Created in COMSOL Multiphysics 6.2

Ply Drop-off in a Composite Panel

Introduction

Ply drop-off often occurs in composite laminates to tailor the laminate thickness based on the loading requirements. This example illustrates the modeling of ply drop-off in a composite panel. The panel considered for the analysis has symmetric angle-ply layup with three sections–thick, taper, and thin. The thick section has sixteen plies divided into the core, top-bottom belts, and dropped plies. Out of sixteen plies in the thick section, eight plies are dropped gradually in the taper section with specified stagger distance.

The Carbon–Epoxy material having orthotropic material properties is used as a ply material. The Epoxy material having isotropic material properties is used in the pockets near dropped plies. The Layerwise theory based approach together with the Stack Zone modeling and variable thickness layer functionality is used for the detailed representation of the ply drop scenario. A stationary analysis is performed to compute stresses in different plies of various sections of the composite panel under the applied external load.

Model Definition

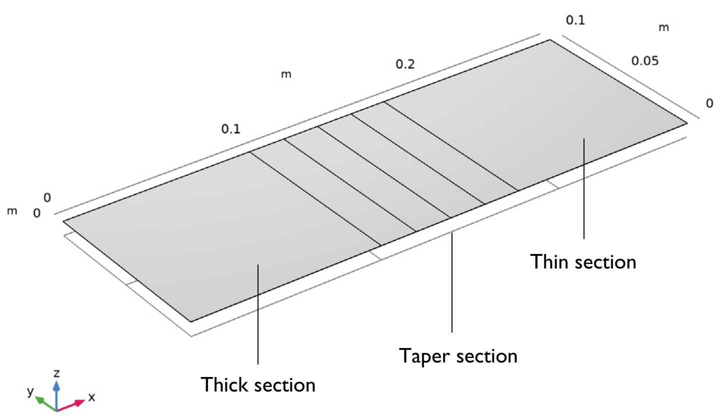

Figure 1: Model geometry of a composite panel with thick, thin, and taper sections having multiple ply drops in the taper section.

GEOMETRY

The model geometry of a composite panel having multiple ply drops is shown in Figure 1. The panel has three sections:

- Thick section
- Taper section
- Thin section

The thick section has 16 plies and some of them are dropped in the taper section. Finally there are 8 plies in the thin section of the composite panel.

The 3D representation of model geometry is shown in Figure 2. Note that geometry is scaled by a factor of 5 in the thickness direction for the visualization purpose.

BOUNDARY CONDITIONS

- Left end of the composite panel is fixed.
- A total load of 100 kN is applied to the right end of the panel in the form of a boundary load as shown in Figure 2.

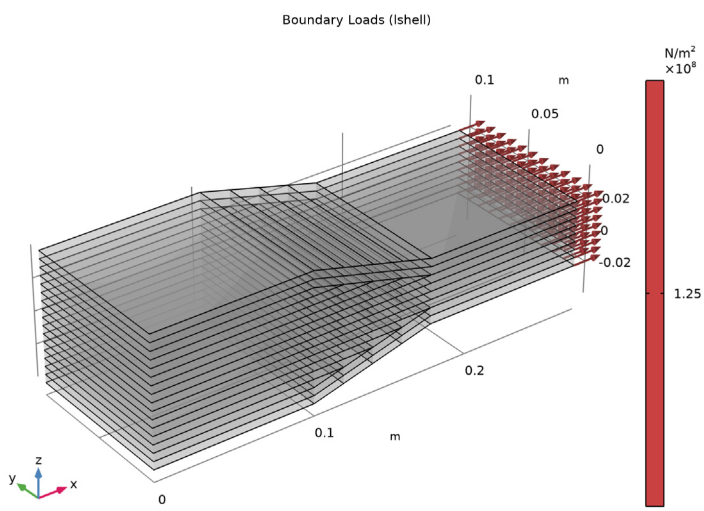

Figure 2: The 3D representation of model geometry together with the applied boundary load. Note that geometry is scaled by a factor of 5 in the thickness direction for the visualization purpose.

STACKING SEQUENCE

The laminate considered for the analysis has symmetric angle-ply layup. The stacking sequence of thick and thin sections are as follows:

- Thick section (16 plies): $[0/90/-30/30/-60/60/-45/45]_s$
- Thin section (8 plies): $[0/90/-45/45]_s$

The plies in the above half of the thick section can be divided in different segments as follows:

- Top belt: [0/90]
- Dropped plies: [-30/30/-60/60]
- Core: [-45/45]

The three sections of the composite panel and different ply segments in the thick section can be seen in Figure 5.

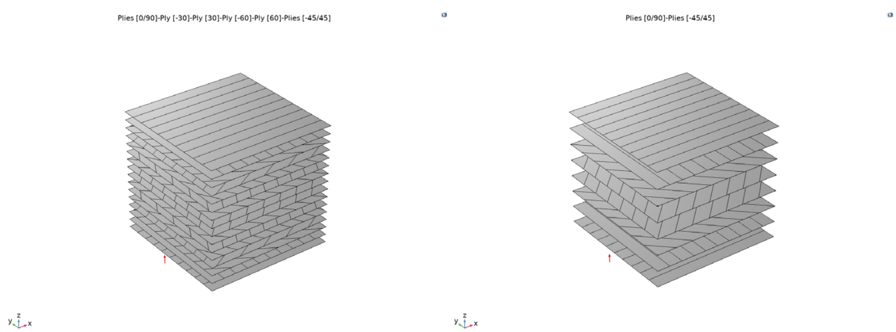

Figure 3: Stacking sequence in the thick and thin sections of the composite panel respectively.

MATERIAL PROPERTIES

Each ply of composite panel is assumed to be made of carbon fibers in an epoxy resin. The homogenized transversely isotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given below:

TABLE 1: MATERIAL PROPERTIES OF A PLY.

Material property	**Value**
$\{E_1, E_2\}$	{134, 9.2} GPa
$\{G_{12}\}$	{4.8} GPa
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.28, 0.28}

The resin pockets are assumed to be modeled using epoxy material. The material properties of resin is given below:

TABLE 2: EPOXY RESIN MATERIAL PROPERTIES.

Material property	Value
E	4 GPa
υ	0.35

TAPER SECTION MODELING

The taper section of the composite panel has multiple ply drops. It is assumed that the void created immediately after a ply drop is filled with same resin material. Total 8 plies are dropped in the entire taper section having 4 ply drops in above half of the laminate. The ply drops are placed in a staggered manner having only one ply drop at a time (in half of the laminate). This divides the entire taper section into 4 zones as shown in Figure 4.

The six zones in the Figure 4 corresponds to the three sections of panel as follows:

- Thick section: Zone 1
- Taper section: Zone 2–5
- Thin section: Zone 6

It can also be seen in Figure 4 that each taper zone has one ply in half of the laminate which is sectioned. This ply corresponds to a variable thickness ply representing resin pocket. Its thickness changes from left to right having thickness equal to ply thickness on the left end of the zone and close to zero thickness at the right end of the zone.

The stagger distance, in this example, is taken same as the resin pocket length. The resin pocket length is computed using a taper angle of 3 degrees. The resin pockets can be seen in yellow color in Figure 5. Figure also shows the resin pocket length and stagger distance.

The modeling of different sections/zones of the tapered composite laminate is achieved using Layered Material Stack node in combination with the Scale option for modeling variable thickness layers.

Once different zones are created by the Layered Material Stack node, a Continuity node is used in the physics to connect the layers of two respective zones sharing a common edge. One of the examples of the continuity condition between zone 2 and zone 3 is shown in Figure 6.

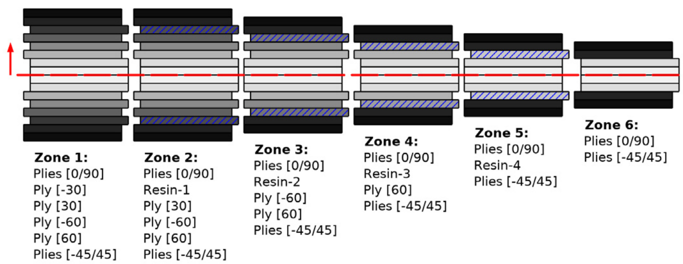

Figure 4: Stacking sequence in different zones of the composite panel. Here zone 1 corresponds to thick section, zone 6 corresponds to thin section, and rest of the zones correspond to taper section of the panel.

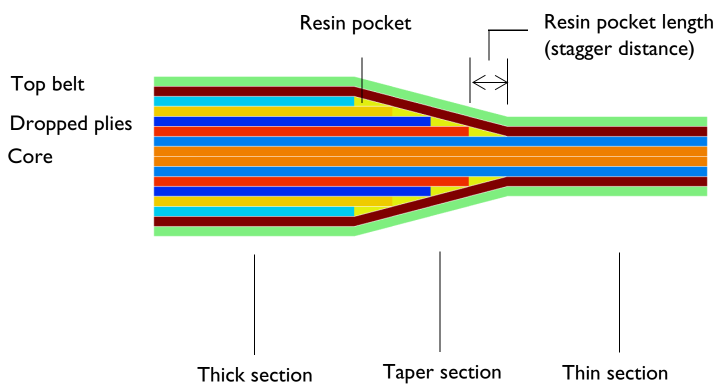

Figure 5: Cross-sectional view of the composite panel showing top belt, dropped plies, and core plies in the thick section. The resin pockets (yellow color) and stagger distance are also shown.

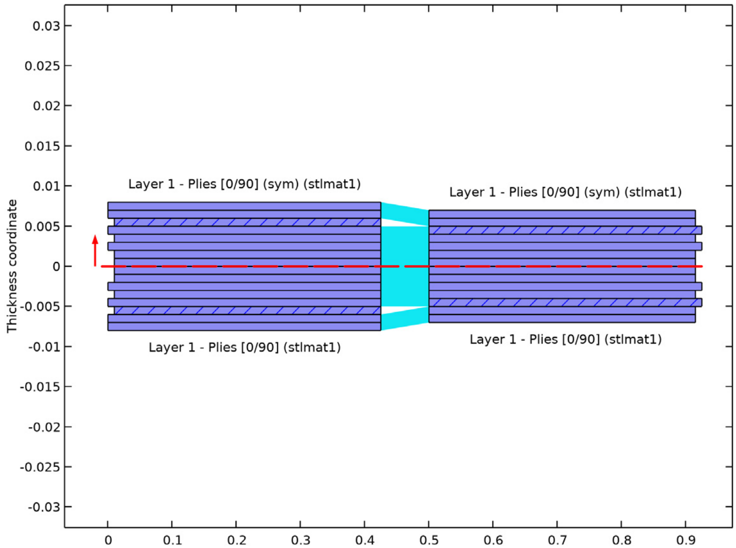

Figure 6: Continuity condition at the interface of zone 2 and zone 3 of the composite panel. The sectioned plies are the variable thickness resin plies having maximum thickness in the left of zone and zero thickness in the right of zone.

The laminate coordinate system showing the first principal direction and the total thickness of composite panel as a function of space is shown in Figure 7. First principal material direction showing the fiber orientation in each layer of the physical geometry is shown in Figure 8.

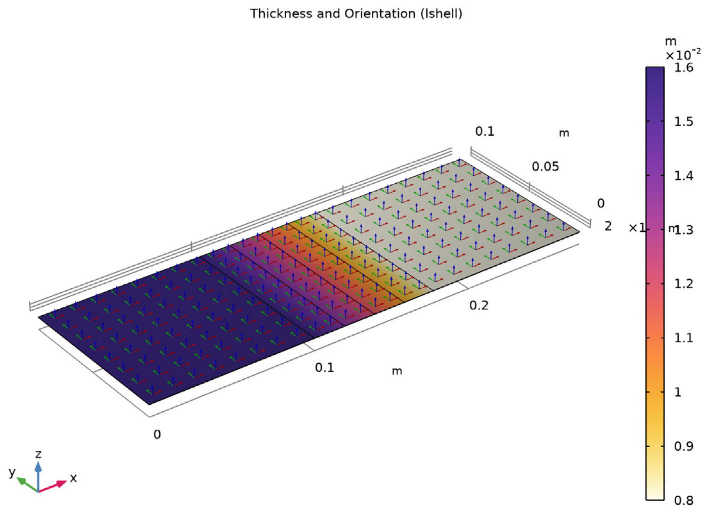

Figure 7: The laminate coordinate system showing the first principal direction and the total thickness of composite panel.

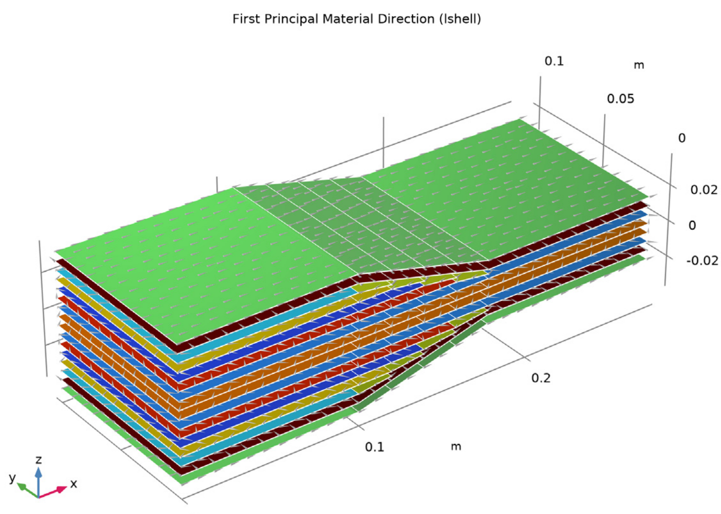

Figure 8: First principal material direction showing the fiber orientation in each layer of the physical geometry. Ply angle is used as a color for each layer.

Results and Discussion

The composite panel with symmetric angle-ply layup having multiple ply drops is subjected to an axial load. The distribution of effective (von Mises) stress for in each ply of all three sections is shown in Figure 9. It can be seen that the stress levels are quite different in each ply because of different fiber orientations. The region near the ply drops has slightly high stresses however they are still well below the stress levels at other locations. One of the reasons for not having very high stresses near the ply drops is the low taper angle and staggered approach of dropping the plies gradually.

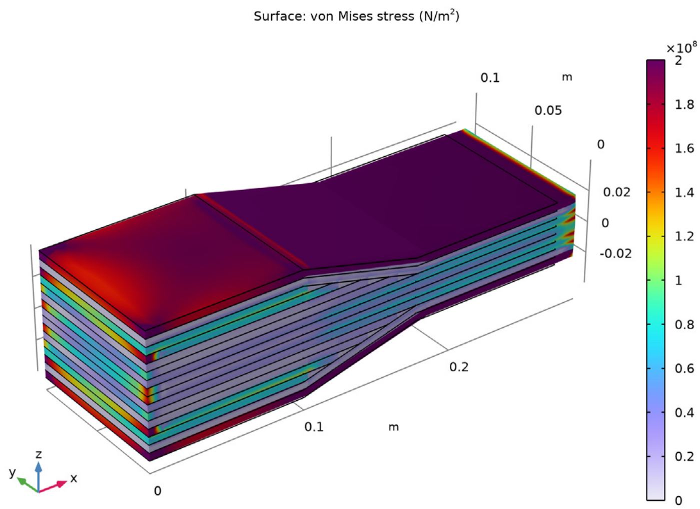

Figure 9: Von Mises stress distribution in various sections of the composite panel.

The von Mises stress distribution in the middle of each ply of the thick section of the composite panel is shown in Figure 10. Different stress distribution and maximum stress value can be seen in each ply of the thick section.

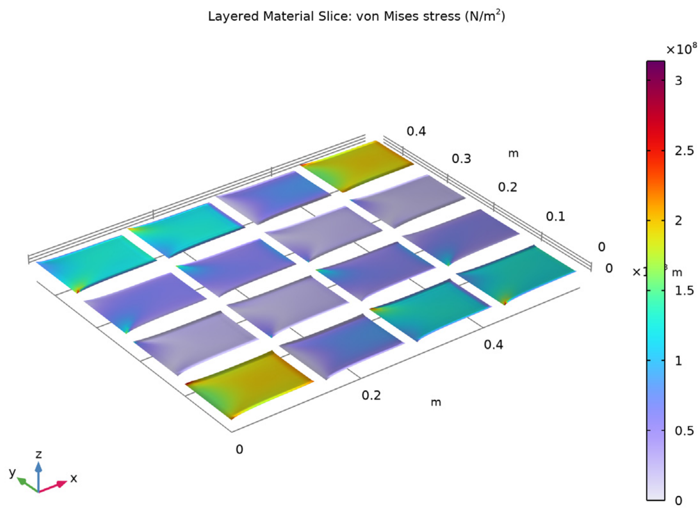

Figure 10: Von Mises stress distribution in each ply of the thick section of the composite panel.

Notes About the COMSOL Implementation

- The **Layered Material Stack** node is used to define various zones/sections of the tapered composite panel having thick, thin, and taper sections.
- At the ply drop location, variable thickness resin pockets are modeled as standard scaled layers using **Scale** option in the **Layered Material Link** node having thickness scale factor changing from 1 to 0 along the specified resin pocket length.
- The **Continuity** node is used in the physics to define the connection between the layers of two zones of the panel meeting side by side.

Application Library path: `Composite_Materials_Module/Tutorials/ply_drop_off_in_a_composite_panel`

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 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 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 `ply_drop_off_in_a_composite_panel_parameters.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.

Material: Resin

1 Right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Material: Resin` in the **Label** text field.

Plies [0/90]

1 Right-click **Materials** and choose **Layered Material**.
2 In the **Settings** window for **Layered Material**, locate the **Layer Definition** section.
3 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 1`	Material: Carbon-Epoxy (mat1)	`0.0`	`th`	`2`

4 Click + **Add**.

5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 2	Material: Carbon-Epoxy (mat1)	90.0	th	2

6 In the **Label** text field, type `Plies [0/90]`.

Ply [-30]

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

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

3 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 1	Material: Carbon-Epoxy (mat1)	-30.0	th	2

4 In the **Label** text field, type `Ply [-30]`.

5 Right-click **Ply [-30]** and choose **Duplicate**.

Ply [30]

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Ply [-30] 1 (lmat3)**.

2 In the **Settings** window for **Layered Material**, type `Ply [30]` 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: Carbon-Epoxy (mat1)	30.0	th	2

4 Right-click **Ply [30]** and choose **Duplicate**.

Ply [-60]

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Ply [30] 1 (lmat4)**.

2 In the **Settings** window for **Layered Material**, type `Ply [-60]` 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: Carbon-Epoxy (mat1)	-60.0	th	2

4 Right-click **Ply [-60]** and choose **Duplicate**.

Ply [60]

1 In the **Model Builder** window, click **Ply [-60] 1 (lmat5)**.

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

3 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 1`	Material: Carbon-Epoxy (mat1)	`60.0`	`th`	`2`

4 In the **Label** text field, type `Ply [60]`.

Plies [0/90] (lmat1)

In the **Model Builder** window, right-click **Plies [0/90] (lmat1)** and choose **Duplicate**.

Plies [-45/45]

1 In the **Model Builder** window, click **Plies [0/90] 1 (lmat6)**.

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

3 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 1`	Material: Carbon-Epoxy (mat1)	`-45.0`	`th`	`2`
`Layer 2`	Material: Carbon-Epoxy (mat1)	`45.0`	`th`	`2`

4 In the **Label** text field, type `Plies [-45/45]`.

Ply [60] (lmat5)

In the **Model Builder** window, right-click **Ply [60] (lmat5)** and choose **Duplicate**.

Resin

1 In the **Model Builder** window, click **Ply [60] 1 (lmat7)**.

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

3 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 1`	Material: Resin (mat2)	`20.0`	`th`	`2`

4 In the **Label** text field, type `Resin`.

Resin is an isotropic material and rotation here is set to 20 degree merely to distinguish it with 0 degree ply in a ply angle plot.

Create the geometry using specified resin pocket length.

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 `lr`.

4 In the **Height** text field, type `a`.

5 Locate the **Position** section. In the **xw** text field, type `a`.

Work Plane 1 (wp1)>Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

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

3 Select the object **r1** only.

4 In the **Model Builder** window, click **Array 1 (arr1)**.

5 In the **Settings** window for **Array**, locate the **Displacement** section.

6 In the **xw** text field, type `lr`.

7 Locate the **Size** section. In the **xw size** text field, type `4`.

Work Plane 1 (wp1)>Square 2 (sq2)

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`.

4 Locate the **Position** section. In the **xw** text field, type `a+4*lr`.

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

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

Now define plies and their selection in order to create thick, thin, and taper zones in the geometry. Use scale option for defining variable thickness resin pockets.

MATERIALS

Layered Material Stack 1 (stlmat1)

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

Plies [0/90]

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

2 Right-click **Plies [0/90]** and choose **Duplicate**.

Ply [-30]

1 In the **Model Builder** window, click **Plies [0/90] 1 (stlmat1.stllmat2)**.

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

3 From the **Material** list, choose **Ply [-30] (lmat2)**.

4 In the **Label** text field, type `Ply [-30]`.

5 Locate the **Boundary Selection** section. Click **Clear Selection**.

6 Select Boundary 1 only.

7 Right-click **Ply [-30]** and choose **Duplicate**.

Resin-1

1 In the **Model Builder** window, click **Ply [-30] 1 (stlmat1.stllmat3)**.

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

3 From the **Material** list, choose **Resin (lmat7)**.

4 Select the **Scale** check box. In the associated text field, type `1.0-0.99*(X-a)/lr`.

5 Locate the **Boundary Selection** section. Click **Clear Selection**.

6 Select Boundary 2 only.

7 In the **Label** text field, type `Resin-1`.

Ply [-30] (stlmat1.stllmat2)

In the **Model Builder** window, right-click **Ply [-30] (stlmat1.stllmat2)** and choose **Duplicate**.

Ply [30]

1 In the **Model Builder** window, click **Ply [-30] 1 (stlmat1.stllmat4)**.

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

3 From the **Material** list, choose **Ply [30] (lmat3)**.

4 Select Boundaries 1 and 2 only.

5 In the **Label** text field, type `Ply [30]`.

Resin-1 (stlmat1.stllmat3)

In the **Model Builder** window, right-click **Resin-1 (stlmat1.stllmat3)** and choose **Duplicate**.

Resin-2

1 In the **Model Builder** window, click **Resin-1.1 (stlmat1.stllmat5)**.

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

3 In the **Scale** text field, type `1.0-0.99*(X-a-lr)/lr`.

4 Locate the **Boundary Selection** section. Click **Clear Selection**.

5 Select Boundary 3 only.

6 In the **Label** text field, type `Resin-2`.

Ply [30] (stlmat1.stllmat4)

In the **Model Builder** window, right-click **Ply [30] (stlmat1.stllmat4)** and choose **Duplicate**.

Ply [-60]

1 In the **Model Builder** window, click **Ply [30] 1 (stlmat1.stllmat6)**.

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

3 From the **Material** list, choose **Ply [-60] (lmat4)**.

4 Select Boundaries 1–3 only.

5 In the **Label** text field, type `Ply [-60]`.

Resin-2 (stlmat1.stllmat5)

In the **Model Builder** window, right-click **Resin-2 (stlmat1.stllmat5)** and choose **Duplicate**.

Resin-3

1 In the **Model Builder** window, click **Resin-2.1 (stlmat1.stllmat7)**.

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

3 Click **Clear Selection**.

4 Select Boundary 4 only.

5 Locate the **Link Settings** section. In the **Scale** text field, type `1.0-0.99*(X-a-2*lr)/lr`.

6 In the **Label** text field, type `Resin-3`.

Ply [-60] (stlmat1.stllmat6)

In the **Model Builder** window, right-click **Ply [-60] (stlmat1.stllmat6)** and choose **Duplicate**.

Ply [60]

1 In the **Model Builder** window, click **Ply [-60] 1 (stlmat1.stllmat8)**.

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

3 From the **Material** list, choose **Ply [60] (lmat5)**.

4 Select Boundaries 1–4 only.

5 In the **Label** text field, type `Ply [60]`.

Resin-3 (stlmat1.stllmat7)

In the **Model Builder** window, right-click **Resin-3 (stlmat1.stllmat7)** and choose **Duplicate**.

Resin-4

1 In the **Model Builder** window, click **Resin-3.1 (stlmat1.stllmat9)**.

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

3 Click **Clear Selection**.

4 Select Boundary 5 only.

5 Locate the **Link Settings** section. In the **Scale** text field, type `1.0-0.99*(X-a-3*lr)/lr`.

6 In the **Label** text field, type `Resin-4`.

Plies [0/90] (stlmat1.stllmat1)

In the **Model Builder** window, right-click **Plies [0/90] (stlmat1.stllmat1)** and choose **Duplicate**.

Plies [-45/45]

1 In the **Model Builder** window, click **Plies [0/90] 1 (stlmat1.stllmat10)**.

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

3 From the **Material** list, choose **Plies [-45/45] (lmat6)**.

4 In the **Label** text field, type `Plies [-45/45]`.

Layered Material Stack 1 (stlmat1)

1 In the **Model Builder** window, click **Layered Material Stack 1 (stlmat1)**.

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

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

4 Click **Layer Stack Preview** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Layer Stack Preview**.

5 Click **Layer Cross-Section Preview** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Create Layer Cross-Section Plot**.

RESULTS

Layer Cross-Section Preview

1 In the **Model Builder** window, expand the **Results** node, then click **Layer Cross-Section Preview**.

2 In the **Layer Cross-Section Preview** toolbar, click **Plot**.

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 Select the **Transversely isotropic** 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
Young's modulus	{Evect1, Evect2}	{E1, E2}	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	{nu12, nu23}	1	Transversely isotropic
Shear modulus	Gvect1	{G12}	N/m^2	Transversely isotropic
Density	rho	1	kg/m^3	Basic

In order to define isotropic resin material properties, switch the solid model to isotropic. Later, switch it back to transversely isotropic and isotropic properties are automatically converted to orthotropic properties.

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 **Isotropic**.

GLOBAL DEFINITIONS

Material: Resin (mat2)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material: Resin (mat2)**.

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
Young's modulus	E	Er	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nur	1	Young's modulus and Poisson's ratio
Density	rho	1	kg/m³	Basic

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 **Orthotropic**.

Define continuity condition manually between different zones of the panel.

Continuity 1

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

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone1)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

5 In the **Selection** table, enter the following settings:

	Layered material	Offset (m)
√	Resin-1	0
√	Resin-1 (sym)	0

Continuity 2

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

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone3)**.

5 In the **Selection** table, enter the following settings:

	Layered material	Offset (m)
√	Resin-2	0
√	Resin-2 (sym)	0

6 Click **Layer Cross-Section Preview** in the upper-right corner of the **Layer Selection** section. From the menu, choose **Layer Cross-Section Preview**.

Continuity 3

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

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone3)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone4)**.

5 In the **Selection** table, enter the following settings:

	Layered material	Offset (m)
√	Resin-3	0
√	Resin-3 (sym)	0

Continuity 4

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

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone4)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone5)**.

5 In the **Selection** table, enter the following settings:

	Layered material	Offset (m)
√	Resin-4	0
√	Resin-4 (sym)	0

Continuity 5

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

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone5)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone6)**.

Fixed Constraint 1

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

2 Select Edge 1 only.

Boundary Load 1

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

2 Select Edge 19 only.

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

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

100[kN]	x
0	y
0	z

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**.

4 Click **Build All**.

STUDY 1

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

RESULTS

Stress (lshell)

In the **Model Builder** window, expand the **Stress (lshell)** node.

Surface 1

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

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

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

4 In the **Maximum** text field, type `2e8`.

5 In the **Stress (lshell)** toolbar, click **Plot**.

In order to have better visualization in thickness direction, set the scale factor to 5 in all the datasets.

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 `5`.

Stress (lshell)

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

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**, click to expand the **Selection** section.

2 From the **Geometric entity level** list, choose **Boundary**.

3 Select Boundary 1 only.

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

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 **Through-Thickness Location** section.

3 From the **Location definition** list, choose **Layer midplanes**.

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

5 In the **Relative x-separation** text field, type `0.15*3`.

Stress, Slice (lshell)

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

2 In the **Model Builder** window, click **Stress, Slice (lshell)**.

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

4 From the **View** list, choose **New view**.

5 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>Geometry and Layup (lshell)**.

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

RESULTS

Shell Geometry (lshell)

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

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

3 Click **Go to Source**.

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 5.

Shell Geometry (lshell)

1 In the **Model Builder** window, under **Results>Geometry and Layup (lshell)** click **Shell Geometry (lshell)**.
2 In the **Shell Geometry (lshell)** toolbar, click **Plot**.

Thickness and Orientation (lshell)

1 In the **Model Builder** window, click **Thickness and Orientation (lshell)**.
2 In the **Thickness and Orientation (lshell)** toolbar, click **Plot**.

First Principal Material Direction (lshell)

1 In the **Model Builder** window, click **First Principal Material Direction (lshell)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 Click **Go to Source**.

Layered Material 2 (Material Direction)

1 In the **Model Builder** window, under **Results>Datasets** 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 5.

First Principal Material Direction (lshell)

1 Click the **Go to Default View** button in the **Graphics** toolbar.
2 In the **Model Builder** window, under **Results>Geometry and Layup (lshell)** click **First Principal Material Direction (lshell)**.
3 In the **First Principal Material Direction (lshell)** toolbar, click **Plot**.

Geometry and Layup (lshell)

In the **Model Builder** window, collapse the **Results>Geometry and Layup (lshell)** node.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.
2 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Applied Loads (lshell)**.

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

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

RESULTS

Boundary Loads (Ishell)

1 In the **Model Builder** window, under **Results>Applied Loads (Ishell)** click **Boundary Loads (Ishell)**.

2 In the **Boundary Loads (Ishell)** toolbar, click **Plot**.

Stress (Ishell)

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