Bending of a Simply Supported Composite Laminate

Introduction

A composite material is a heterogeneous material formed of two or more constituents integrated together to achieve enhanced structural performance. Owing to the improved strength and reduced weight compared to conventional materials, the range of applications of composite materials spans diverse fields. This necessitates a thorough understanding of the behavior of these materials under various loading conditions.

This verification example analyses a simply supported composite plate subjected to a transverse sinusoidal distributed load. The plate is made of three layers with a cross-ply configuration (0/90/0). The analysis is performed using both the layerwise and the Equivalent Single Layer (ESL) theories, and the results are compared with the exact 3D elasticity solutions [Ref. 1]. The estimates of the in-plane normal stress, the transverse normal stress, and the transverse shear stresses in each layer show an exact match with the reference solution.

Model Definition

The model geometry consists of a square composite plate of 200 mm side length. All the edges of the plate are simply supported. In addition to constraining the out-of-plane translation, such support conditions also constrain the in-plane translation in the edge direction.

For the bending analysis, a sinusoidally varying distributed load is applied on the top surface of the plate in the transverse direction. Because of symmetry in geometry and loading, only one quarter of the plate is modeled, with the symmetry boundary conditions at two edges as shown in Figure 1.

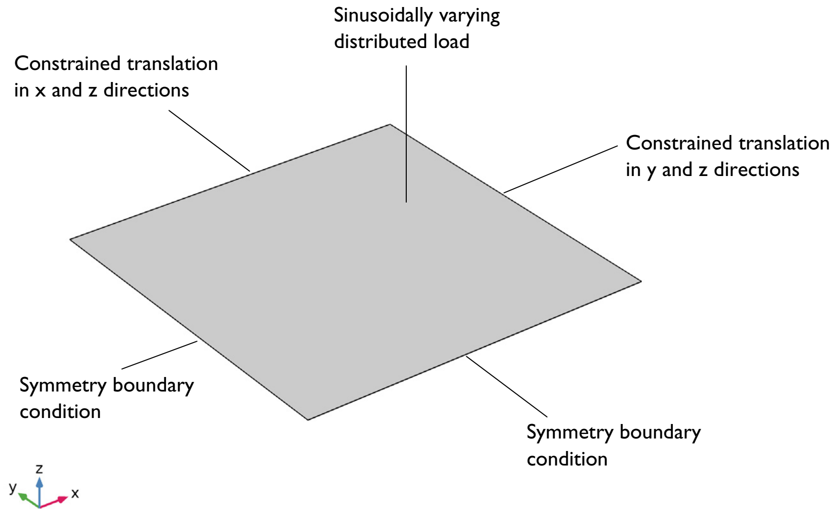

Figure 1: Model geometry showing one quarter of the laminated composite plate.

LAMINA MATERIAL PROPERTIES

The homogenized orthotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) of the lamina are given in Table 1.

TABLE 1: MATERIAL PROPERTIES OF A LAMINA.

Material property	Value
$\{E_1, E_2, E_3\}$	{25e6,1e6,1e6}(psi)
$\{G_{12}, G_{23}, G_{13}\}$	{0.5e6,0.2e6,0.5e6}(psi)
$\{\upsilon_{12}, \upsilon_{23}, \upsilon_{13}\}$	{0.25,0.25,0.25}

THICKNESS AND STACKING SEQUENCE

The composite plate considered in this example is rather thick, having a thickness to side ratio of 1/4. Thus the total thickness of the laminate is 50 mm.

The laminate consists of three layers in a cross-ply layup. The stacking sequence of the laminate is shown in Figure 2, and the corresponding fiber orientations are given in Table 2. The fiber orientations are presented with respect to the first axis of the laminate coordinate system as shown in Figure 3.

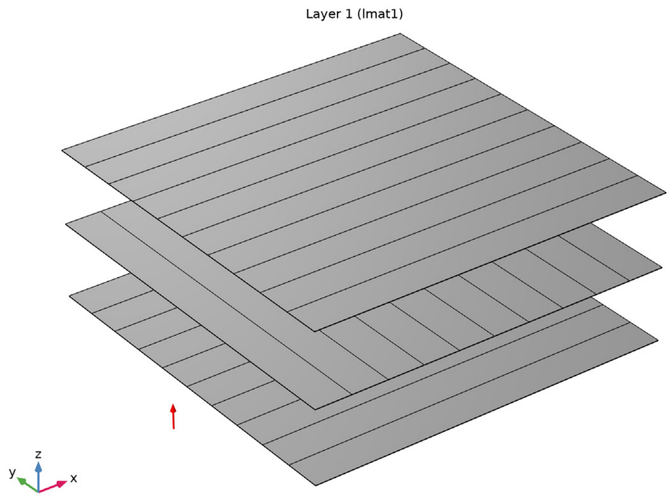

Figure 2: Stacking sequence of laminated composite plate showing fiber orientation in each layer from bottom to top.

TABLE 2: FIBER ORIENTATION IN LAYERED MATERIAL

Layer number	**Fiber orientation in layered material (°)**
1	0
2	90
3	0

FINITE ELEMENT MESH

The structure (quarter plate) is discretized using a mapped mesh as shown in Figure 4. The distribution of elements along the edges is taken as 2-by-2 [Ref. 1]. Additionally, two sub-layers are used in each material layer of the plate as given in the reference.

Cubic serendipity shape functions are used in layerwise theory in order to capture the shear stress profiles accurately.

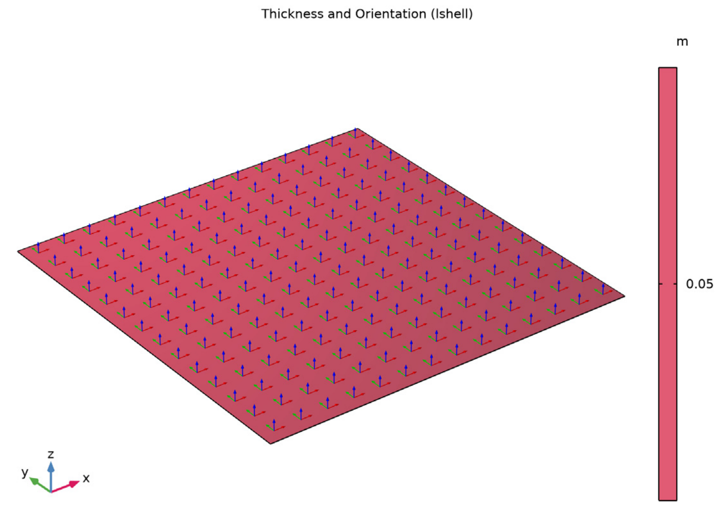

Figure 3: The laminate coordinate system used in the model.

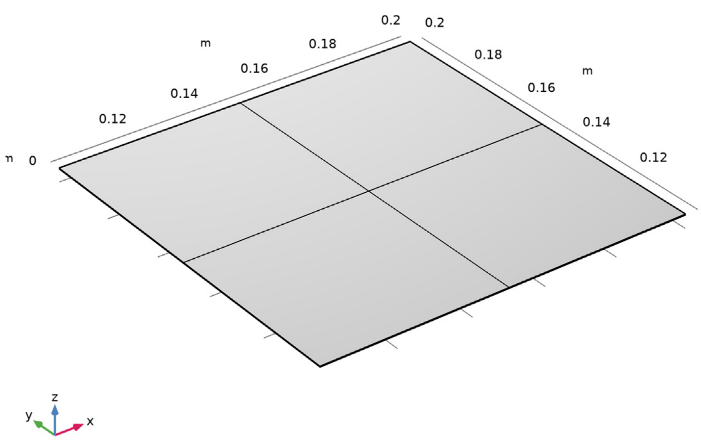

Figure 4: Finite element mesh with 4 elements.

NONDIMENSIONAL STRESS EXPRESSIONS

The following nondimensional stress expressions are defined in order to compare to the reference solution:

$$\bar{\sigma}_{xx} = \sigma_{xx}(A, A, z) \times \frac{h^2}{q_0 a^2}$$

$$\bar{\sigma}_{zz} = \sigma_{zz}(A, A, z) \times \frac{1}{q_0}$$

$$\bar{\sigma}_{yz} = \sigma_{yz}(A, B, z) \times \frac{h}{q_0 a}$$

$$\bar{\sigma}_{xz} = \sigma_{xz}(B, A, z) \times \frac{h}{q_0 a}$$

$$A = 1.105 \times \frac{a}{2}$$

$$B = 1.894 \times \frac{a}{2}$$

where h is the laminate thickness, a is the side length, and q_0 is the transverse load amplitude. The three point locations (A, A), (A, B), and (B, A) on the composite plate is shown below:

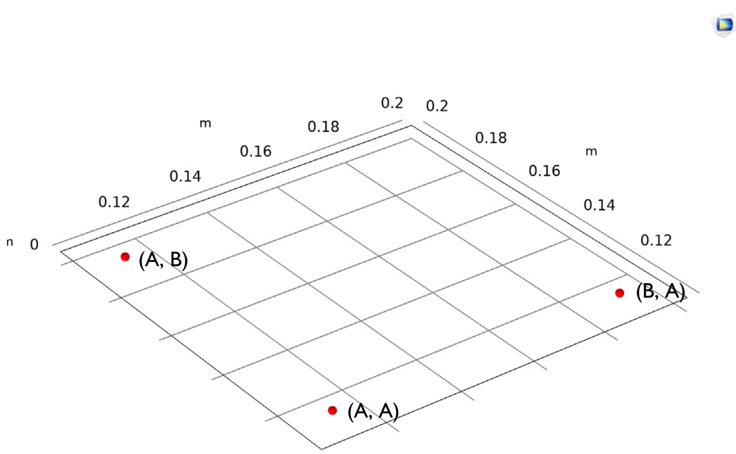

Figure 5: Points on the laminated composite plate where different stress values are compared.

Results and Discussion

Figure 6 and Figure 7 show the von Mises stress distribution in one quarter of the composite laminate computed using the layerwise theory and the ESL theory, respectively. The distribution of von Mises stress in the full composite laminate is shown in Figure 8.

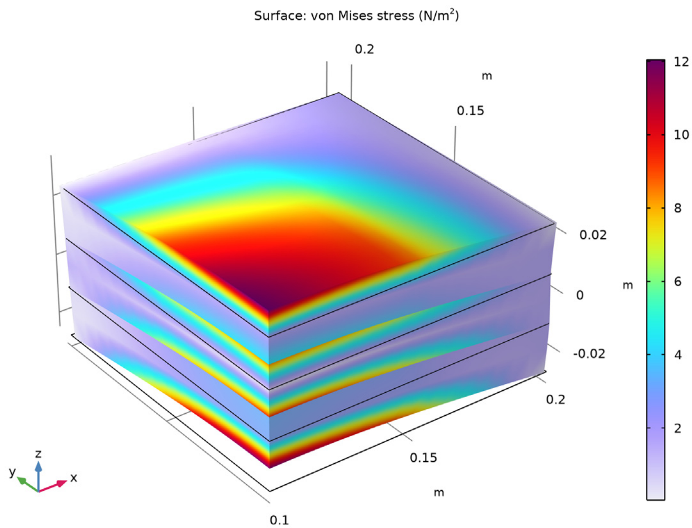

Figure 6: von Mises stress distribution computed using layerwise theory.

You can also compute the through-thickness variation of different stress components using the layerwise and ESL theories. Both theories can be used to study thin to moderately thick laminates. The fact that the layerwise formulation has degrees of freedom available at each layer makes it preferable for the analysis of thick laminates. As the thickness to side ratio for the current problem is 1/4, there is good agreement between the layerwise theory and the stress values computed from 3D elastic solutions [Ref. 1] as shown in Figure 9, Figure 10, Figure 11, and Figure 12.

The transverse shear stresses, in Figure 11 and Figure 12, depict the interlaminar stress values at the junction/interface of the two respective layers. It can be seen that the in-plane normal stress, shown in Figure 9, is discontinuous across the layers and having different stress values in two layers at the junction/interface whereas the transverse shear stress is continuous across the layers. This unique value of transverse shear stress at the junction/interface is known as interlaminar stress, which is one of the deciding factors for interfacial failures like delamination.

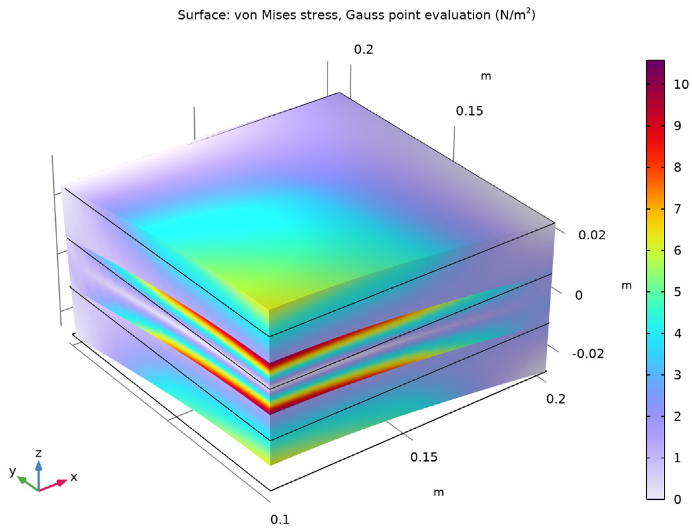

Figure 7: von Mises stress distribution computed using the ESL theory.

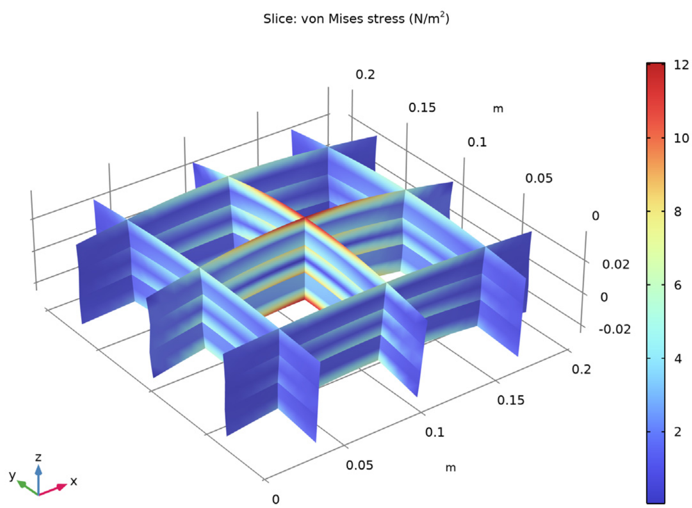

Figure 8: von Mises stress distribution in the full laminate computed using the layerwise theory.

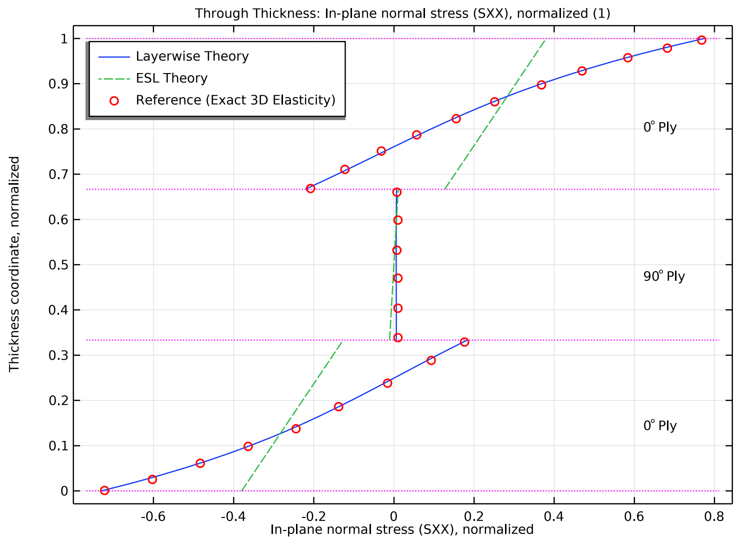

Figure 9: Through-thickness variation of normalized in-plane normal stress (SXX).

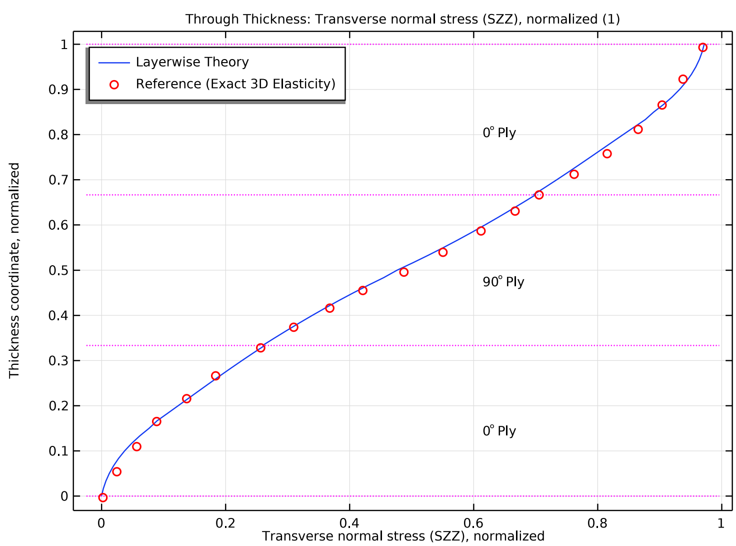

Figure 10: Through-thickness variation of normalized transverse normal stress (SZZ).

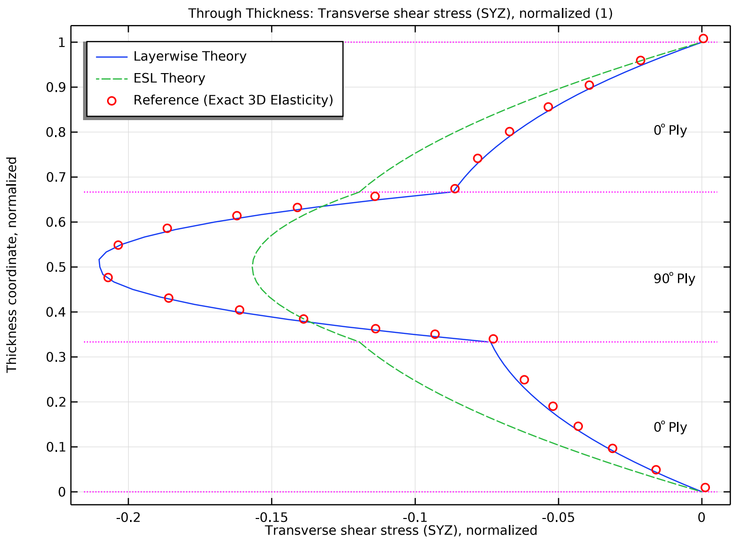

Figure 11: Through-thickness variation of normalized transverse shear stress (SYZ).

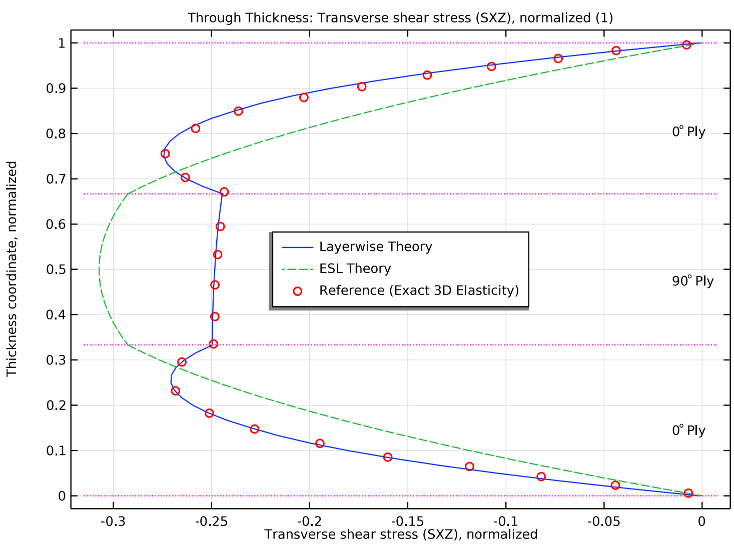

Figure 12: Through-thickness variation of normalized transverse shear stress (SXZ).

Notes About the COMSOL Implementation

- Modeling a composite laminated shell requires a surface geometry (2D), in general called a base surface, and a **Layered Material** node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other having different thicknesses, material properties, and fiber orientations. You can optionally specify the interface materials between the layers and control mesh elements in each layer.
- From a constitutive model point of view, you can either use the *layerwise* theory based **Layered Shell** interface, or the *Equivalent Single Layer* theory based **Layered Linear Elastic Material** node in the **Shell** interface.
- In Layered Shell interface, in order to increase the accuracy in the through thickness profile at the same time keeping the total degrees of freedom less, a cubic shape order is used in out-of-plane direction keeping the in-plane shape order as quadratic.

Reference

1. J. N. Reddy, *Mechanics of Laminated Composite Plates and Shells: Theory and Analysis, Second Edition*, CRC Press, 2004.

Application Library path: `Composite_Materials_Module/Verification_Examples/simply_supported_composite_laminate`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Layered Shell (lshell)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

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

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 Click **Load from File**.

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

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

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/2`.

4 Locate the **Position** section. In the **xw** text field, type `a/2`.

5 In the **yw** text field, type `a/2`.

6 Click **Build Selected**.

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

DEFINITIONS

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**.

GLOBAL DEFINITIONS

Material 1 (mat1)

In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

Layered Material: [0/90/0]

1 Right-click **Materials** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `Layered Material: [0/90/0]` in the **Label** text field.

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

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 1	Material 1 (mat1)	0.0	th1	2

4 Click **Add** two times.

5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 2	Material 1 (mat1)	90	th1	2
Layer 3	Material 1 (mat1)	0	th1	2

6 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type `0.6`.

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

MATERIALS

Layered Material Link 1 (llmat1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Layers>Layered Material Link**.

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	{Evector1, Evector2, Evector3}	{E1,E2, E3}	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	{nu12, nu23, nu13}	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	{G12, G23, G13}	N/m²	Orthotropic
Density	rho	1	kg/m³	Basic

In order to have more accuracy in the through-thickness direction, use **Quadratic-cubic serendipity** element defining quadratic variation of displacement field on the base surface and cubic variation along the through-thickness direction.

LAYERED SHELL (LSHELL)

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

2 In the **Settings** window for **Layered Shell**, click to expand the **Discretization** section.

3 From the **Displacement field** list, choose **Quadratic-cubic serendipity**.

Prescribed Displacement 1

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

2 Select Edge 3 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

1 Right-click **Prescribed Displacement 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Edge 4 only.

5 Locate the **Prescribed Displacement** section. Clear the **Prescribed in x direction** check box.

6 Select the **Prescribed in y direction** check box.

Symmetry 1

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

2 Select Edges 1 and 2 only.

Face Load 1

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

2 In the **Settings** window for **Face Load**, locate the **Interface Selection** section.

3 From the **Apply to** list, choose **Top interface**.

4 Select Boundary 1 only.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	x
0	y
sdl	z

SHELL (SHELL)

In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.

Layered Linear Elastic Material 1

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

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

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

4 Select Boundary 1 only.

5 Click the **Show More Options** button in the **Model Builder** toolbar.

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

7 Click **OK**.

8 In the **Settings** window for **Layered Linear Elastic Material**, click to expand the **Shear Correction Factor** section.

9 From the list, choose **Based on 3D elasticity theory**.

Simply Supported 1

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

2 Select Edges 3 and 4 only.

3 In the **Settings** window for **Simply Supported**, locate the **In-Plane Displacement Constraints** section.

4 Clear the **Perpendicular to edge** check box.

Symmetry 1

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

2 Select Edges 1 and 2 only.

Face Load 1

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

2 Select Boundary 1 only.

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

4 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
sdl	z

MESH 1

Mapped 1

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

2 Select Boundary 1 only.

Distribution 1

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

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

3 From the **Selection** list, choose **All edges**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type 2.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

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

RESULTS

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

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

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

4 Locate the **Plane Data** section. In the **x-coordinate** text field, type `a/2`.

Mirror 3D 2

1 Right-click **Mirror 3D 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

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

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

Cut Point 3D: (A, A)

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

2 In the **Settings** window for **Cut Point 3D**, type `Cut Point 3D: (A, A)` in the **Label** text field.

3 Locate the **Point Data** section. In the **X** text field, type `A`.

4 In the **Y** text field, type `A`.

5 In the **Z** text field, type `0`.

Cut Point 3D: (A, B)

1 Right-click **Cut Point 3D: (A, A)** and choose **Duplicate**.

2 In the **Settings** window for **Cut Point 3D**, type `Cut Point 3D: (A, B)` in the **Label** text field.

3 Locate the **Point Data** section. In the **Y** text field, type `B`.

Cut Point 3D: (B, A)

1 Right-click **Cut Point 3D: (A, B)** and choose **Duplicate**.

2 In the **Settings** window for **Cut Point 3D**, type `Cut Point 3D: (B, A)` in the **Label** text field.

3 Locate the **Point Data** section. In the **X** text field, type B.

4 In the **Y** text field, type A.

Stress (shell)

Follow the instructions below to plot the von Mises stress distribution in the full composite plate as shown in Figure 8.

Stress (LW Theory): Full Laminate

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Stress (LW Theory): Full Laminate` in the **Label** text field.

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

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Slice (yz-planes)

1 Right-click **Stress (LW Theory): Full Laminate** and choose **Slice**.

2 In the **Settings** window for **Slice**, type `Slice (yz-planes)` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `lshell.mises`.

4 Locate the **Plane Data** section. In the **Planes** text field, type 3.

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

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

7 Click **OK**.

Deformation 1

Right-click **Slice (yz-planes)** and choose **Deformation**.

Slice (zx-planes)

1 In the **Model Builder** window, right-click **Slice (yz-planes)** and choose **Duplicate**.

2 In the **Settings** window for **Slice**, type `Slice (zx-planes)` in the **Label** text field.

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

4 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

5 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

6 In the **Planes** text field, type 3.

7 In the **Stress (LW Theory): Full Laminate** toolbar, click **Plot**.

Next, import the exact 3D elasticity solution and compare it with the computed solution.

Table: SXX

1 In the **Results** toolbar, click **Table**.
2 In the **Settings** window for **Table**, type `Table: SXX` in the **Label** text field.
3 Locate the **Data** section. Click **Import**.
4 Browse to the model's Application Libraries folder and double-click the file `simply_supported_composite_laminate_stress_xx.txt`.

Table: SZZ

1 In the **Results** toolbar, click **Table**.
2 In the **Settings** window for **Table**, type `Table: SZZ` in the **Label** text field.
3 Locate the **Data** section. Click **Import**.
4 Browse to the model's Application Libraries folder and double-click the file `simply_supported_composite_laminate_stress_zz.txt`.

Table: SYZ

1 In the **Results** toolbar, click **Table**.
2 In the **Settings** window for **Table**, type `Table: SYZ` in the **Label** text field.
3 Locate the **Data** section. Click **Import**.
4 Browse to the model's Application Libraries folder and double-click the file `simply_supported_composite_laminate_stress_yz.txt`.

Table: SXZ

1 In the **Results** toolbar, click **Table**.
2 In the **Settings** window for **Table**, type `Table: SXZ` in the **Label** text field.
3 Locate the **Data** section. Click **Import**.
4 Browse to the model's Application Libraries folder and double-click the file `simply_supported_composite_laminate_stress_xz.txt`.

Follow the instructions below to plot the through-thickness variation of the in-plane normal stress as shown in Figure 9.

In-plane Normal Stress (SXX)

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `In-plane Normal Stress (SXX)` in the **Label** text field.
3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type `In-plane normal stress (SXX), normalized`.

5 Select the **y-axis label** check box. In the associated text field, type `Thickness coordinate, normalized`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Through Thickness 1

1 In the **In-plane Normal Stress (SXX)** toolbar, click **More Plots** and choose **Through Thickness**.

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

3 From the **Dataset** list, choose **Cut Point 3D: (A, A)**.

4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>SXX_lw - In-plane normal stress (SXX), normalized**.

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

6 In the **Expression** text field, type `lshell.xd_rel`.

7 Find the **Interface positions** subsection. From the **Show interface positions** list, choose **All interfaces**.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
Layerwise Theory

11 In the **In-plane Normal Stress (SXX)** toolbar, click **Plot**.

Through Thickness 2

1 Right-click **Through Thickness 1** and choose **Duplicate**.

2 In the **Settings** window for **Through Thickness**, click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>SXX_esl - In-plane normal stress (SXX), normalized**.

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

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

5 Find the **Interface positions** subsection. From the **Show interface positions** list, choose **None**.

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

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

Legends
ESL Theory

8 In the **In-plane Normal Stress (SXX)** toolbar, click **Plot**.

Table Graph 1

1 In the **Model Builder** window, right-click **In-plane Normal Stress (SXX)** and choose **Table Graph**.

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

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

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

5 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

6 From the **Positioning** list, choose **In data points**.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Reference (Exact 3D Elasticity)

10 In the **In-plane Normal Stress (SXX)** toolbar, click **Plot**.

In-plane Normal Stress (SXX)

In the **Model Builder** window, click **In-plane Normal Stress (SXX)**.

Table Annotation 1

1 In the **In-plane Normal Stress (SXX)** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	**y-coordinate**	**Annotation**
`0.6`	`0.17`	`\[0^\circ \,\,` `\textrm{Ply}\]`
`0.6`	`0.5`	`\[90^\circ \,\,` `\textrm{Ply}\]`
`0.6`	`0.83`	`\[0^\circ \,\,` `\textrm{Ply}\]`

5 Select the **LaTeX markup** check box.

6 Locate the **Coloring and Style** section. Clear the **Show point** check box.

7 In the **In-plane Normal Stress (SXX)** toolbar, click **Plot**.

In-plane Normal Stress (SXX)

1 In the **Model Builder** window, click **In-plane Normal Stress (SXX)**.

2 Click **Plot**.

Follow the instructions below to plot the through-thickness variation of the transverse normal stress as shown in Figure 10.

Transverse Normal Stress (SZZ)

1 Right-click **In-plane Normal Stress (SXX)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Transverse Normal Stress (SZZ)` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `Transverse normal stress (SZZ), normalized`.

Through Thickness 1

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

2 In the **Settings** window for **Through Thickness**, click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>SZZ_lw - Transverse normal stress (SZZ), normalized**.

Through Thickness 2

In the **Model Builder** window, under **Results>Transverse Normal Stress (SZZ)** right-click **Through Thickness 2** and choose **Delete**.

Table Graph 1

1. In the **Model Builder** window, under **Results>Transverse Normal Stress (SZZ)** click **Table Graph 1**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. From the **Table** list, choose **Table: SZZ**.

Transverse Normal Stress (SZZ)

1. In the **Model Builder** window, click **Transverse Normal Stress (SZZ)**.
2. In the **Transverse Normal Stress (SZZ)** toolbar, click **Plot**.

Follow the instructions below to plot the through-thickness variation of the transverse shear stress as shown in Figure 11.

Transverse Shear Stress (SYZ)

1. In the **Model Builder** window, right-click **In-plane Normal Stress (SXX)** and choose **Duplicate**.
2. In the **Settings** window for **1D Plot Group**, type `Transverse Shear Stress (SYZ)` in the **Label** text field.
3. Locate the **Plot Settings** section. In the **x-axis label** text field, type `Transverse shear stress (SYZ), normalized`.

Through Thickness 1

1. In the **Model Builder** window, expand the **Transverse Shear Stress (SYZ)** node, then click **Through Thickness 1**.
2. In the **Settings** window for **Through Thickness**, locate the **Data** section.
3. From the **Dataset** list, choose **Cut Point 3D: (A, B)**.
4. Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>SYZ_lw - Transverse shear stress (SYZ), normalized**.

Through Thickness 2

1. In the **Model Builder** window, click **Through Thickness 2**.
2. In the **Settings** window for **Through Thickness**, locate the **Data** section.
3. From the **Dataset** list, choose **Cut Point 3D: (A, B)**.
4. Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>SYZ_esl - Transverse shear stress (SYZ), normalized**.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.
2 In the **Settings** window for **Table Graph**, locate the **Data** section.
3 From the **Table** list, choose **Table: SYZ**.

Table Annotation 1

1 In the **Model Builder** window, click **Table Annotation 1**.
2 In the **Settings** window for **Table Annotation**, locate the **Data** section.
3 From the **Source** list, choose **Local table**.
4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
`-0.02`	`0.17`	`\[0^\circ \,\, \textrm{Ply}\]`
`-0.02`	`0.5`	`\[90^\circ \,\, \textrm{Ply}\]`
`-0.02`	`0.83`	`\[0^\circ \,\, \textrm{Ply}\]`

Transverse Shear Stress (SYZ)

1 In the **Model Builder** window, click **Transverse Shear Stress (SYZ)**.
2 In the **Transverse Shear Stress (SYZ)** toolbar, click **Plot**.

Follow the instructions below to plot the through-thickness variation of the transverse shear stress as shown in Figure 12.

Transverse Shear Stress (SXZ)

1 Right-click **Transverse Shear Stress (SYZ)** and choose **Duplicate**.
2 In the **Settings** window for **1D Plot Group**, type `Transverse Shear Stress (SXZ)` in the **Label** text field.
3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `Transverse shear stress (SXZ), normalized`.
4 Locate the **Legend** section. From the **Position** list, choose **Center**.

Through Thickness 1

1 In the **Model Builder** window, expand the **Transverse Shear Stress (SXZ)** node, then click **Through Thickness 1**.
2 In the **Settings** window for **Through Thickness**, locate the **Data** section.
3 From the **Dataset** list, choose **Cut Point 3D: (B, A)**.

4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>SXZ_lw - Transverse shear stress (SXZ), normalized**.

5 Locate the **y-Axis Data** section. Find the **Interface positions** subsection. From the **Show interface positions** list, choose **None**.

Through Thickness 2

1 In the **Model Builder** window, click **Through Thickness 2**.

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

3 From the **Dataset** list, choose **Cut Point 3D: (B, A)**.

4 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>SXZ_esl - Transverse shear stress (SXZ), normalized**.

5 Locate the **y-Axis Data** section. Find the **Interface positions** subsection. From the **Show interface positions** list, choose **All interfaces**.

Table Graph 1

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

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

3 From the **Table** list, choose **Table: SXZ**.

Transverse Shear Stress (SXZ)

1 In the **Model Builder** window, click **Transverse Shear Stress (SXZ)**.

2 In the **Transverse Shear Stress (SXZ)** toolbar, click **Plot**.