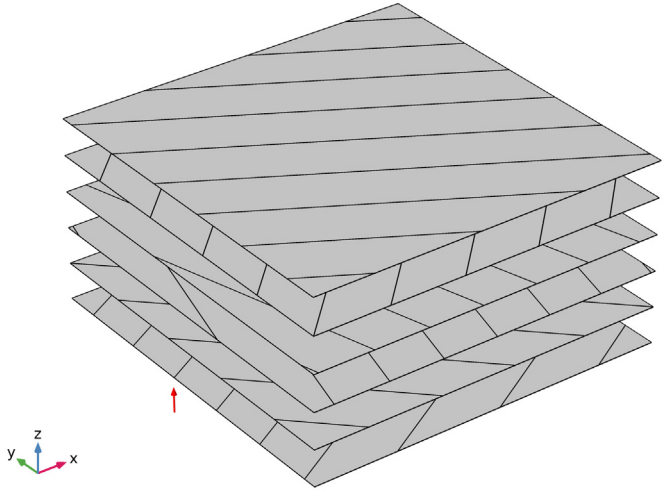

Figure 2: Stacking sequence [30/-45/75/-75/45/30], showing the fiber orientation in each layer from bottom to top.

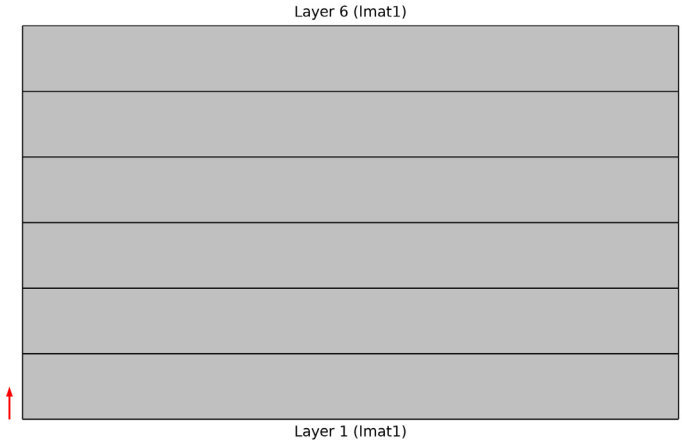

Figure 3: Through thickness view of the laminated composite shell showing the thickness (0.125 mm) of each layer.

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 1: LAMINA ELASTICITY MATRIX

Elasticity Matrix	Value (GPa)
$\{D_{11},D_{12},D_{13},D_{22},D_{23},D_{33},D_{44},D_{55},D_{66}\}$	{141.34,3.35,3.35,10.25,2.83, 10.25,4.52,2.95,4.52}

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

TABLE 2: LAMINA THERMAL CONDUCTIVITY

Thermal Conductivity	Value (W/(m-K))
$\{k_{11},k_{22},k_{33}\}$	{6.2,0.5,0.5}

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

Material Properties	Value	Description
V_f	0.6	Fiber volume fraction
V_m	0.4	Matrix volume fraction
E_{1f}	230[GPa]	Fiber Young's modulus in fiber direction
E_m	4[GPa]	Matrix Young's modulus
ν_{12f}	0.2	Fiber Poisson's ratio
ν_m	0.35	Matrix Poisson's ratio
α_{1f}	-0.6E-6[1/K]	Fiber thermal expansion coefficient in fiber direction
α_{2f}	8.5E-6[1/K]	Fiber thermal expansion coefficient perpendicular to fiber direction
α_m	55E-6[1/K]	Matrix thermal expansion coefficient

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):

$$\alpha_{11} = \frac{V_f \alpha_{1f} E_{1f} + V_m \alpha_m E_m}{V_f E_{1f} + V_m E_m} \tag{1}$$

$$\upsilon_{12} = \upsilon_{12f} V_f + \upsilon_m V_m \tag{2}$$

$$\alpha_{22} = \alpha_{33} = (1 + \upsilon_m) V_m \alpha_m + \left(1 + \upsilon_{12f} \frac{\alpha_{1f}}{\alpha_{2f}}\right) V_f \alpha_{2f} - \upsilon_{12} \alpha_{11} \tag{3}$$

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 diection.

TABLE 4: LAMINA THERMAL EXPANSION COEFFICIENTS

Thermal Expansion Coefficient	Value (1/K)
$\{\alpha_{11}, \alpha_{22}, \alpha_{33}\}$	{3.72E-8, 3.47E-5, 3.47E-5}

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^2K) 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.

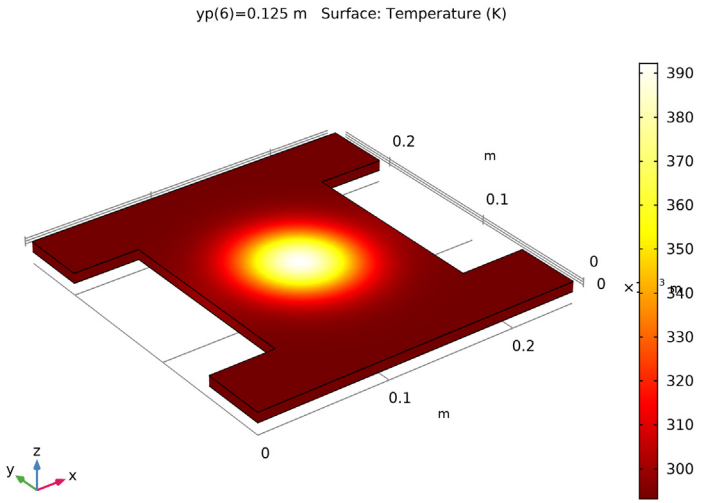

Figure 4: Temperature profile at yp = 12.5 cm.

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.

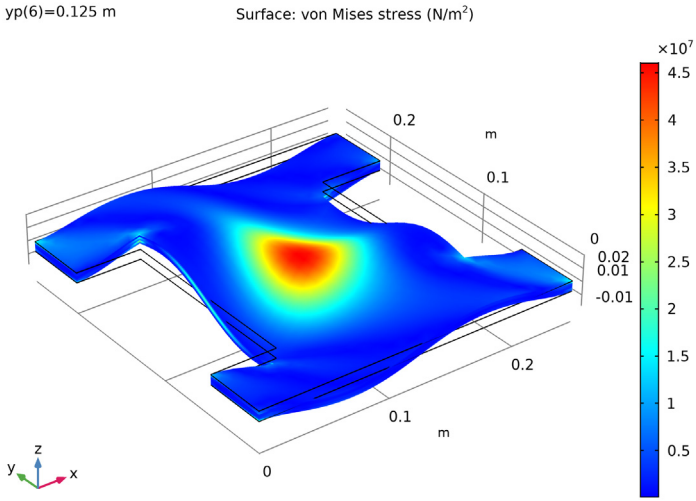

Figure 5: von Mises stress distribution in laminated composite shell at yp = 0.125 m.

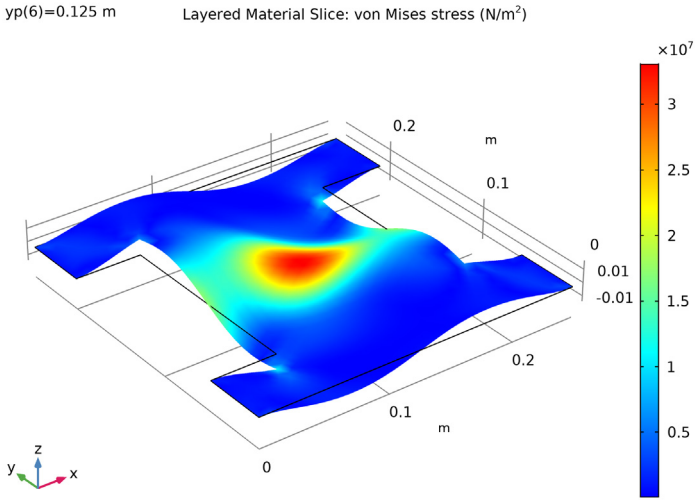

Figure 6: von Mises stress distribution at the mid-plane of laminated composite shell at yp = 0.125 m.

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.

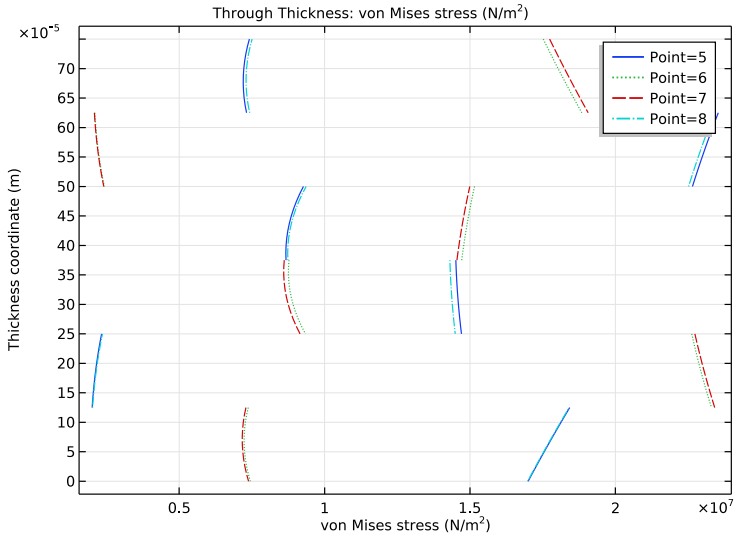

Figure 7: Through-thickness variation of von Mises stress at selected points when yp = 0.125 m.

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.

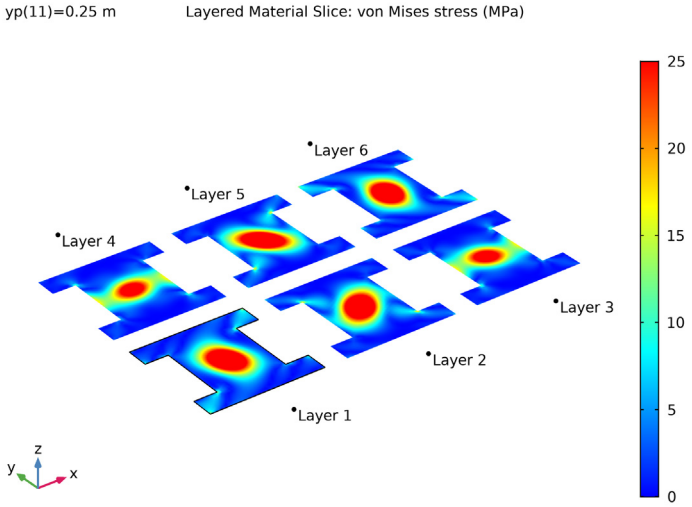

Figure 8: von Mises stress in laminate coordinate system at the mid-plane of each layer when yp = 0.125 m.

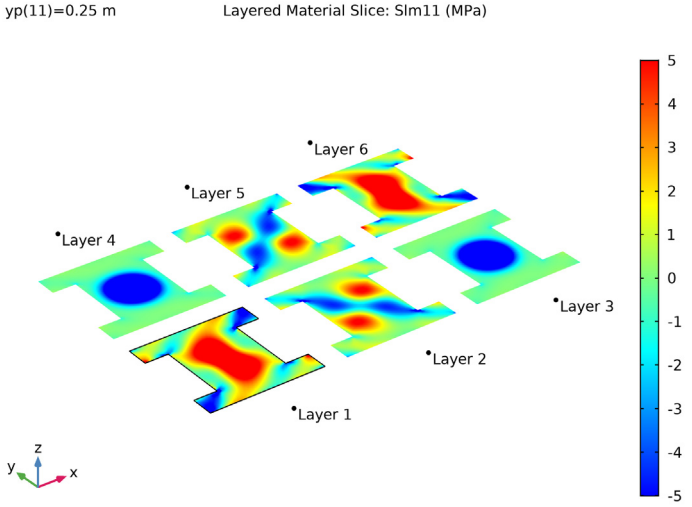

Figure 9: Stress component 11 (fiber direction) at the mid-plane of each layer when yp = 0.125 m.

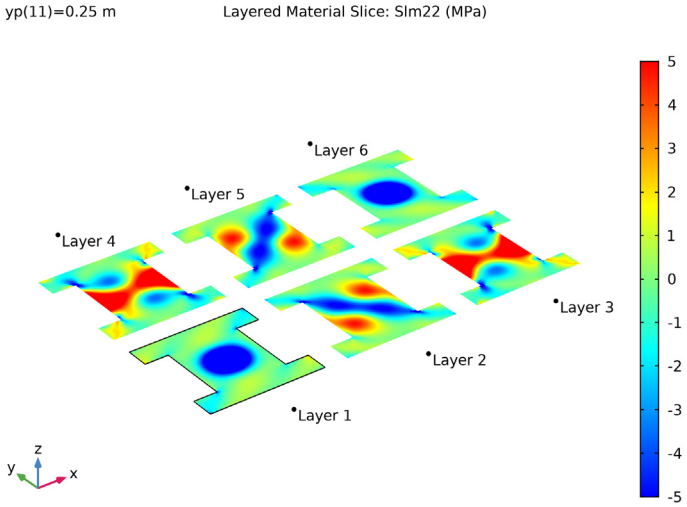

Figure 10: Stress component 22 (transverse to fiber direction) at the mid-plane of each layer when yp = 0.125 m.

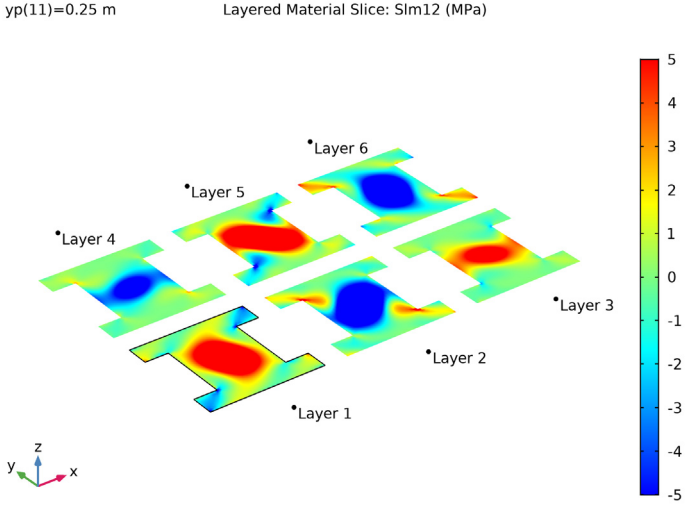

Figure 11: Stress component 12 (in-plane shear) i at the mid-plane of each layer when yp = 0.125 m.

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

1. N. Srisuk, *A Micromechanics Model of Thermal Expansion Coefficient in Fiber Reinforced Composites,* Master Thesis-The University of Texas st Arlington, 2010.

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_laminated_composite_shell_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_laminated_composite_shell_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 (r1)

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 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 (COMP1)

Boundary System 1 (sys1)

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

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

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

LAYERED SHELL (LSHELL)

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:

Property	Variable	Value	Unit	Property group
Thermal conductivity	{k11, k22, k33} ; kij = 0	{k1, k2, k2}	W/(m·K)	Basic
Density	rho	1	kg/m³	Basic
Heat capacity at constant pressure	Cp	1	J/(kg·K)	Basic

Property	Variable	Value	Unit	Property group
Elasticity matrix	{D11, D12, D22, D13, D23, D33, D14, D24, D34, D44, D15, D25, D35, D45, D55, D16, D26, D36, D46, D56, D66} ; Dij = Dji	`{D_11, D_12, D_22, D_13, D_23, D_33,0, 0,0, D_44,0, 0,0,0, D_55,0, 0,0,0,0, D_66}`	Pa	Anisotropic
Coefficient of thermal expansion	{alpha11, alpha22, alpha33} ; alphaij = 0	`{alpha1, alpha2, alpha2}`	1/K	Basic

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 (lshell)** and choose **Heat Transfer in Shells (htlsh)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** 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 (llmat1)-[lmat1]**.

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 `P0`.

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 (llmat1)-[lmat1]**.

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:

Parameter name	Parameter value list	Parameter unit
yp (y-position of beam)	`range(0,0.1*a,a)`	`m`

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 (lshell)

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

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 (lshell)** toolbar, click **Plot**.

Stress, Slice (lshell)

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

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 (lshell)** toolbar, click **Plot**.

Stress, Through Thickness (lshell)

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

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 (lshell)** 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 (lshell)** 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

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

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 7

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

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.