Model created in COMSOL Multiphysics 6.4

Buckling of a Composite Cylinder

Introduction

Buckling is a structural instability that can lead to failure of a component even without initial material failure. Computation of the critical buckling loads and mode shapes can therefore be important from a design viewpoint, even though it has previously been determined that the loading of the component only causes elastic deformations. This applies to components made from laminated composite materials, where elastic properties, ply thicknesses and stacking sequence of a composite laminate will affect buckling loads and mode shapes.

This example illustrates a linear buckling analysis of a composite cylinder under compressive loading and fixed-end conditions. The composite cylinder is made up of eight layers (plies) of a carbon fiber reinforced epoxy material having different fiber orientations. An Equivalent Single Layer (ESL) theory based approach is used for this analysis. The effect of stacking sequence on the critical load factor is analyzed for different types of balanced laminates, such as a symmetric angle-ply laminate and an antisymmetric angle-ply laminate.

Read more about the Composite Materials Module in the COMSOL blog, Introduction to the Composite Materials Module.

Model Definition

The model geometry consists of a composite cylinder with height of 0.4 m and radius of 0.15 m. The bottom end of the cylinder is fixed whereas the top end is free only to translate in the *z* direction, as shown in Figure 1. In order to perform a linear buckling analysis, a unit compressive load is applied on the top end of the cylinder in the downward direction.

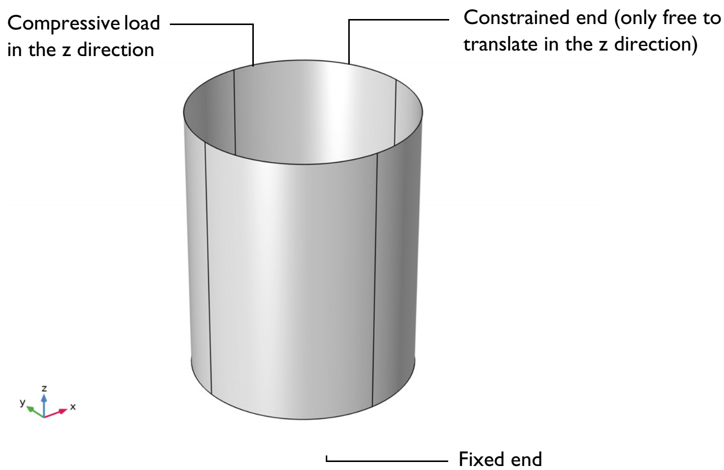

Figure 1: Model geometry of the laminated composite cylinder.

LAMINA MATERIAL PROPERTIES

The composite lamina 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 in Table 1.

TABLE 1: MATERIAL PROPERTIES OF A LAMINA.

Material property	**Value**
$\{E_{11}, E_{22}\}$	{134, 9.2} GPa
G_{12}	4.8 GPa
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.28, 0.28}

The density of the lamina is taken as 1700 kg/m^3.

STACKING SEQUENCE

The composite laminate consists of eight layers where each layer (ply) has a thickness of 0.125 mm. This makes the total thickness of the composite laminate 1 mm. In order to study the effect of a stacking sequence on the buckling behavior of the composite cylinder, four different laminates are compared:

- Layered Material 1: $[0/0/45/\text{-}45]_s$ (Symmetric angle-ply laminate)

- Layered Material 2: [90/90/45/-45]$_s$ (Symmetric angle-ply laminate)
- Layered Material 3: [90/0/90/0]$_s$ (Symmetric cross-ply laminate)
- Layered Material 4: [45/45/45/45]$_{as}$ (Antisymmetric angle-ply laminate)

The stacking sequence of each laminate is shown in Figure 2 and their 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.

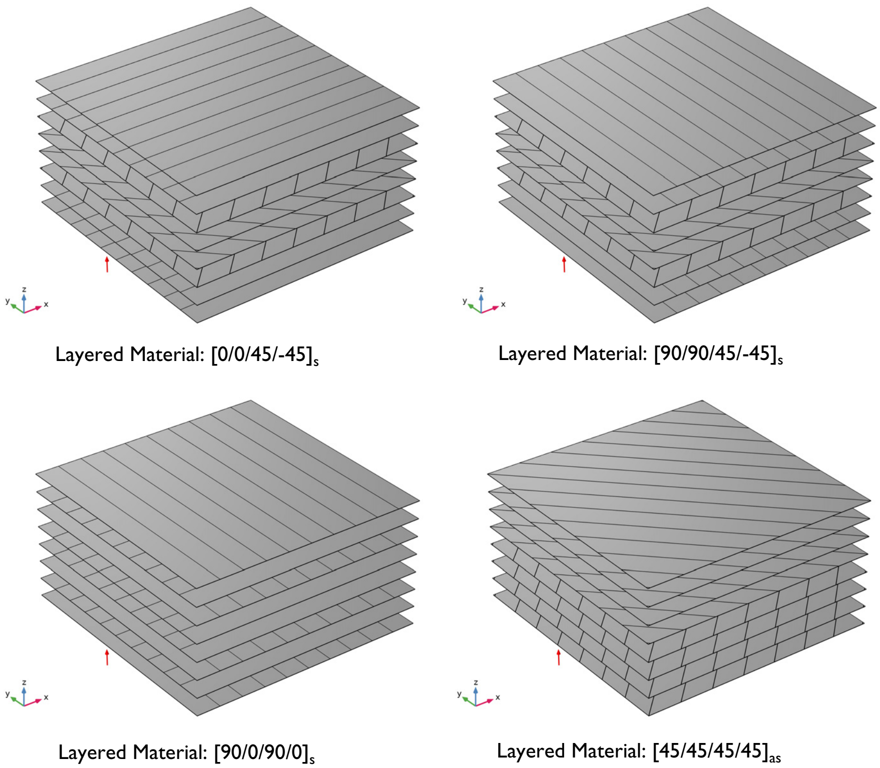

Figure 2: Stacking sequence of laminated composite cylinder showing fiber orientation in each layer from bottom to top.

TABLE 2: STACKING SEQUENCES CONSIDERED.

Layer Number	Fiber orientation in Layered material 1 (°)	Fiber orientation in Layered material 2 (°)	Fiber orientation in Layered material 3 (°)	Fiber orientation in Layered material 4 (°)
1	0	90	90	45
2	0	90	0	45
3	45	45	90	45
4	-45	-45	0	45
5	-45	-45	0	-45
6	45	45	90	-45
7	0	90	0	-45
8	0	90	90	-45

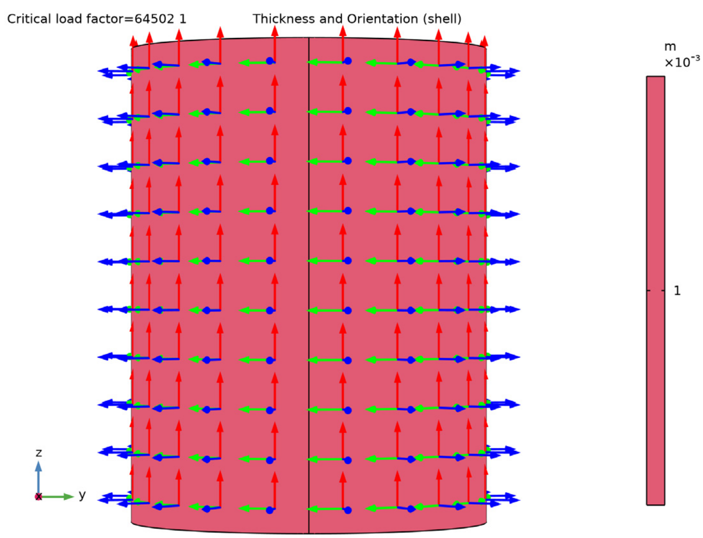

Figure 3: The laminate coordinate system showing the first principal direction along the cylinder axis.

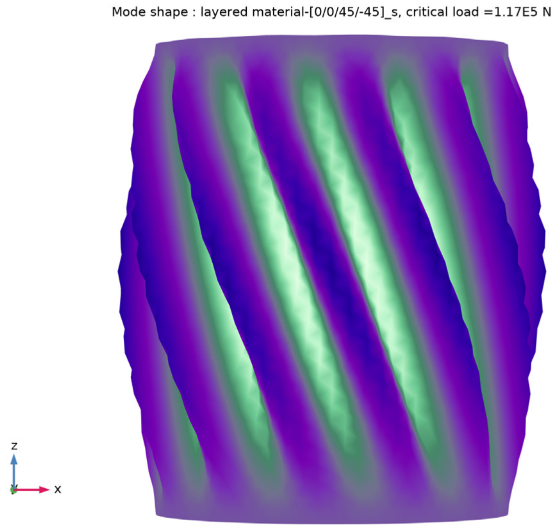

Figure 4: First buckling mode shape and its corresponding critical load for Layered Material: [0/0/45/-45]_s.

The buckling analysis of a composite laminate with different stacking sequences shows that the critical buckling load and its corresponding mode shape is highly dependent on the stacking sequence of the individual laminae (plies).

For the first two stacking sequences, where the symmetric angle-ply arrangement is used, the first buckling mode is spiral-shaped as shown in Figure 4 and Figure 5. A notable difference between the two is the pitch of the spiraling.

For the third stacking sequence, where the symmetric cross-ply arrangement is used, the buckling mode is diamond-shaped, as shown in Figure 6. Last, for the fourth stacking sequence, in which an antisymmetric angle-ply arrangement is used, the buckling mode shape is axisymmetric, as shown in Figure 7.

TABLE 3: CRITICAL BUCKLING LOAD FOR DIFFERENT LAMINATES.

Stacking sequence	**Type of laminate**	**Critical buckling load (kN)**	**Mode shape**
[0/0/45/-45]_s	Symmetric angle-ply	117	Spiral
[90/90/45/-45]_s	Symmetric angle-ply	101.52	Spiral

TABLE 3: CRITICAL BUCKLING LOAD FOR DIFFERENT LAMINATES.

Stacking sequence	**Type of laminate**	**Critical buckling load (kN)**	**Mode shape**
[90/0/90/0]_s	Symmetric cross-ply	92.13	Diamond
[45/45/45/45]_as	Antisymmetric angle-ply	64.50	Axisymmetric

The critical buckling loads for all the stacking sequences are listed in Table 3. The first stacking sequence, where the symmetric angle-ply arrangement is used and in which four out of eight ply angles are zero, has the highest critical load factor. Not surprisingly, the fourth stacking sequence, where an antisymmetric angle-ply arrangement is used, has the lowest critical load factor.

Interestingly, by *only* changing the stacking sequence from the fourth to the first, the critical buckling load can be increased by a factor of about two in the present model.

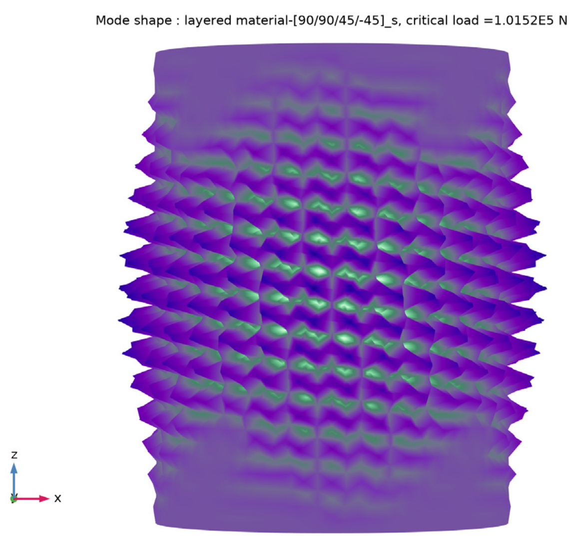

Figure 5: First buckling mode shape and its corresponding critical load for Layered Material: [90/90/45/-45]_s.

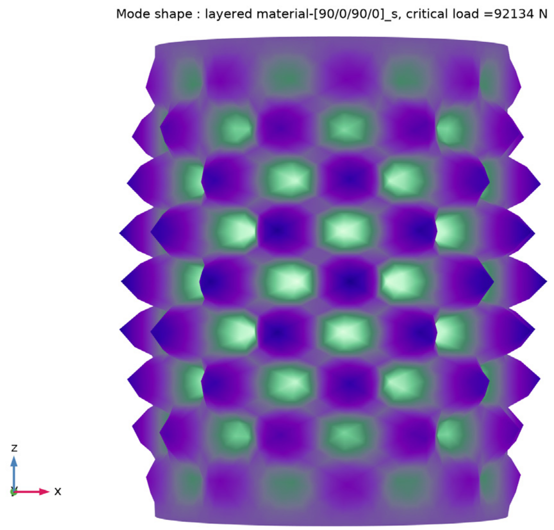

Figure 6: First buckling mode shape and its corresponding critical load for Layered Material: [90/0/90/0]_s.

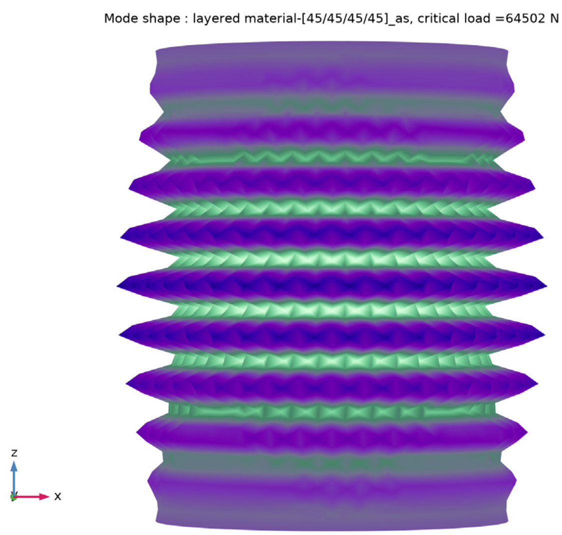

Figure 7: First buckling mode shape and its corresponding critical load for Layered Material: [45/45/45/45]_as.

Notes About the COMSOL Implementation

- Modeling a composite laminate as a layered shell requires a surface geometry, in general referred to as 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 the number of through-thickness mesh elements for each layer.
- The third direction for the selected coordinate system in the **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack** represents the normal direction of the **Layered Shell** or **Shell** physics. This is also the direction in which the layer stacking is interpreted from bottom to top, and therefore, it is crucial to know it during modeling. There are two ways to achieve this:
  - Using physics symbols: Go to the physics settings, find the **Physics Symbols** section, and select the **Enable physics symbols** checkbox. Then go to the material feature, for instance, **Linear Elastic Material**, to see the normal direction represented by green arrows in the geometry.
  - Using result templates: When a solution dataset is available, use the result template **Thickness and Orientation** to plot the normal direction.
- From a constitutive model point of view, you can either use the *Layerwise (LW)* theory based **Layered Shell** interface, or the *Equivalent Single Layer (ESL)* theory based **Linear Elastic Material, Layered** node in the **Shell** interface. The laminated composite presented in the current model is modeled using a **Linear Elastic Material, Layered** node in the **Shell** interface.
- The built-in **Composites** material library contains data for fiber and matrix constituents as well as for unidirectional and bidirectional laminae.
- In order to perform a buckling analysis, a special **Linear Buckling** study is used. This consists of a **Stationary** study step and a **Linear Buckling** study step. The stationary study step performs the stress analysis for the applied load whereas buckling study uses eigenvalue solver and computes the critical load factors for the applied load.
- In order to run the analysis for various layered materials and compare the results, all the layered materials can be defined using a **Switch** node in **Global Materials**. This **Switch** node can be selected in the **Layered Material Link** node and a **Material Sweep** node is added in the study. Note that while performing material sweep over layered materials in a prestressed buckling analysis, prestress stationary study step results are stored only for the last layered material.

Application Library path: `Composite_Materials_Module/Buckling/composite_cylinder_buckling`

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** > **Shell (shell)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Linear Buckling**.
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 In the table, enter the following settings:

Name	Expression	Value	Description
r	0.15[m]	0.15 m	Cylinder radius
l	0.4[m]	0.4 m	Cylinder length

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 In the table, enter the following settings:

Name	Expression	Unit	Description
Fc	shell.LFcrit*1[N]		Critical buckling load
un	u*nX+v*nY+w*nZ	m	Normal displacement

GEOMETRY 1

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Object Type** section.

3 From the **Type** list, choose **Surface**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `r`.

5 In the **Height** text field, type `l`.

6 Click **Build Selected**.

You may want to import material data from a different file and use it while modeling. In the present example, the material properties are loaded from the file `composite_cylinder_buckling_material.mph` stored in the model's Application Libraries folder.

MATERIALS

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

MATERIAL BROWSER

1 In the **Material Browser** window, click **Import Material Library**.

2 Browse to the model's Application Libraries folder and double-click the file `composite_cylinder_buckling_material.mph`.

3 Click **Done**.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **composite cylinder buckling material** > **Layered Material: [0/0/45/-45]_s**.

4 Click the **Add to Global Materials** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

GLOBAL DEFINITIONS

Layered Material: [0/0/45/-45]_s (lmat1)

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

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

Material Switch 1 (sw1)

In the **Model Builder** window, right-click **Materials** and choose **Material Switch**.

Drag the **Layered Material: [0/0/45/-45]_s** node to **Material Switch 1** node.

Layered Material: [0/0/45/-45]_s (sw1.lmat1)

Drag and drop **Layered Material: [0/0/45/-45]_s (lmat1)** on **Material Switch 1 (sw1)**.

Layered Material: [90/90/45/-45]_s

1 Right-click **Layered Material: [0/0/45/-45]_s (sw1.lmat1)** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material**, type `Layered Material: [90/90/45/-45]_s` in the **Label** text field.

3 Find the **Layer Definition** section and change the rotation angles in the **Rotation** column as summarized in the table below.

Layer	Rotation
Layer 1	90
Layer 2	90
Layer 3	45
Layer 4	-45
Layer 5	-45
Layer 6	45
Layer 7	90
Layer 8	90

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

Layered Material: [90/0/90/0]_s

1 Right-click **Layered Material: [90/90/45/-45]_s** and choose **Duplicate**.

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

3 Find the **Layer Definition** section and change the rotation angles in the **Rotation** column as summarized in the table below.

Layer	Rotation
Layer 1	90
Layer 2	0
Layer 3	90
Layer 4	0
Layer 5	0
Layer 6	90
Layer 7	0
Layer 8	90

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

Layered Material: [45/45/45/45]_as

1 Right-click **Layered Material: [90/0/90/0]_s** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material**, type `Layered Material: [45/45/45/45]_as` in the **Label** text field.

3 Find the **Layer Definition** section and change the rotation angles in the **Rotation** column as summarized in the table below.

Layer	Rotation
Layer 1	45
Layer 2	45
Layer 3	45
Layer 4	45
Layer 5	-45
Layer 6	-45
Layer 7	-45
Layer 8	-45

4 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**.

SHELL (SHELL)

Linear Elastic Material, Layered 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Linear Elastic Material, Layered**.
2 In the **Settings** window for **Linear Elastic Material, Layered**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 Locate the **Linear Elastic Material** section. From the **Material symmetry** list, choose **Orthotropic**.
5 Select the **Transversely isotropic** checkbox.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.
2 Select Edges 2, 3, 7, and 10 only.

Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement/Rotation**.
2 Select Edges 4, 5, 8, and 11 only.
3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.
4 From the **Displacement in x direction** list, choose **Prescribed**.
5 From the **Displacement in y direction** list, choose **Prescribed**.
6 Locate the **Prescribed Rotation** section. From the **By** list, choose **Rotation**.

Edge Load 1

1 In the **Physics** toolbar, click **Edges** and choose **Edge Load**.
2 Select Edges 4, 5, 8, and 11 only.
3 In the **Settings** window for **Edge Load**, locate the **Force** section.
4 From the **Load type** list, choose **Total force**.

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

0	x
0	y
-1[N]	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 In the table, clear the **Use** checkbox for **Geometric Analysis, Detail Size**.

5 Click **Build All**.

STUDY 1

In the **Model Builder** window, collapse the **Study 1** node.

Material Sweep

1 In the **Study** toolbar, click **More Study Extensions** and choose **Material Sweep**.

2 In the **Settings** window for **Material Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the **Study** toolbar, click **Compute**.

RESULTS

Mode Shape: [0/0/45/-45]_s

1 Click the **Go to XZ View** button in the **Graphics** toolbar.

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

Use the following instructions to plot the first critical buckling mode shape of the first laminate as shown in Figure 4.

3 In the **Settings** window for **3D Plot Group**, type `Mode Shape: [0/0/45/-45]_s` in the **Label** text field.

4 Locate the **Data** section. From the **Material Switch 1** list, choose **Layered Material: [0/0/45/-45]_s**.

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

6 In the **Title** text area, type `Mode shape : layered material-[0/0/45/-45]_s, critical load =eval(lambda) N`.

7 Clear the **Parameter indicator** text field.

8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 In the **Model Builder** window, expand the **Mode Shape: [0/0/45/-45]_s** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

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

4 In the **Mode Shape: [0/0/45/-45]_s** toolbar, click **Plot**.

Follow the instructions below to plot the first critical buckling mode shape of the second laminate as shown in Figure 5.

Mode Shape: [90/90/45/-45]_s

1 In the **Model Builder** window, right-click **Mode Shape: [0/0/45/-45]_s** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape: [90/90/45/-45]_s` in the **Label** text field.

3 Locate the **Data** section. From the **Material Switch 1** list, choose **Layered Material: [90/90/45/-45]_s**.

4 Locate the **Title** section. In the **Title** text area, type `Mode shape : layered material-[90/90/45/-45]_s, critical load =eval(lambda) N`.

Follow the instructions below to plot the first critical buckling mode shape of the third laminate as shown in Figure 6.

Mode Shape: [90/0/90/0]_s

1 Right-click **Mode Shape: [90/90/45/-45]_s** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape: [90/0/90/0]_s` in the **Label** text field.

3 Locate the **Data** section. From the **Material Switch 1** list, choose **Layered Material: [90/0/90/0]_s**.

4 Locate the **Title** section. In the **Title** text area, type `Mode shape : layered material-[90/0/90/0]_s, critical load =eval(lambda) N`.

5 In the **Mode Shape: [90/0/90/0]_s** toolbar, click **Plot**.

Follow the instructions below to plot the first critical buckling mode shape of the fourth laminate as shown in Figure 7.

Mode Shape: [45/45/45/45]_as

1 Right-click **Mode Shape: [90/0/90/0]_s** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Mode Shape: [45/45/45/45]_as` in the **Label** text field.
3 Locate the **Data** section. From the **Material Switch 1** list, choose **Layered Material: [45/45/45/45]_as**.
4 Locate the **Title** section. In the **Title** text area, type `Mode shape : layered material-[45/45/45/45]_as, critical load =eval(lambda) N`.
5 In the **Mode Shape: [45/45/45/45]_as** toolbar, click **Plot**.

Follow the instructions below to compare the buckling mode shape for all four laminates.

Mode Shape: Comparison

1 Right-click **Mode Shape: [45/45/45/45]_as** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Mode Shape: Comparison` in the **Label** text field.
3 Locate the **Title** section. From the **Title type** list, choose **Label**.
4 Find the **Layout** subsection. Clear the **Use parameter indicator for solution and phase** checkbox.
5 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.
6 From the **Array plane** list, choose **yz**.
7 In the **Relative column padding** text field, type `0.5`.

Surface 1

1 In the **Model Builder** window, expand the **Mode Shape: Comparison** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Layered Material**.
4 From the **Material Switch 1** list, choose **Layered Material: [0/0/45/-45]_s**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Scale** section.
3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Solution Array 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Solution Array**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

3 From the **Critical load factor selection** list, choose **First**.

4 Locate the **Plot Array** section. From the **Array shape** list, choose **Square**.

Table Annotation 1

1 In the **Model Builder** window, right-click **Mode Shape: Comparison** 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	z-coordinate	Annotation
0	0	-0.15*l	[0/0/45/-45]_s
0	1.3*l	-0.15*l	[90/90/45/-45]_s
0	0	-0.15*l+1.3*l	[90/0/90/0]_s
0	1.3*l	-0.15*l+1.3*l	[45/45/45/45]_as

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 From the **Anchor point** list, choose **Lower middle**.

Mode Shape: Comparison

1 In the **Model Builder** window, click **Mode Shape: Comparison**.

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

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

4 Click the **Go to YZ View** button in the **Graphics** toolbar.

5 In the **Mode Shape: Comparison** toolbar, click **Plot**.

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

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Thickness and Orientation (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Thickness and Orientation (shell)

In the **Thickness and Orientation (shell)** toolbar, click **Plot**.

Use an **Evaluation Group** instead of **Derived Values** nodes to compute the critical buckling load.

Critical Buckling Load

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Critical Buckling Load` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

Global Evaluation 1

1 Right-click **Critical Buckling Load** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
Fc	kN	Critical buckling load

4 In the **Critical Buckling Load** toolbar, click **Evaluate**.

Enable automatic reevaluation of evaluation groups when the model is re-solved.

5 In the **Model Builder** window, click **Results**.

6 In the **Settings** window for **Results**, locate the **Update of Results** section.

7 Select the **Reevaluate all evaluation groups after solving** checkbox.