Created in COMSOL Multiphysics 6.2

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 makes 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, such as in 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. A layerwise approach is used to model the structural part of the shell.

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

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 250 mm, the web thickness is 150 mm, the flange width is 250 mm, and the flange thickness is 50 mm. The geometry of the laminate is shown in Figure 1.

STACKING SEQUENCE

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 midplane of the laminate. Each layer of the composite shell has a thickness of 0.125 mm as shown in Figure 3.

The 3D representation of the geometry as well as first principal material direction showing the fiber orientation in each layer of the physical geometry are shown in Figure 4 and Figure 5 respectively.

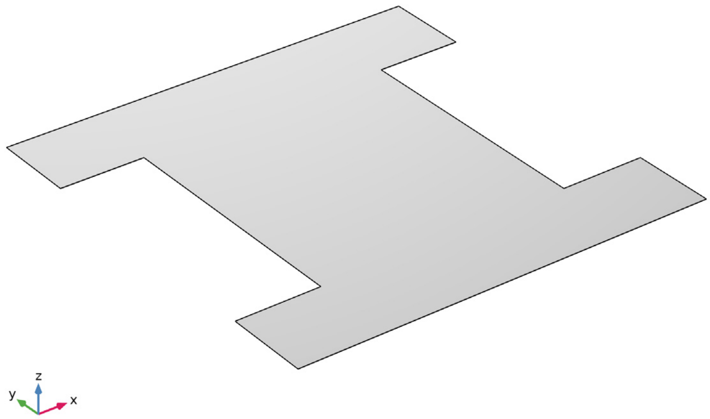

Figure 1: Geometry of the laminated composite shell.

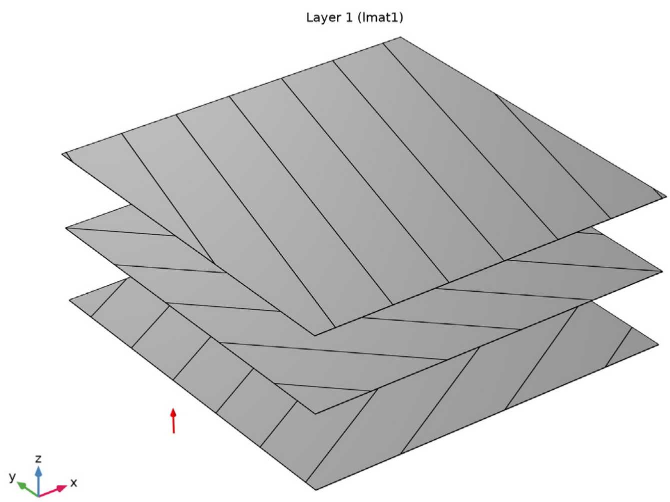

Figure 2: Stacking sequence [30/-45/75/-75/45/30], showing the fiber orientation in each layer from bottom to top.

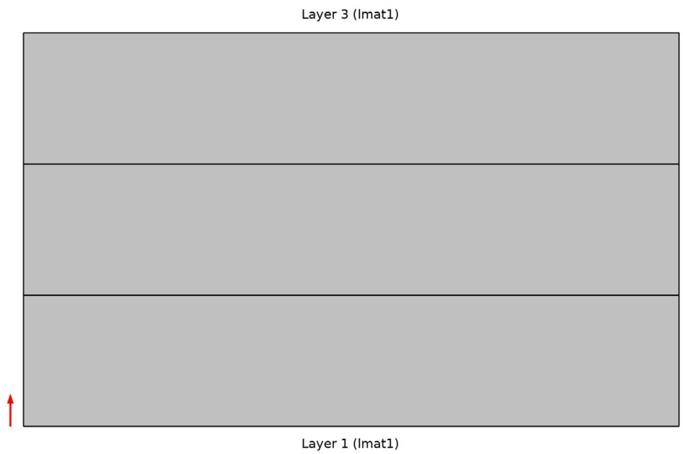

Figure 3: Through thickness view of the laminate with a layer thickness of 0.125 mm.

MATERIAL PROPERTIES

All the layers 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 nonzero 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.

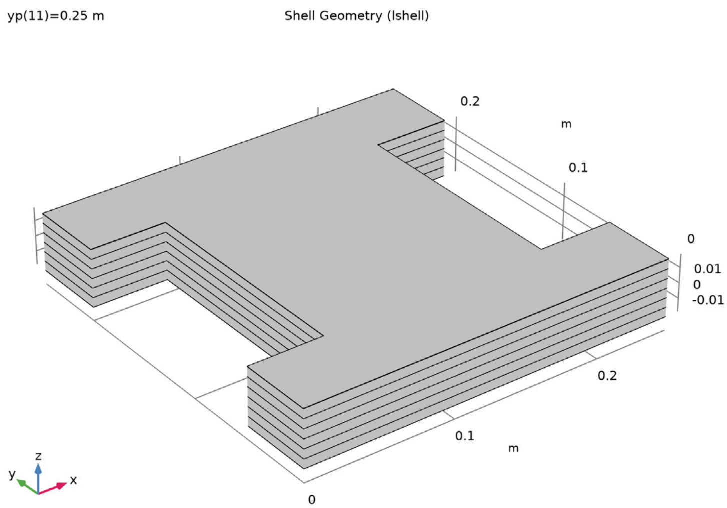

Figure 4: 3D geometric representation of the laminated composite shell.

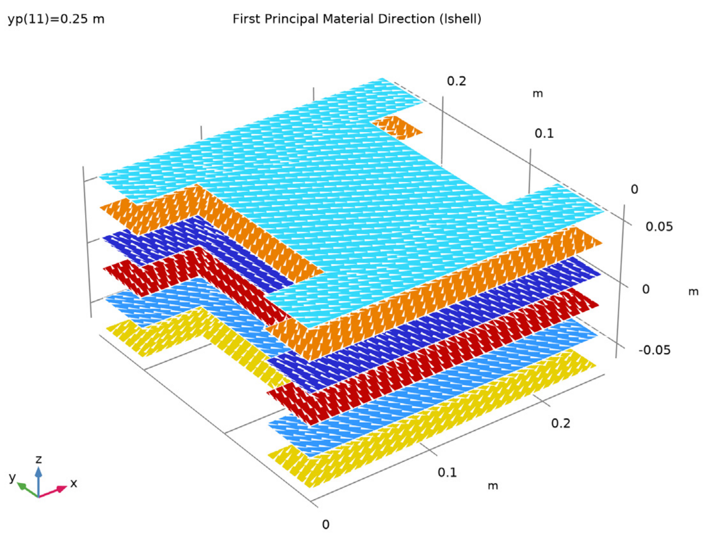

Figure 5: First principal material direction showing the fiber orientation in each layer of the physical geometry. Ply angle is used as a color for each layer.

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 direction.

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 $X = 250$ mm are fixed.
- Thermal boundary conditions: The temperature is set to room temperature at the edges at $X = 0$ and $X = 250$ mm. A convective heat flux with a heat transfer coefficient of 20 $W/(m^2 \cdot K)$ 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-positions of the beam source are fixed in space at 125 mm and 250 mm, whereas the y-position of the beam is varied from 0 to 250 mm. The standard deviation of the beam is taken as 1/10 of its height (or z-position), which is 25 mm.

Results and Discussion

The temperature profile in the composite shell when the beam power heat source is above its center is shown in Figure 6. 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 inspected by creating line plots along the X- and Y-axes as shown in the model.

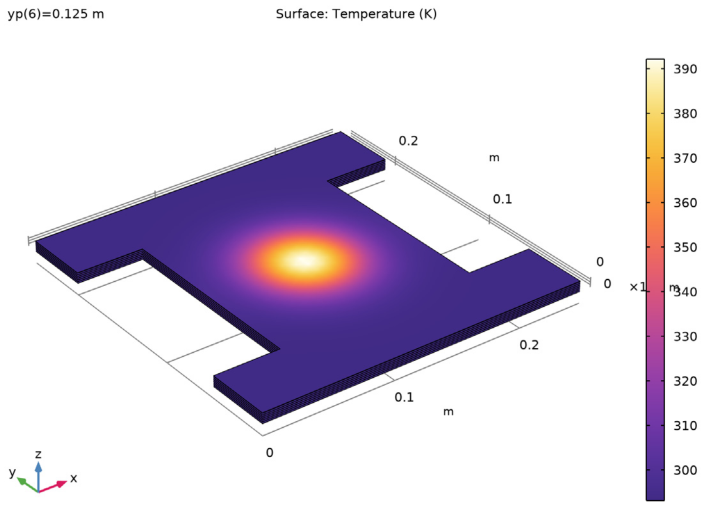

Figure 6: Temperature profile at yp = 125 mm.

The effects of the material orthotropy and layer orientations are evident in the thermal stresses and deformations pattern as shown in Figure 7. The overall thermal stress pattern is similar to the temperature profile shown in Figure 6, as the shell is only subjected to thermal loads. An interesting deformation pattern caused by the orthotropy and layer orientations can however be observed.

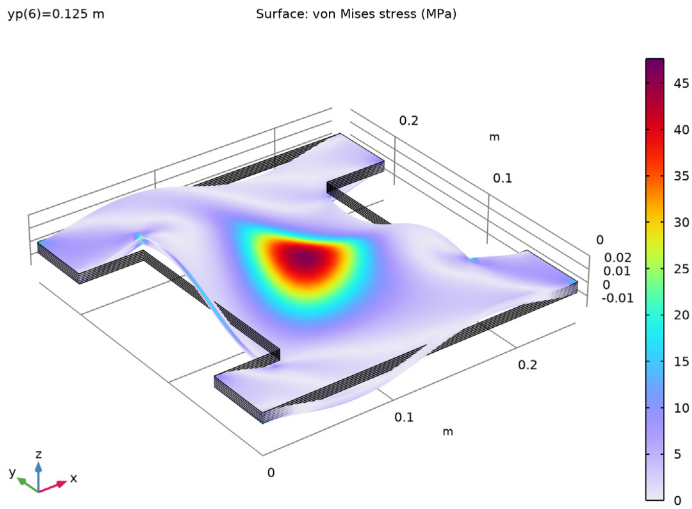

Figure 7: von Mises stress distribution in the laminate yp= 0.125 m.

To see the effect of layer orientation on the von Mises stress distribution, a Layered Material Slice plot is generated at the midplane of the laminated composite shell, as shown in Figure 8. It can be seen that it has a different stress distribution as well as magnitude when compared to Figure 7 in which the stress distribution is shown for the top layer.

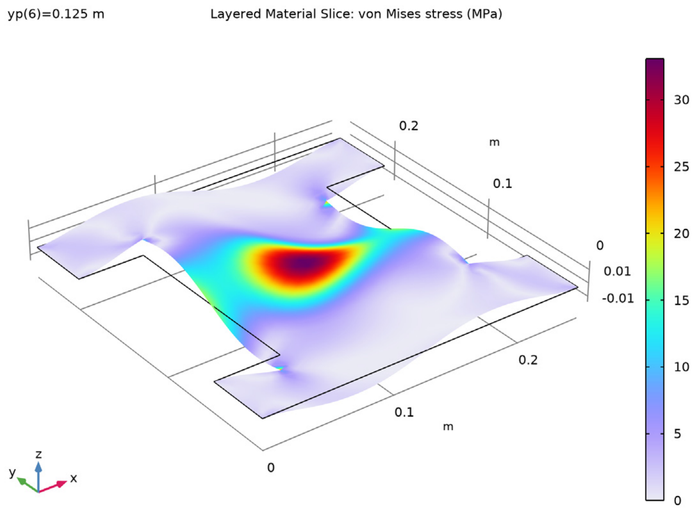

Figure 8: von Mises stress distribution at the midplane of the laminate at yp = 125 mm.

Figure 9 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 that are diagonally opposite.

.

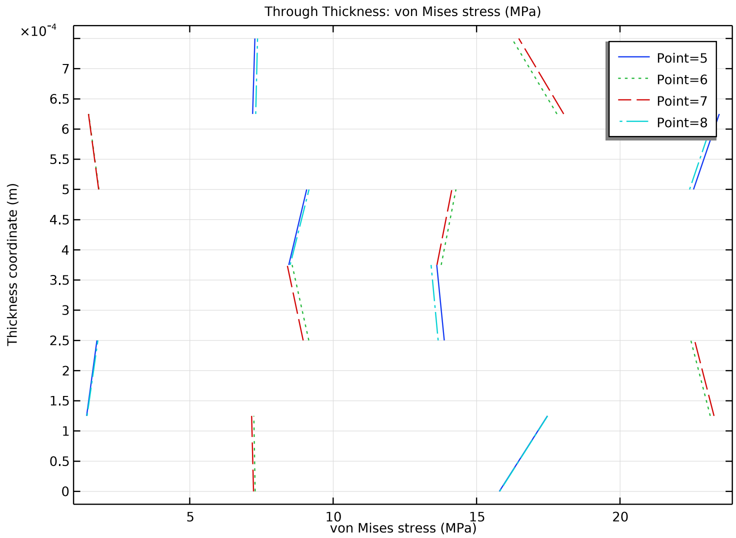

Figure 9: Through-thickness variation of von Mises stress at selected points when yp = 125 mm.

Figure 10 through Figure 13 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 midplane of each layer. The effect of the antisymmetric layup is clearly seen in Figure 10, Figure 11, and Figure 12. For example, the stress patterns in Layer 1 and Layer 6 are similar, but antisymmetric about the midplane of the laminate.

Figure 13 shows the shear stress distribution and also has an antisymmetric pattern. Also, the sign of the stress is reversed when comparing the top and bottom layers because of the antisymmetry.

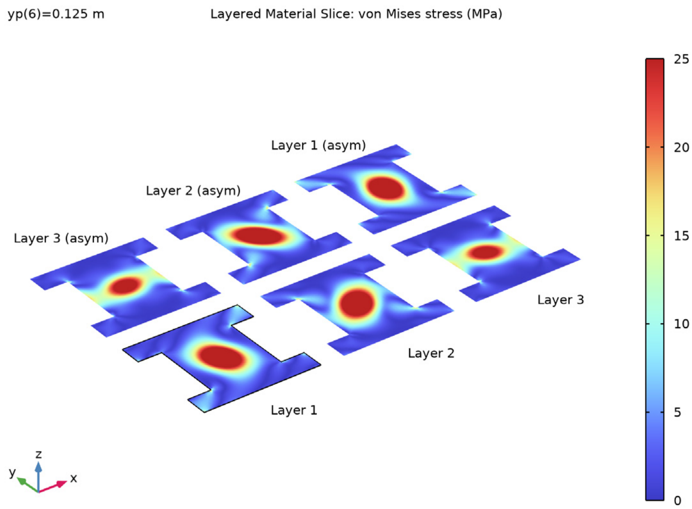

Figure 10: von Mises stress in laminate coordinate system at the midplane of each layer when yp = 125 mm.

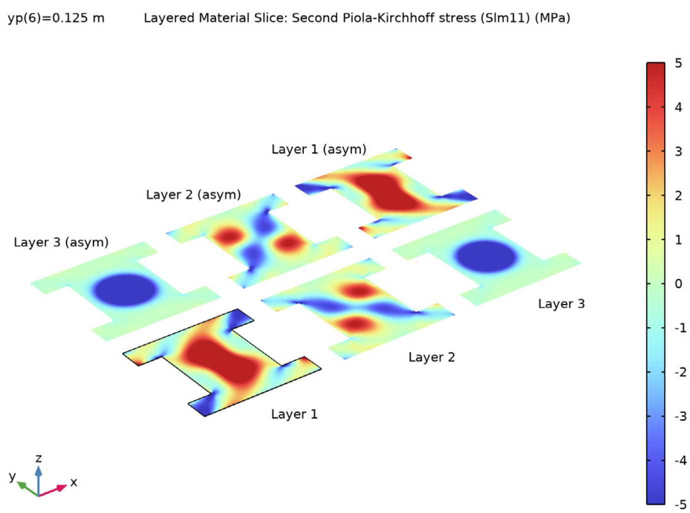

Figure 11: Stress component 11 (fiber direction) at the midplane of each layer when yp = 125 mm.

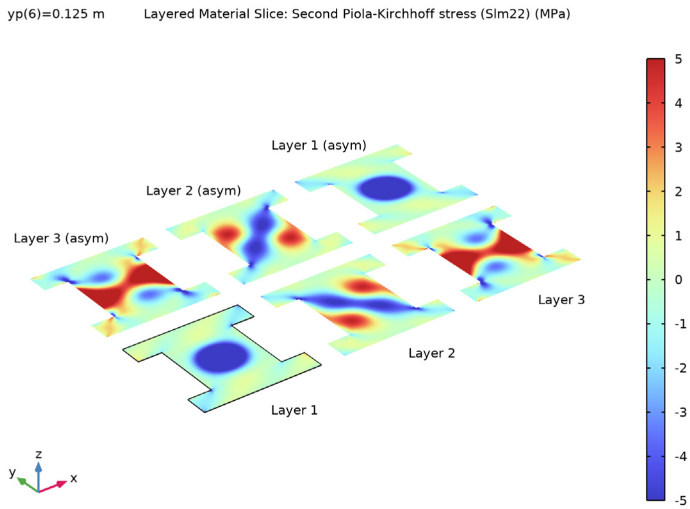

Figure 12: Stress component 22 (transverse to fiber direction) at the midplane of each layer when yp = 125 mm.

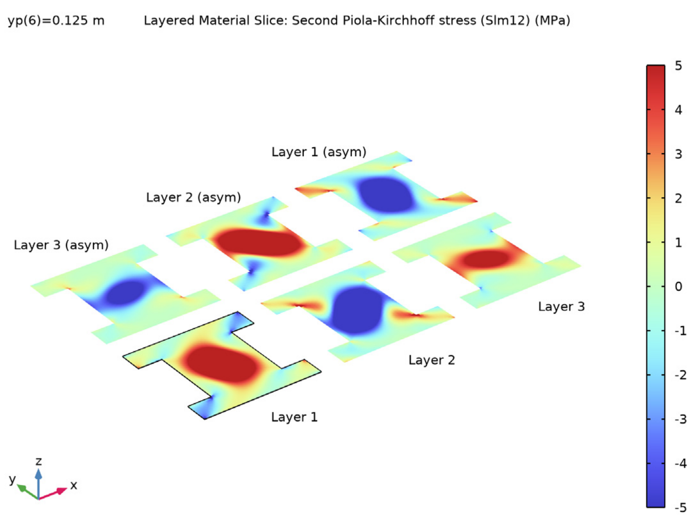

Figure 13: Stress component 12 (in-plane shear) at the midplane of each layer when yp = 125 mm.

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 using the Layered Shell interface or the *Equivalent Single Layer (ESL)* theory using the **Linear Elastic Material, Layered** 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 the 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 **Structural Mechanics>Thermal–Structure Interaction>Thermal Stress, Layered Shell**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters: General

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`.

In a separate **Parameters** node, load the thermal expansion parameters from a file.

Parameters: Thermal Expansion

1 In the **Home** toolbar, click Pi **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: Carbon–Epoxy

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. The laminate is antisymmetric. Only half of the laminate layers are listed in the file. The transformation into the full laminate is performed through the layered material settings in the **Layered Material Link** node.

Layered Material: [30/-45/75]_as

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.

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

5 Click **Layer Cross-Section Preview** in the upper-right corner of the **Layer Definition** section.

6 Click to expand the **Preview Plot Settings** section. In the **Distance between the orientation lines** text field, type `0.15`.

7 In the **Thickness-to-width ratio** text field, type `0.6`.

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

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

10 Click **Layer Stack Preview** in the upper-right corner of the **Layer Definition** section.

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node.

2 Right-click **Geometry 1** and choose **Work Plane**.

Work Plane 1 (wp1)>Square 1 (sq1)

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

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

3 In the **Side length** text field, type `a`.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **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 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **copy1** and **r1** only.

6 In the **Work Plane** toolbar, click **Build All**.

Create an edge selection for applying structural and thermal boundary conditions.

DEFINITIONS

Fixed Edges

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Fixed Edges` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edges 1, 4, 11, and 12 only.

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 laminate, partially defined in the **Layered Material** node, can be transformed into full antisymmetric laminate using a transform option in the layered material settings.

2 In the **Settings** window for **Layered Material Link**, locate the **Layered Material Settings** section.

3 From the **Transform** list, choose **Antisymmetric**.

The geometry is in the XY-plane, in which the fibers are oriented along the X direction. Therefore, align the first axis of the laminate coordinate system with the X direction.

4 Locate the **Orientation and Position** section. Click **Go to Source** for **Coordinate system**.

DEFINITIONS (COMP1)

Boundary System 1 (sys1)

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

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

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

LAYERED SHELL (LSHELL)

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 **Material symmetry** list, choose **Anisotropic**.

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**
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
Density	rho	1	kg/m³	Basic
Thermal conductivity	{k11, k22, k33} ; kij = 0	`{k1, k2, k2}`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	1	J/(kg·K)	Basic
Coefficient of thermal expansion	{alpha11, alpha22, alpha33} ; alphaij = 0	`{alpha1, alpha2, alpha2}`	1/K	Basic

HEAT TRANSFER IN SHELLS (HTLSH)

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Shells (htlsh)** click **Solid 1**.

2 In the **Settings** window for **Solid**, locate the **Layer Model** section.

3 From the **Layer type** list, choose **General**.

Use a deposited beam as a heat source through a **Deposited Beam Power, Interface** node. Select a beam orientation and origin point 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 **Apply to** list, choose **Selected interfaces**.

5 In the **Selection** table, clear the check boxes for **Layer 1 down**, **Layer 1-Layer 2**, **Layer 2-Layer 3**, **Layer 3 up**, **Layer 2-Layer 3 (asym)**, and **Layer 1-Layer 2 (asym)**.

6 Locate the **Beam Orientation** section. 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

The standard deviation of the beam is taken as 1/10 of its height which is 25 mm.

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 **Apply to** list, choose **Selected interfaces**.

5 In the **Selection** table, clear the check box for **Layer 1 down (asym)**.

6 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

7 In the h text field, type `ht`.

Temperature 1

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

2 In the **Settings** window for **Temperature**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Fixed Edges**.

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 In the **Settings** window for **Fixed Constraint**, locate the **Edge Selection** section.
3 From the **Selection** list, choose **Fixed Edges**.
4 In the **Model Builder** window, click **Layered Shell (lshell)**.
5 In the **Settings** window for **Layered Shell**, click to expand the **Default Through-Thickness Result Location** section.
6 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 the **Layered Material** datasets to 10 to improve visualization.

RESULTS

Layered Material

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Layered Material**.
2 In the **Settings** window for **Layered Material**, locate the **Layers** section.
3 In the **Scale** text field, type `10`.

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**.

Surface 1

1 In the **Model Builder** window, expand the **Stress (lshell)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.
4 In the **Stress (lshell)** toolbar, click **Plot**.

Temperature, Shell (htlsh)

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

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.
2 Go to the **Add Predefined Plot** window.
3 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Stress, Slice (lshell)**.
4 Click **Add Plot** in the window toolbar.

RESULTS

Stress, Slice (lshell)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2 From the **Parameter value (yp (m))** list, choose **0.125**.

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Stress, Slice (lshell)** node, then click **Layered Material Slice 1**.
2 In the **Settings** window for **Layered Material Slice**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.
4 In the **Stress, Slice (lshell)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Stress, Through Thickness (lshell)**.

3 Click **Add Plot** in the window toolbar.

RESULTS

Stress, Through Thickness (lshell)

1 In the **Model Builder** window, expand the **Results>Stress, Through Thickness (lshell)** node, then click **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, 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 Locate the **x-Axis Data** section. From the **Unit** list, choose **MPa**.

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

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

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Geometry and Layup (lshell)>Shell Geometry (lshell)**.

3 Click **Add Plot** in the window toolbar.

4 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Geometry and Layup (lshell)>First Principal Material Direction (lshell)**.

5 Click **Add Plot** in the window toolbar.

6 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Layered Material 2 (Shell Geometry)

1 In the **Model Builder** window, under **Results>Datasets** click **Layered Material 2 (Shell Geometry)**.

2 In the **Settings** window for **Layered Material**, locate the **Layers** section.

3 In the **Scale** text field, type 50.

Layered Material 2 (Material Direction)

1 In the **Model Builder** window, click **Layered Material 2 (Material Direction)**.

2 In the **Settings** window for **Layered Material**, locate the **Layers** section.

3 In the **Scale** text field, type 200.

Shell Geometry (lshell)

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

2 In the **Shell Geometry (lshell)** toolbar, click **Plot**.

First Principal Material Direction (lshell)

1 In the **Model Builder** window, click **First Principal Material Direction (lshell)**.

2 In the **First Principal Material Direction (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.

6 Right-click **Cut Line 3D 1** and choose **Duplicate**.

Cut Line 3D 2

1 In the **Model Builder** window, click **Cut Line 3D 2**.

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.

Temperature Distribution along X-axis

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

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

5 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 **y-Axis Data** section.

3 In the **Expression** text field, type `T`.

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

In the **Model Builder** window, right-click **Temperature Distribution along X-axis** and choose **Duplicate**.

Temperature Distribution along Y-axis

1 In the **Model Builder** window, under **Results** click **Temperature Distribution along X-axis 1**.

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

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

Line Graph 1

1 In the **Model Builder** window, expand the **Temperature Distribution along Y-axis** node, then click **Line Graph 1**.

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

3 In the **Expression** text field, type `Y`.

4 In the **Temperature Distribution along Y-axis** toolbar, click **Plot**.

Stress: von Mises

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

Layered Material Slice 1

1 Right-click **Stress: von Mises** 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 From the **Unit** list, choose **MPa**.

5 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Layer midplanes**.

6 Locate the **Layout** section. From the **Displacement** list, choose **Rectangular**.

7 In the **Relative x-separation** text field, type `0.2`.

8 In the **Relative y-separation** text field, type `0.2`.

9 Select the **Show descriptions** check box.

10 In the **Relative separation** text field, type `0.35`.

11 Locate the **Coloring and Style** section. Click **Change Color Table**.

12 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.

13 Click **OK**.

14 In the **Settings** window for **Layered Material Slice**, click to expand the **Range** section.

15 Select the **Manual color range** check box.

16 In the **Maximum** text field, type `25`.

Stress: von Mises

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

2 In the **Model Builder** window, click **Stress: von Mises**.

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

In order to plot different normal and shear components of the stress tensor in the laminate coordinate system at the midplane of each layer, duplicate the previous plot and change the plot expressions to `lshell.Slm11`, `lshell.Slm22`, and `lshell.Slm12`, respectively.

Create an animation under the **Export** node to visualize the temperature and stress profiles as the deposited beam heat source moves in the Y direction.