Created in COMSOL Multiphysics 6.2

Forced Vibration Analysis of a Composite Laminate

Introduction

Delamination or the separation of layers is a common failure mode in laminated composite materials. Various factors, including loading, defects in the material, and environmental conditions can trigger the initiation and propagation of layer separation. This leads to degraded structural performance and sometimes even complete failure of the structure.

This example considers the response of a delaminated plate under forced vibration. The composite plate is analyzed for two different locations of delamination and compared with the intact plate, without delamination. The plate is made of three layers with [90/45/0] stacking. The delamination is assumed to occur in a circular or semi-circular region between the second and third layers of the laminate and modeled using the layerwise theory.

The frequency response analysis of the composite plate is performed under bending as well as twisting loads and the response of the plate in terms of impedance and transverse velocity is measured at different points on the plate.

Model Definition

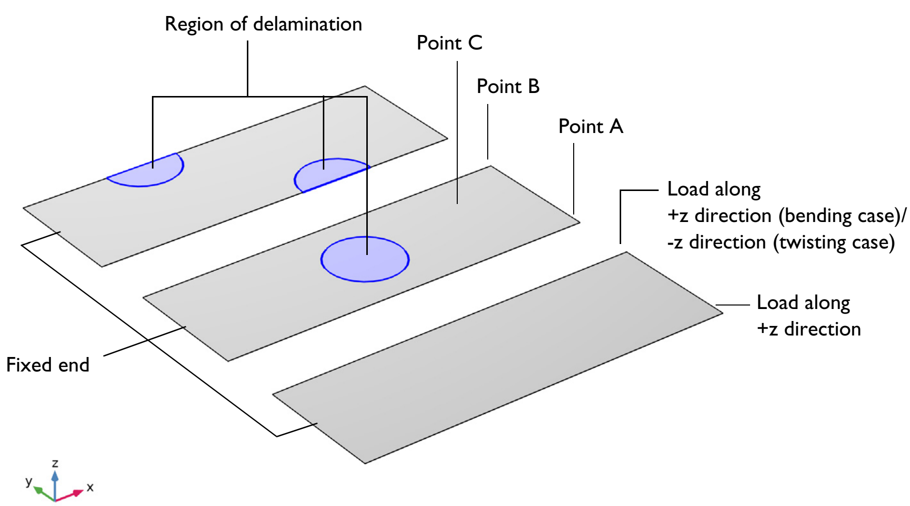


Figure 1: Model geometry showing the boundary conditions, loading conditions, and regions of delamination.

The model geometry consists of a rectangular plate of 300 mm length and 100 mm width. In order to study the effect of delamination on the dynamics of the structure, three different cases are considered. In the first case, the structure is assumed intact with no delamination. The second and third cases model the structure with delamination occurring at different locations. For the second case, the region of delamination is chosen as a circle of 60 mm diameter at the center of the plate, and the regions of delamination in third case are two semicircles with the same diameter, as shown in Figure 1.

Material properties, boundary conditions, and loading for the three plates are the same. The left end of each plate is fixed. Two transverse unit loads are applied to each plate at the right end, as shown in Figure 1.

Frequency domain analyses under two different loading conditions are used to understand the dynamic behavior:

- Bending load: The first analysis is carried out for a bending load case in which both the end points are subjected to a unit load in the positive z direction.
- Twisting load: The second analysis is performed for a twisting load case in which the unit load is applied in the positive z direction at one point and in the negative z direction at the other.

LAMINA MATERIAL PROPERTIES

The lamina is made of carbon fibers in an epoxy resin. The density of lamina is taken as 1520 kg/m^3 and the components of the elasticity matrix for the material is shown in Table 1.

TABLE 1: MATERIAL PROPERTIES OF A LAMINA.

Elasticity Matrix Components	**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}

THICKNESS AND STACKING SEQUENCE

The laminate consists of three layers of 0.5 mm thickness. The orientation of fibers in each layer, starting from the bottom, is taken as 90, 45, 0 degrees, as shown in Figure 2.

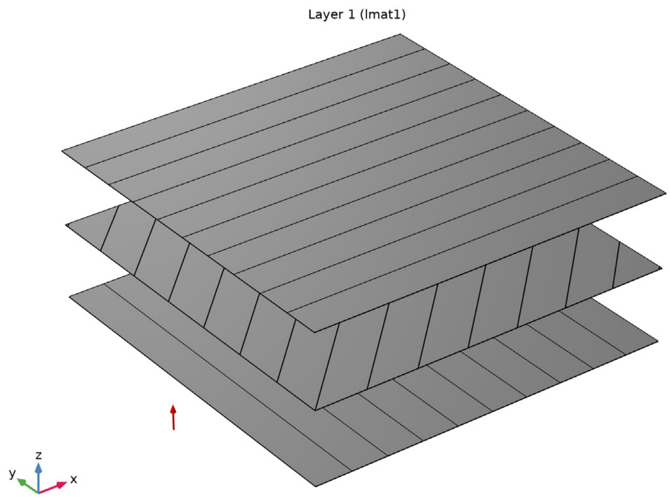


Figure 2: Stacking sequence of the plate, showing the fiber orientation in each layer.

In this example, it is assumed that the layers are perfectly bonded together. The delamination or separation of layers happens at the interface between layers two and three, as shown in Figure 3. The delamination of the interface between layers two and three is modeled using the **Delamination** feature available in the Layered Shell interface.

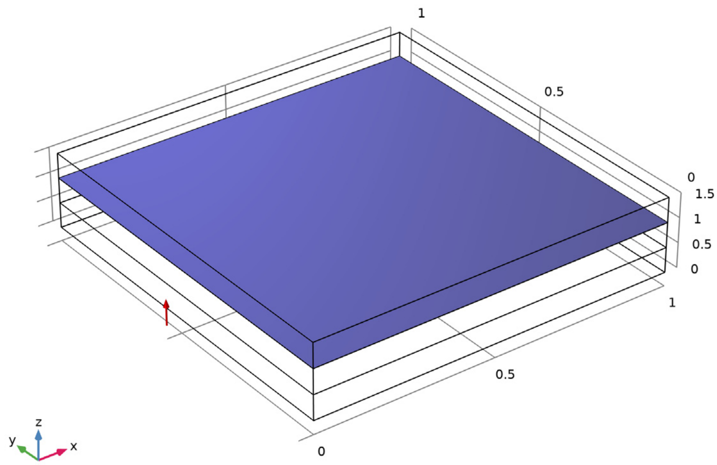


Figure 3: Schematic representation of the delaminated interface between layers two and three.

Results and Discussion

Figure 4 shows the distribution of von Mises stress in all three plates for the case of the bending load. The through-thickness variation of von Mises stress at a position along the fixed edge is shown in Figure 5. The plot shows that the stress distribution for the intact plate (the first case) and the plate delaminated at the center (the second case) are similar, but notably different from the stress distribution for the plate delaminated at the edges (the third case).

The impedance at any point in the structure under forced vibration is calculated as the ratio of the force and the velocity of that point. Figure 6 and Figure 7 show the impedance amplitude at the two points (Point A and Point B) located on the right end of the plates, where the unit loads are applied, see Figure 1. The transverse velocity at a point (Point C) located inside the plate is shown in Figure 8.

At lower frequencies, the impedance of the three plates matches very well. At higher frequencies, the impedance of the delaminated plates starts to deviate the impedance of the intact plate. This suggests that nondestructive testing (NDT) techniques can be used to detect the delamination by measuring the structural impedance using high frequency waves.

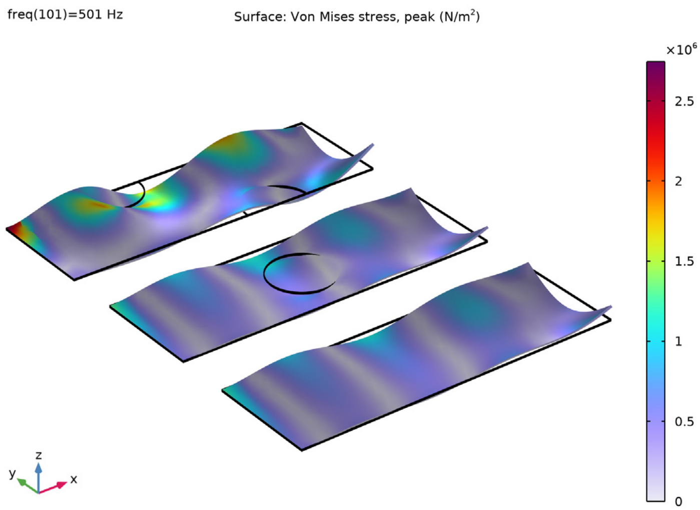


Figure 4: von Mises stress distribution for the bending load, at frequency = 500 Hz.

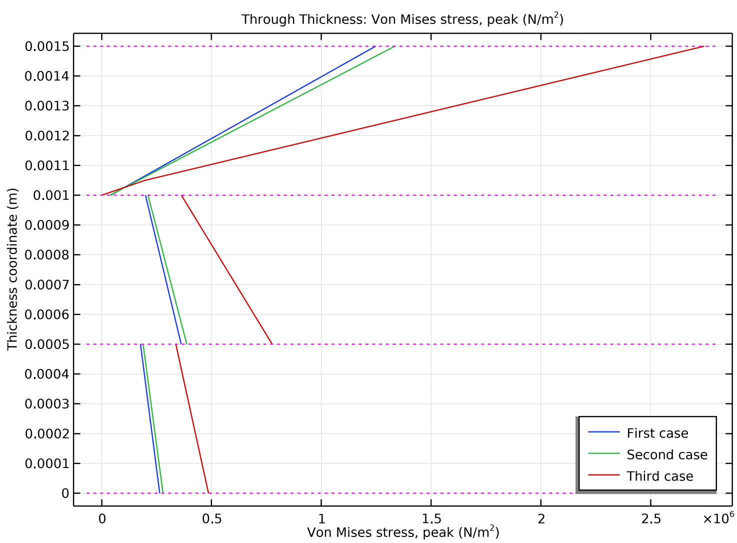


Figure 5: Through-thickness variation of von Mises stress for the bending load, at frequency 500 Hz.

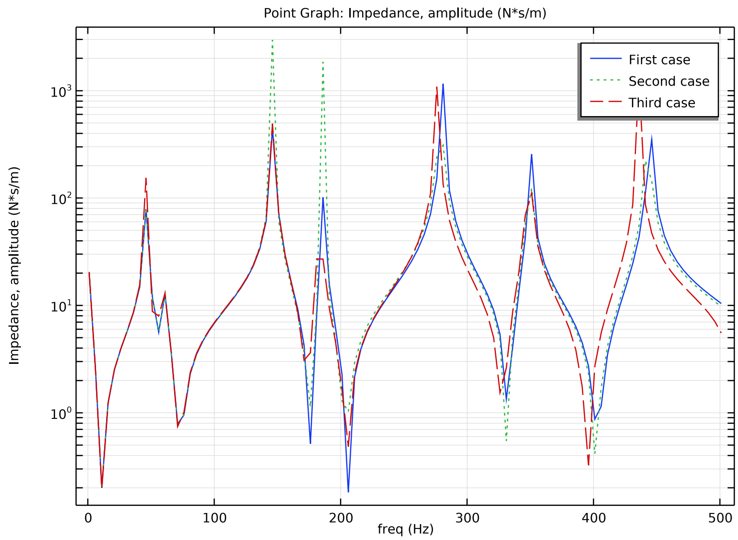


Figure 6: Impedance amplitude at Point A for the bending load.

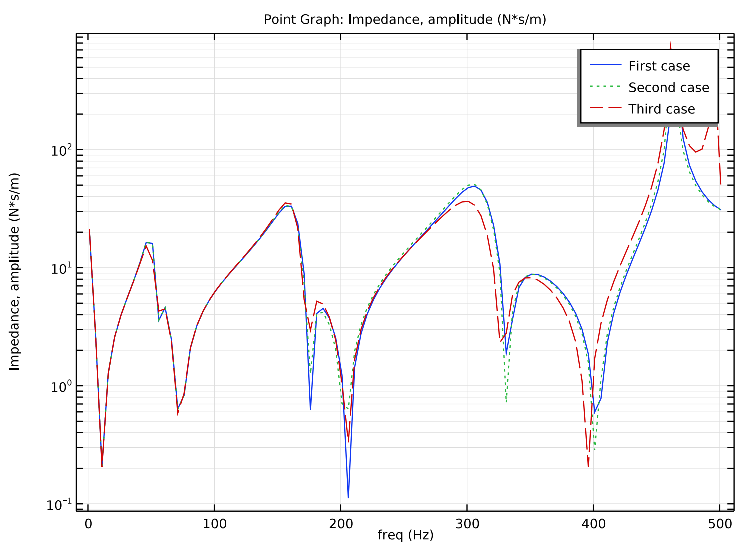


Figure 7: Impedance amplitude at Point B for the bending load.

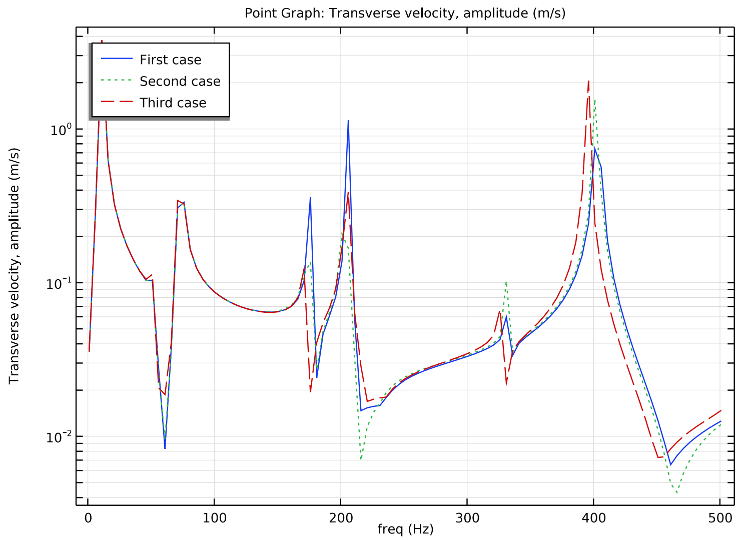


Figure 8: Transverse velocity amplitude at Point C for the bending load.

Figure 9 shows the distribution of von Mises stress in the three plates under twisting load. The through-thickness variation of von Mises at a point along the fixed edges of the plates is shown in Figure 10.

As in the bending case, it can be seen from the plot that the stress distribution for the intact plate (first case) and the plate delaminated at the center (second case) are similar but different from the stress distribution for the plate delaminated at the edges (third case).

Figure 11 and Figure 12 show the amplitude of impedance at Point A and Point B for the twisting load case. The transverse velocity at a point located inside the plates (Point C) is shown in Figure 13.

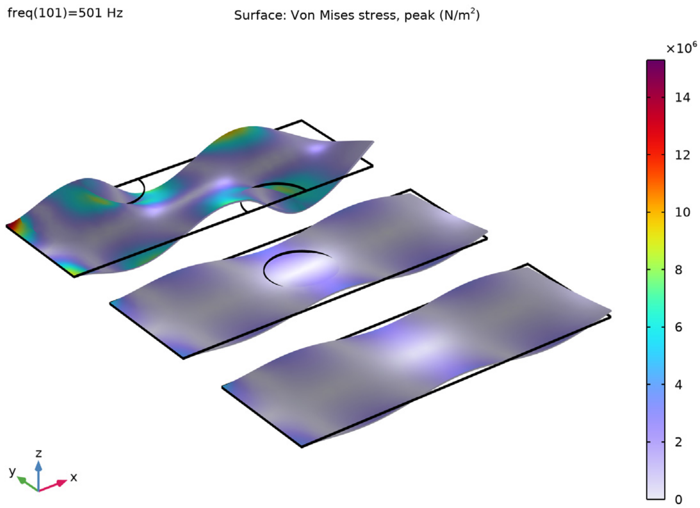


Figure 9: von Mises stress distribution in the plate for the twisting load, at frequency = 500 Hz.

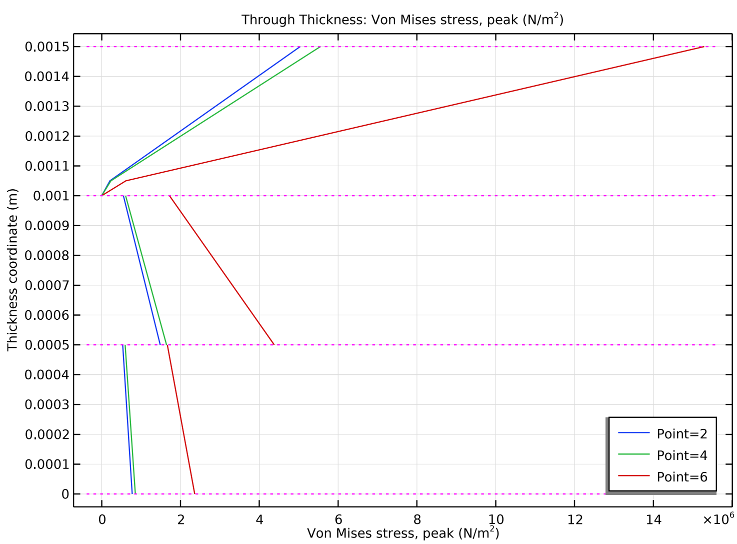


Figure 10: Through-thickness variation of von Mises stress for the twisting load, at frequency = 500 Hz.

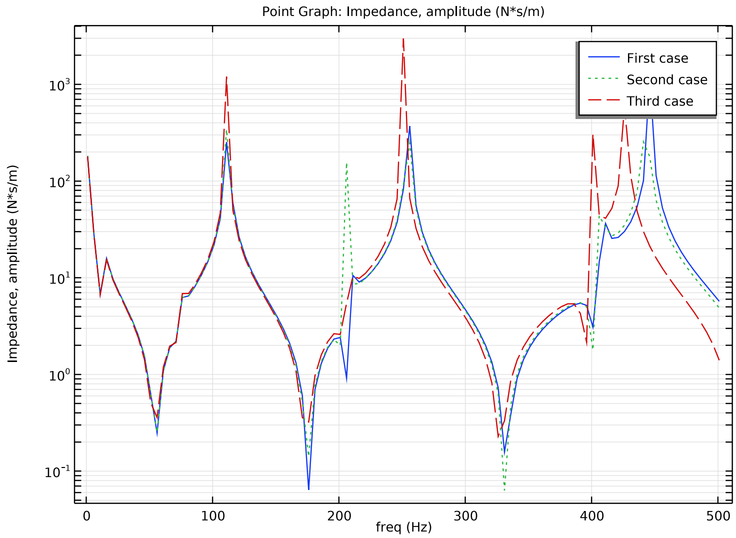


Figure 11: Impedance amplitude at Point A for the twisting load.

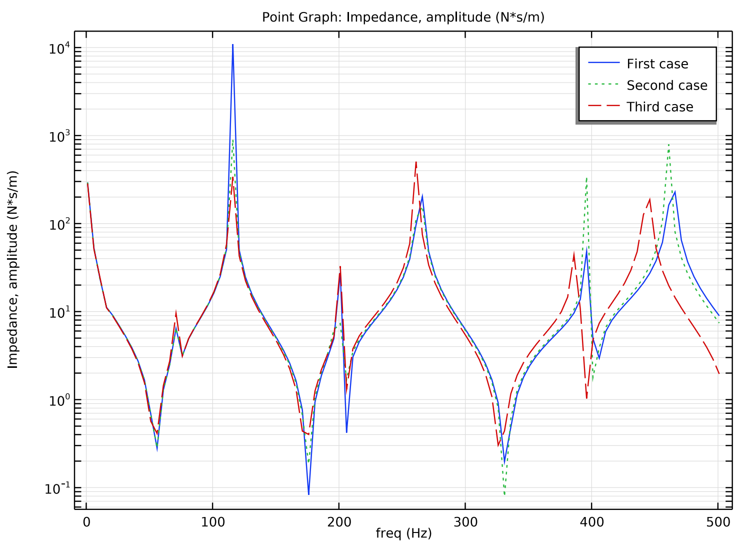


Figure 12: Impedance amplitude at Point B for the twisting load.

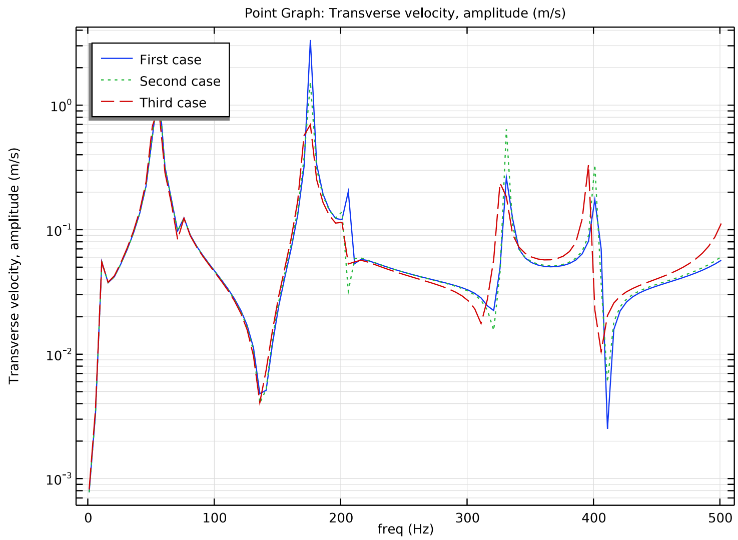


Figure 13: Transverse velocity amplitude at Point C for the twisting load.

Notes About the COMSOL Implementation

- Modeling a laminated composite 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 use the *layerwise (LW)* theory based **Layered Shell** interface. This interface is used to apply various loads and constraints on different layers, and to solve for stresses and other relevant variables in each layer of the shell.
- The delamination between two layers is modeled using the **Delamination** feature available in the Layered Shell interface. At the locations of initially delaminated zones, the initial state is set to **Delaminated** instead of **Bonded**.

Application Library path: `Composite_Materials_Module/Dynamics_and_Vibration/forced_vibration_of_a_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 Click **Study**.

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

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 `forced_vibration_of_a_composite_laminate_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.

Layered Material: [90/45/0]

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

2 In the **Settings** window for **Layered Material**, type `Layered Material: [90/45/0]` 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: Carbon-Epoxy (mat1)	90	`th`	1

4 Click **Add** two times.

5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 2`	Material: Carbon-Epoxy (mat1)	45	`th`	1
`Layer 3`	Material: Carbon-Epoxy (mat1)	0	`th`	1

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.

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
`vel_z`	`abs(lshell.u_tZ)`		`Transverse velocity, amplitude`
`Z_z`	`abs(F/lshell.u_tZ)`		`Impedance, amplitude`

Boundary System 1 (sys1)

1 In the **Model Builder** window, 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**.

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)>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 `300[mm]`.
4 In the **Height** text field, type `100[mm]`.

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

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.
2 Select the object **r1** only.
3 In the **Settings** window for **Array**, locate the **Size** section.
4 In the **yw size** text field, type `3`.
5 Locate the **Displacement** section. In the **yw** text field, type `150[mm]`.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `30[mm]`.
4 Locate the **Position** section. In the **xw** text field, type `150[mm]`.
5 In the **yw** text field, type `200[mm]`.
6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `180`.
7 Right-click **Circle 1 (c1)** and choose **Duplicate**.

Work Plane 1 (wp1)>Circle 2 (c2)

1 In the **Model Builder** window, click **Circle 2 (c2)**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Sector angle** text field, type `180`.
4 Locate the **Position** section. In the **xw** text field, type `100[mm]`.
5 In the **yw** text field, type `400[mm]`.
6 Right-click **Circle 2 (c2)** and choose **Duplicate**.

Work Plane 1 (wp1)>Circle 3 (c3)

1 In the **Model Builder** window, click **Circle 3 (c3)**.
2 In the **Settings** window for **Circle**, locate the **Position** section.
3 In the **xw** text field, type `200[mm]`.
4 In the **yw** text field, type `300[mm]`.
5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `0`.
6 In the **Work Plane** toolbar, click **Build All**.

Form Union (fin)

In the **Home** toolbar, click **Build All**.

MATERIALS

Layered Material Link 1 (llmat1)

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

LAYERED SHELL (LSHELL)

Linear Elastic Material 1

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

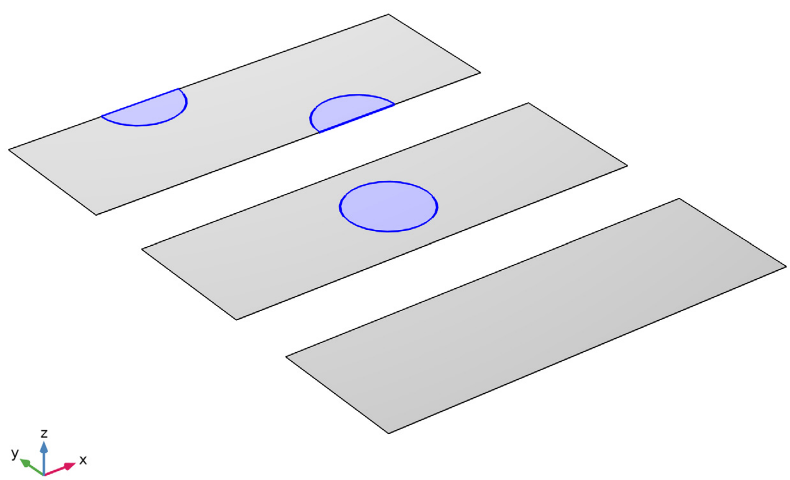
Property	Variable	Value	Unit	Property group
Elasticity matrix	{D11, D12, D22, D13, D23, D33, D14, D24, D34, D44, D15, D25, D35, D45, D55, D16, D26, D36, D46, D56, D66} ; Dij = Dji	`{D_11, D_12, D_22, D_13, D_23, D_33, 0, 0, 0, D_44, 0, 0, 0, 0, D_55, 0, 0, 0, 0, 0, D_66}`	Pa	Anisotropic
Density	rho	`rho_l`	kg/m³	Basic

LAYERED SHELL (LSHELL)

Delamination 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Layered Shell (lshell)** and choose **Material Models>Delamination**.

2 Select Boundaries 4–6 only.



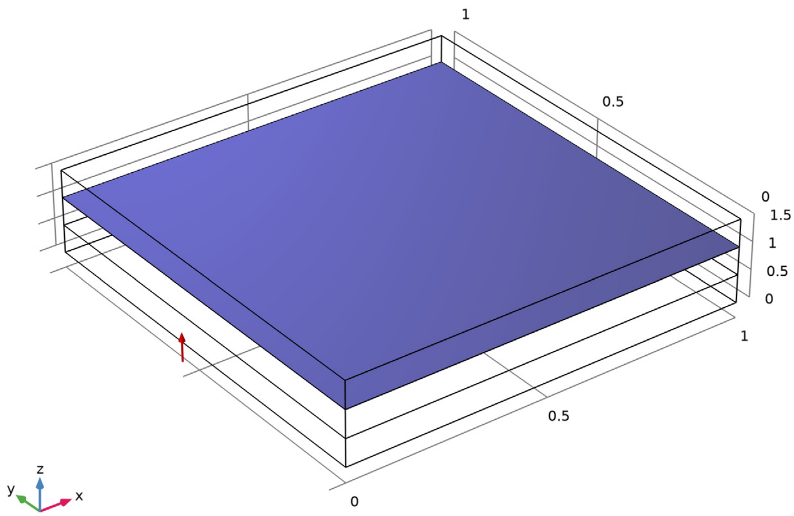
3 In the **Settings** window for **Delamination**, locate the **Interface Selection** section.

4 From the **Apply to** list, choose **Selected interfaces**.

5 Click Clear All.

6 In the **Selection** table, select the check box for **Layer 2-Layer 3**.

7 Click **Layer 3D Preview** in the upper-right corner of the **Interface Selection** section.



8 Locate the **Initial State** section. From the list, choose **Delaminated**.

9 Locate the **Contact** section. In the p_n text field, type `1e-3*lshell.Eequ/lshell.d_ad`.

Fixed Constraint 1

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

2 Select Edges 1, 4, and 7 only.

Line Load, Point-A

1 In the **Physics** toolbar, click **Points** and choose **Line Load**.

2 In the **Settings** window for **Line Load**, type `Line Load, Point-A` in the **Label** text field.

3 Select Points 19, 21, and 23 only.

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

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

0	x
0	y
F	z

6 Right-click **Line Load, Point-A** and choose **Duplicate**.

Line Load, Point-B: [Bending]

1 In the **Model Builder** window, under **Component 1 (comp1)>Layered Shell (lshell)** click **Line Load, Point-A 1**.

2 In the **Settings** window for **Line Load**, type `Line Load, Point-B: [Bending]` in the **Label** text field.

3 Locate the **Point Selection** section. Click **Clear Selection**.

4 Select Points 20, 22, and 24 only.

5 Right-click **Line Load, Point-B: [Bending]** and choose **Duplicate**.

Line Load, Point-B: [Twisting]

1 In the **Model Builder** window, under **Component 1 (comp1)>Layered Shell (lshell)** click **Line Load, Point-B: [Bending] 1**.

2 In the **Settings** window for **Line Load**, type `Line Load, Point-B: [Twisting]` in the **Label** text field.

3 Locate the **Force** section. Specify the $\mathbf{F}_{tot}$ vector as

0	x
0	y
-F	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**.

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

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

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 4–6 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click **Build All**.

STUDY: BENDING LOAD

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study: Bending Load` in the **Label** text field.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study: Bending Load** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(1,5,501)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Layered Shell (lshell)>Line Load, Point-B: [Twisting]**.

6 Right-click and choose **Disable**.

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

RESULTS

Cut Point 3D: A

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material**.

4 Locate the **Point Data** section. In the **x** text field, type `0.3 0.3 0.3`.

5 In the **y** text field, type `0 0.15 0.3`.

6 In the **z** text field, type `0 0 0`.

7 Right-click **Cut Point 3D: A** and choose **Duplicate**.

Cut Point 3D: B

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Point 3D: A 1**.

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

3 Locate the **Point Data** section. In the **y** text field, type `0.1 0.25 0.4`.

4 Right-click **Cut Point 3D: B** and choose **Duplicate**.

Cut Point 3D: C

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Point 3D: B 1**.

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

3 Locate the **Point Data** section. In the **x** text field, type `0.25 0.25 0.25`.

4 In the **y** text field, type `0.07 0.22 0.37`.

Stress (lshell)

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

2 In the **Stress (lshell)** 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: Bending Load/Solution 1 (sol1)>Layered Shell>Stress, Slice (lshell)**.

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

RESULTS

Stress, Slice (lshell)

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: Bending Load/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 **Settings** window for **1D Plot Group**, locate the **Data** section.

2 From the **Parameter selection (freq)** list, choose **Manual**.

3 In the **Parameter indices (1-101)** text field, type `101`.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Through Thickness 1

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

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `2 4 6` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Through Thickness**, locate the **y-Axis Data** section.

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

9 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`First case`
`Second case`
`Third case`

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

Impedance, Point A [Bending Load]

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Impedance, Point A [Bending Load]` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D: A**.

4 Locate the **Axis** section. Select the **y-axis log scale** check box.

Point Graph 1

1 Right-click **Impedance, Point A [Bending Load]** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

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

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

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

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

7 In the table, enter the following settings:

Legends
`First case`
`Second case`
`Third case`

8 In the **Impedance, Point A [Bending Load]** toolbar, click **Plot**.

Impedance, Point A [Bending Load]

In the **Model Builder** window, right-click **Impedance, Point A [Bending Load]** and choose **Duplicate**.

Impedance, Point B [Bending Load]

1 In the **Model Builder** window, under **Results** click **Impedance, Point A [Bending Load] 1**.

2 In the **Settings** window for **1D Plot Group**, type `Impedance, Point B [Bending Load]` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D: B**.

4 In the **Impedance, Point B [Bending Load]** toolbar, click **Plot**.

5 Right-click **Impedance, Point B [Bending Load]** and choose **Duplicate**.

Velocity, Point C [Bending Load]

1 In the **Model Builder** window, under **Results** click **Impedance, Point B [Bending Load] 1**.

2 In the **Settings** window for **1D Plot Group**, type `Velocity, Point C [Bending Load]` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D: C**.

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

Point Graph 1

1 In the **Model Builder** window, expand the **Velocity, Point C [Bending Load]** node, then click **Point Graph 1**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

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

4 In the **Velocity, Point C [Bending Load]** toolbar, click **Plot**.

Animation [Bending Load]

1. In the **Results** toolbar, click **Animation** and choose **Player**.
2. In the **Settings** window for **Animation**, type `Animation [Bending Load]` in the **Label** text field.
3. Locate the **Animation Editing** section. From the **Sequence type** list, choose **Dynamic data extension**.
4. Locate the **Frames** section. In the **Frame number** text field, type `25`.
5. Click the **Play** button in the **Graphics** toolbar.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: TWISTING LOAD

1. In the **Model Builder** window, click **Study 2**.
2. In the **Settings** window for **Study**, type `Study: Twisting Load` in the **Label** text field.

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study: Twisting Load** click **Step 1: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3. In the **Frequencies** text field, type `range(1,5,501)`.
4. Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
5. In the tree, select **Component 1 (comp1)>Layered Shell (lshell)>Line Load, Point-B: [Bending]**.
6. Right-click and choose **Disable**.
7. In the **Home** toolbar, click **Compute**.

RESULTS

Cut Point 3D: A

In the **Model Builder** window, under **Results>Datasets** right-click **Cut Point 3D: A** and choose **Duplicate**.

Cut Point 3D: A 1

1 In the **Model Builder** window, click **Cut Point 3D: A 1**.

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

3 From the **Dataset** list, choose **Layered Material 2**.

Cut Point 3D: B

In the **Model Builder** window, right-click **Cut Point 3D: B** and choose **Duplicate**.

Cut Point 3D: B 1

1 In the **Model Builder** window, click **Cut Point 3D: B 1**.

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

3 From the **Dataset** list, choose **Layered Material 2**.

Cut Point 3D: C

In the **Model Builder** window, right-click **Cut Point 3D: C** and choose **Duplicate**.

Cut Point 3D: C 1

1 In the **Model Builder** window, click **Cut Point 3D: C 1**.

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

3 From the **Dataset** list, choose **Layered Material 2**.

Stress (lshell) 1

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

2 In the **Stress (lshell) 1** toolbar, click **Plot**.

ADD PREDEFINED PLOT

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

2 In the tree, select **Study: Twisting Load/Solution 2 (sol2)>Layered Shell>Stress, Slice (lshell)**.

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

RESULTS

Stress, Slice (lshell) 1

In the **Stress, Slice (lshell) 1** toolbar, click **Plot**.

ADD PREDEFINED PLOT

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

2 In the tree, select **Study: Twisting Load/Solution 2 (sol2)>Layered Shell>Stress, Through Thickness (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

Stress, Through Thickness (lshell) 1

1 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

2 From the **Parameter selection (freq)** list, choose **Manual**.

3 In the **Parameter indices (1-101)** text field, type `101`.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Through Thickness 1

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

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `2 4 6` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Through Thickness**, locate the **y-Axis Data** section.

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

9 In the **Stress, Through Thickness (lshell) 1** toolbar, click **Plot**.

Impedance, Point A [Bending Load]

In the **Model Builder** window, under **Results** right-click **Impedance, Point A [Bending Load]** and choose **Duplicate**.

Impedance, Point A [Twisting Load]

1 In the **Model Builder** window, under **Results** click **Impedance, Point A [Bending Load] 1**.
2 In the **Settings** window for **1D Plot Group**, type `Impedance, Point A [Twisting Load]` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D: A 1**.
4 In the **Impedance, Point A [Twisting Load]** toolbar, click **Plot**.

Impedance, Point B [Bending Load]

In the **Model Builder** window, right-click **Impedance, Point B [Bending Load]** and choose **Duplicate**.

Impedance, Point B [Twisting Load]

1 In the **Model Builder** window, under **Results** click **Impedance, Point B [Bending Load] 1**.
2 In the **Settings** window for **1D Plot Group**, type `Impedance, Point B [Twisting Load]` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D: B 1**.
4 In the **Impedance, Point B [Twisting Load]** toolbar, click **Plot**.

Velocity, Point C [Bending Load]

In the **Model Builder** window, right-click **Velocity, Point C [Bending Load]** and choose **Duplicate**.

Velocity, Point C [Twisting Load]

1 In the **Model Builder** window, under **Results** click **Velocity, Point C [Bending Load] 1**.
2 In the **Settings** window for **1D Plot Group**, type `Velocity, Point C [Twisting Load]` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D: C 1**.
4 In the **Velocity, Point C [Twisting Load]** toolbar, click **Plot**.

Impedance, Point A [Bending Load], Impedance, Point B [Bending Load], Stress (lshell), Stress, Slice (lshell), Stress, Through Thickness (lshell), Velocity, Point C [Bending Load]

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (lshell)**, **Stress, Slice (lshell)**, **Stress, Through Thickness (lshell)**, **Impedance, Point A [Bending Load]**, **Impedance, Point B [Bending Load]**, and **Velocity, Point C [Bending Load]**.
2 Right-click and choose **Group**.

Bending Load

In the **Settings** window for **Group**, type `Bending Load` in the **Label** text field.

Impedance, Point A [Twisting Load], Impedance, Point B [Twisting Load], Stress (Ishell) 1, Stress, Slice (Ishell) 1, Stress, Through Thickness (Ishell) 1, Velocity, Point C [Twisting Load]

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (Ishell) 1**, **Stress, Slice (Ishell) 1**, **Stress, Through Thickness (Ishell) 1**, **Impedance, Point A [Twisting Load]**, **Impedance, Point B [Twisting Load]**, and **Velocity, Point C [Twisting Load]**.
2 Right-click and choose **Group**.

Twisting Load

In the **Settings** window for **Group**, type `Twisting Load` in the **Label** text field.

Animation [Bending Load]

In the **Model Builder** window, under **Results>Export** right-click **Animation [Bending Load]** and choose **Duplicate**.

Animation [Twisting Load]

1 In the **Model Builder** window, under **Results>Export** click **Animation [Bending Load] 1**.
2 In the **Settings** window for **Animation**, type `Animation [Twisting Load]` in the **Label** text field.
3 Locate the **Scene** section. From the **Subject** list, choose **Stress (Ishell) 1**.
4 Click the **Play** button in the **Graphics** toolbar.