Model created in COMSOL Multiphysics 6.4

Analysis of a Composite Blade Using a Multiple Model Method

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This model is based on the Structural Mechanics Module Application Library model *Vibrations of an Impeller*. The model herein, however, uses a simplified standalone blade geometry, but the boundary conditions and loading are taken from the original model. The model compares three commonly used methods for the analysis of laminated composite shells:

1. An equivalent single layer theory, using first-order shear deformation theory (ESL-FSDT)
2. A layerwise elasticity theory
3. Multiple model methods

The ESL theory reduces a 3D continuum to an equivalent 2D description, thereby reducing the size and computational time involved in solving a problem. The first-order shear deformation theory (FSDT) is implemented in the **Linear Elastic Material, Layered** model in the **Shell** interface.

The layerwise elasticity theory is implemented in the **Layered Shell** interface. The theory considers a composite as a 3D continuum, giving more accurate resolution of stresses and strains, particularly in the through-thickness direction.

The ESL and layerwise theories each have advantages and disadvantages, in terms of solution accuracy and solution time, for different types of problems. By judiciously combining the use of the two theories in a single model, it is possible to obtain high accuracy results, at a low computational cost. The approach of combining the theories in this way is called a *multiple model method*, or a *global-local analysis*. For more details regarding multiple model methods, see Ref. 1.

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

Model Definition

In this model, you will perform an eigenfrequency analysis and a frequency-domain analysis of a composite blade, using the three modeling approaches discussed previously:

- Equivalent single layer theory

- Layerwise theory
- Multiple model method, in which the ESL and layerwise theories are combined in the through-thickness direction of the composite.

Geometry and Boundary Conditions

The geometry of the blade is shown in Figure 1. The boundary conditions and loading are:

- The short end of the blade is fixed.
- A load is pressure excitation applied on the top face of the blade. The load is entered as a normal component of the boundary load using the expression

$$F_n = -p_0 \exp[-jm \operatorname{atan}(Y/X)]$$

using the magnitude of $p_0 = 10^4$ Pa and azimuthal mode number $m = 3$. The excitation frequency is 10 Hz.

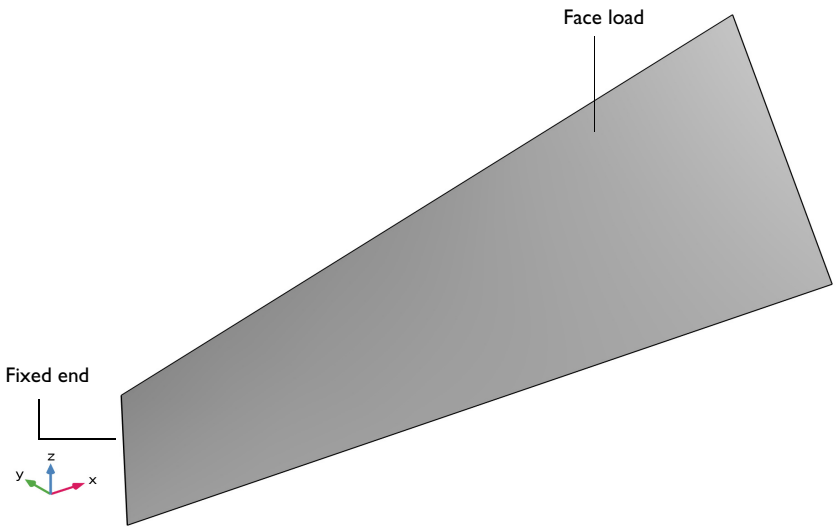

Figure 1: Geometry of the blade, with boundary conditions and loading.

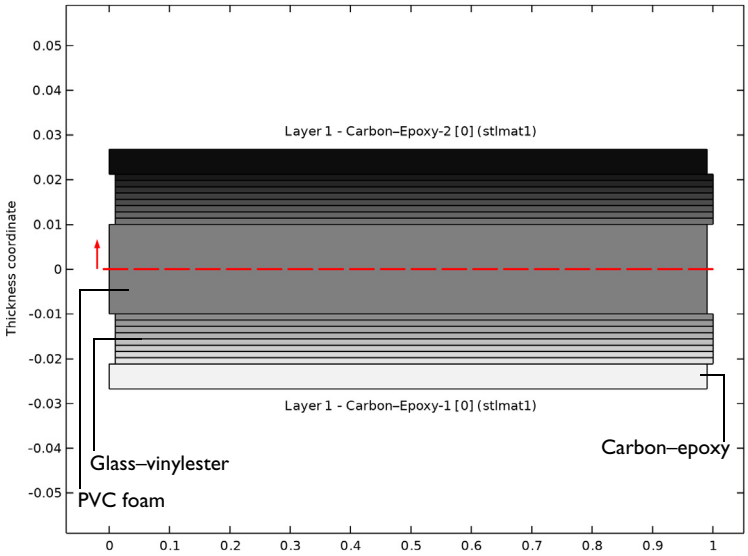

Figure 2: Through-thickness view of the laminate.

LAMINATE MATERIAL PROPERTIES

The composite blade is a sandwich structure consisting of three different material types: carbon–epoxy (outer layer), glass–vinylester, and PVC foam (core), as shown in Figure 2. The stacking sequence is shown in Figure 3. The material properties of the different laminae are taken from the model Stress and Modal Analysis of a Wind Turbine Composite Blade, also in the Composite Materials Module Application Library.

Carbon–Epoxy Layer

The outer layer of the sandwich structure is a single carbon–epoxy layer with a thickness of 5.6 mm, oriented at 0 degrees to the principal axis. The density of the layer is 1560 kg/m^3. The transversely isotropic material properties are given in Table 1.

TABLE 1: MATERIAL PROPERTIES OF THE SINGLE CARBON–EPOXY LAYER.

Material property	Value
$\{E_{11}, E_{22}\}$	{139,9}(GPa)
G_{12}	5.5(GPa)
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.32,0.32}

Glass–Vinylester Laminate

The next layer of the sandwich structure is a glass–vinylester laminate. The density of the laminate is 1890 kg/m^3. The transversely isotropic material properties are given in Table 2.

TABLE 2: MATERIAL PROPERTIES OF THE GLASS–VINYLESTER LAMINA.

Material property	Value
$\{E_{11}, E_{22}\}$	{41,9}(GPa)
G_{12}	4.1(GPa)
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.3,0.3}

This laminate is made of eight layers, each of 1.4 mm thickness, with the stacking sequence shown in Table 3.

TABLE 3: FIBER ORIENTATION IN THE GLASS-VINYLESTER LAMINATE.

Layer number	Fiber orientation
1	0
2	45
3	-45
4	90
5	90
6	-45
7	45
8	0

PVC Foam

The core material of the sandwich structure is a PVC foam of thickness 2 cm. The density of the material is 200 kg/m^3. The values of Young's modulus and Poisson's ratio are 250 MPa and 0.35, respectively.

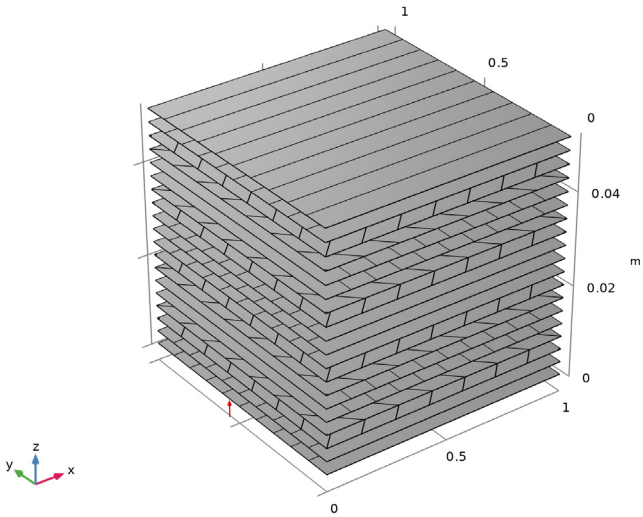

Figure 3: Stacking sequence for the laminate showing the fiber orientation of each layer, from bottom to top.

FINITE ELEMENT MESH

Composites modeled with the **Layered Shell** or **Shell** interfaces are discretized at two levels. The in-plane discretization in done in a standard fashion in the **Mesh** node in the model builder tree. The out-of-plane (thickness) discretization is controlled in the **Layered Material** node. A triangular mesh is used in the plane, as shown in Figure 4. The discretization in the laminate thickness direction (given as a number of elements) is shown in Table 4.

TABLE 4: MESH ELEMENTS IN THICKNESS DIRECTION.

Material	Total thickness (mm)	Mesh Elements
Carbon-Epoxy	11.2	8
Glass-Vinylester	22.4	16
PVC foam	20	10

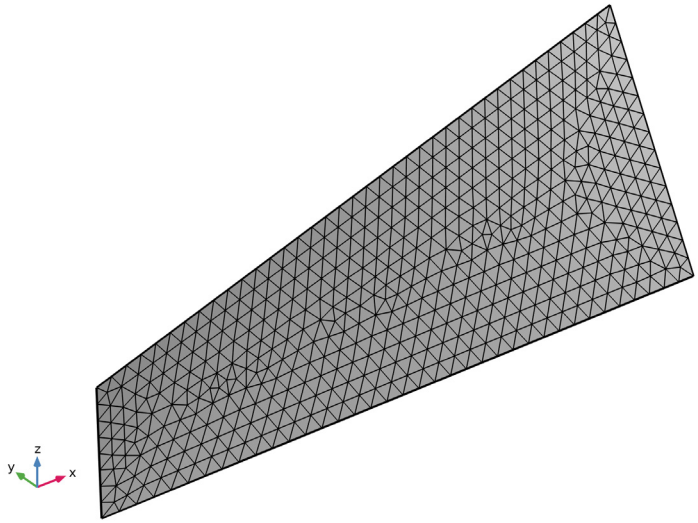

Figure 4: The mesh in the base selection.

Results and Discussion

The layerwise theory uses three dimensional kinematics, and has the capacity to predict stresses and strains to high accuracy. The results from using this theory are therefore used as a benchmark. The results from using the ESL theory and the multiple model method are compared with the layerwise theory predictions.

EIGENFREQUENCY ANALYSIS

The first six eigenmodes, using the multiple model method, are shown in Figure 5. The eigenmodes using the layerwise and ESL theories are essentially indistinguishable from the multiple model method eigenmodes, and they are not shown here. Note that some modes from layerwise and ESL theories are out of phase with those presented in Figure 5. The computed eigenfrequencies differ between the different modeling approaches. The corresponding six eigenfrequencies are shown in Table 5.

TABLE 5: COMPARISON OF EIGENFREQUENCIES.

Multiple model method (Hz)	Layerwise (Hz)	ESL (Hz)
13.023	13.011	13.647
35.321	35.337	44.464

TABLE 5: COMPARISON OF EIGENFREQUENCIES.

Multiple model method (Hz)	Layerwise (Hz)	ESL (Hz)
49.790	49.832	59.443
101.11	100.97	113.20
137.64	137.79	178.59
181.19	181.42	265.11

It is evident that the computationally less expensive predictions from the multiple model method are in very close agreement with the layerwise values. Predictions using the ESL theory deviate significantly, likely owing to the fact that while the ESL theory is computationally inexpensive, it is less accurate for thick to moderately thick shells. This underscores the computational merit of a multiple model method, in which thicker parts of a sandwich structure are modeled using a layerwise theory, and thinner parts are modeled using the ESL theory.

A performance comparison for eigenfrequency study is shown in Table 6. Keep in mind that the solution time depends on the machine used for the simulation, so it is better to consider the relative solution time instead.

TABLE 6: PERFORMANCE COMPARISON FOR THE EIGENFREQUENCY STUDIES.

Comparison	Multiple model method	Layerwise	ESL
Number of DOFs	264579	424557	12306
Relative DOFs	0.62	1	0.028
Solution time (s)	21	32	3
Relative solution time	0.656	1	0.093

FREQUENCY-DOMAIN ANALYSIS

The absolute value of stress component in the fiber direction with each modeling approach is presented in Figure 6. The figure shows that the stress response from the multiple model method closely matches the layerwise case, both in distribution and in peak value. The figure also shows that the stress distribution using the ESL theory differs significantly, and most notably, the peak value is very different.

The frequency-domain results in terms of displacements are presented in Figure 7 for each modeling approach. Again, the multiple model method approach produces results that are in close agreement with the results from the layerwise theory, while the ESL theory fails to accurately predict the displacement distribution and peak value (20% difference in peak value).

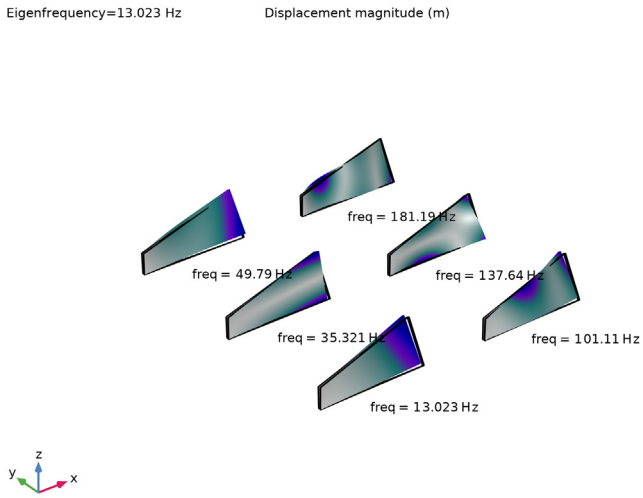

Figure 5: The first six mode shapes and corresponding eigenfrequencies of the composite blade, using the multiple model method.

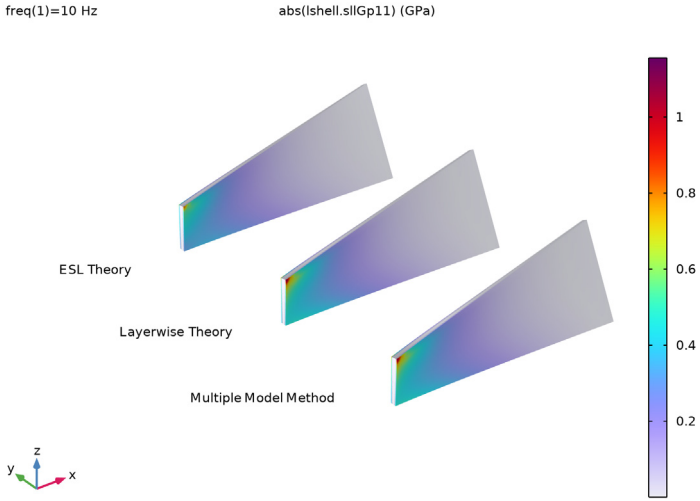

Figure 6: Absolute stress in fiber direction in the composite blade.

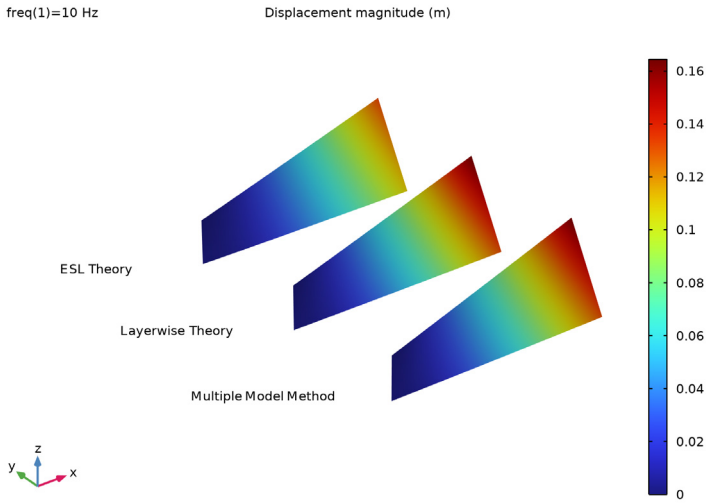

Figure 7: Displacement in the top layer of the composite blade.

The performance of three methods with frequency-domain study is shown in Table 7. Keep in mind that the solution time depends on the machine used for the simulation, so it is better to consider the relative solution time instead.

TABLE 7: PERFORMANCE COMPARISON FOR THE FREQUENCY-DOMAIN STUDIES.

Comparison	Multiple model method	Layerwise	ESL
Number of DOFs	264579	424557	12306
Relative DOFs	0.62	1	0.028
Solution time (s)	17	26	4
Relative solution time	0.653	1	0.1538

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.
- To model composites you can use two approaches: You can use either the **Layered Shell** interface, that uses the layerwise theory, or the **Linear Elastic Material, Layered** node in the **Shell** interface, that uses the Equivalent Single Layer (ESL) theory.
- The multiple model method combines the aforementioned modeling approaches, and in order to combine the **Layered Shell** and **Shell** interfaces in the thickness direction, a **Layered Shell-Shell Connection** multiphysics coupling must be used. You must also use the **Layered Material Stack** node for the through-thickness coupling between the interfaces.
- In a situation where **Layered Shell** and **Shell** interfaces are coupled in-plane, you must use a **Layered Shell-Structural Transition** multiphysics coupling. Here, the same **Single Layer Material, Layered Material Link** or **Layered Material Stack** node must be used in both interfaces. This modeling approach is also a multiple model method.
- It is not advised to use the **Layered Shell** interface for discontinuous layers, as it can create problems in fold-line constraints. No fold-lines exist in the present model, hence the **Layered Shell** interface used to model the PVC foam and the carbon–epoxy layers.

Reference

1. J.N. Reddy, *Mechanics of Laminated Composite Plates and Shells: Theory and Analysis,* 2nd ed., CRC Press, 2004.

Application Library path: `Composite_Materials_Module/Tutorials/composite_blade_multiple_model_method`

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** > **Eigenfrequency**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
th	1.4[mm]	0.0014 m	Layer thickness
thc	2[cm]	0.02 m	Core thickness
mn	3	3	Azimuthal mode number
p0	2e4[Pa]	20000 Pa	Load magnitude

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: CE-[0]

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

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

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

Layer	Material	Rotation	Value	Thickness	Mesh elements
`Layer 1`	Material: Carbon-Epoxy (mat1)	`0.0`	0 rad	`th*4`	4

Material: Glass–Vinylester

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

2 In the **Settings** window for **Material**, type `Material: Glass-Vinylester` in the **Label** text field.

Layered Material: GV-[0/45/-45/90]_s

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

2 In the **Settings** window for **Layered Material**, type `Layered Material: GV-[0/45/-45/90]_s` in the **Label** text field.

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

Layer	Material	Rotation	Value	Thickness	Mesh elements
`Layer 1`	Material: Glass-Vinylester (mat2)	`0.0`	0 rad	`th`	1

4 Click + **Add**.

Add six additional layers so that the material has a total of eight layers.

5 In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
`Layer 2`	Material: Glass-Vinylester (mat2)	45	0.7854 rad	`th`	1
`Layer 3`	Material: Glass-Vinylester (mat2)	`-45`	-0.7854 rad	`th`	1

Layer	Material	Rotation	Value	Thickness	Mesh elements
Layer 4	Material: Glass-Vinylester (mat2)	90	1.5708 rad	th	1
Layer 5	Material: Glass-Vinylester (mat2)	90	1.5708 rad	th	1
Layer 6	Material: Glass-Vinylester (mat2)	-45	-0.7854 rad	th	1
Layer 7	Material: Glass-Vinylester (mat2)	45	0.7854 rad	th	1
Layer 8	Material: Glass-Vinylester (mat2)	0	0 rad	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.

Material: PVC Foam

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

2 In the **Settings** window for **Material**, type `Material: PVC Foam` in the **Label** text field.

Layered Material: PF-[0]

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

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

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

Layer	Material	Rotation	Value	Thickness	Mesh elements
Layer 1	Material: PVC Foam (mat3)	0.0	0 rad	thc	10

GEOMETRY 1

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 Click **Go to Plane Geometry**.

Work Plane 1 (wp1) > Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **yw** text field, type `0.4`.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
-2

4 Select the **Reverse direction** checkbox.

5 Click to expand the **Scales** section. In the table, enter the following settings:

Scales xw	Scales yw
1	2

6 Click to expand the **Twist Angles** section. In the table, enter the following settings:

Twist angles (deg)
30

7 Click **Build All Objects**.

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

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

DEFINITIONS

Boundary System 1 (sys1)

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

MATERIALS

The laminate is modeled using a **Layered Material Stack** node. Multiple layered material links are added to demonstrate the multiple model method.

Layered Material Stack 1 (stlmat1)

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

Carbon–Epoxy-1 [0]

In the **Settings** window for **Layered Material Link**, type `Carbon-Epoxy-1 [0]` in the **Label** text field.

Glass–Vinylester-1 [0/45/-45/90]_s

1 Right-click **Carbon–Epoxy-1 [0]** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material Link**, type `Glass-Vinylester-1 [0/45/-45/90]_s` in the **Label** text field.

3 Locate the **Link Settings** section. From the **Material** list, choose **Layered Material: GV-[0/45/-45/90]_s (lmat2)**.

PVC Foam [0]

1 Right-click **Glass–Vinylester-1 [0/45/-45/90]_s** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material Link**, type `PVC Foam [0]` in the **Label** text field.

3 Locate the **Link Settings** section. From the **Material** list, choose **Layered Material: PF-[0] (lmat3)**.

Glass–Vinylester-2 [0/45/-45/90]_s

1 Right-click **PVC Foam [0]** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material Link**, type `Glass-Vinylester-2 [0/45/-45/90]_s` in the **Label** text field.

3 Locate the **Link Settings** section. From the **Material** list, choose **Layered Material: GV-[0/45/-45/90]_s (lmat2)**.

Carbon–Epoxy-2 [0]

1 Right-click **Glass–Vinylester-2 [0/45/-45/90]_s** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material Link**, type `Carbon-Epoxy-2 [0]` in the **Label** text field.

3 Locate the **Link Settings** section. From the **Material** list, choose **Layered Material: CE-[0] (lmat1)**.

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**, click to expand the **Preview Plot Settings** section.

3 In the **Thickness-to-width ratio** text field, type `0.4`.

4 Click **Layer Cross-Section Preview** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Layer Cross-Section Preview** to enable the through-thickness view of the laminated material, as in Figure 2.

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

6 Locate the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type `1`.

7 Click **Layer Stack Preview** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Layer Stack Preview** to show the stacking sequence, including the fiber orientation, as in Figure 3.

Modeling Instructions (Frequency and Eigenfrequency Analysis Using the Multiple Model Method)

This section describes how to model a composite blade using the multiple model method. To this end, you will use one **Layered Shell** and two **Shell** interfaces.

LAYERED SHELL (MULTIPLE MODEL METHOD)

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

2 In the **Settings** window for **Layered Shell**, type `Layered Shell (Multiple Model Method)` in the **Label** text field.

Add the material properties for the carbon-epoxy, the glass-vinylester, and the PVC foam.

3 Locate the **Shell Properties** section. Clear the **Use all layers** checkbox.

4 Click **Clear All**.

5 In the **Selection** table, select the checkboxes for **Layer 1 - Carbon–Epoxy-1 [0]**, **Layer 1 - PVC Foam [0]**, and **Layer 1 - Carbon–Epoxy-2 [0]**.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Layered Shell (Multiple Model Method) (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** checkbox.

Fixed Constraint 1

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

2 Select Edge 1 only.

3 In the **Settings** window for **Fixed Constraint**, locate the **Shell Properties** section.

4 Select the **Use all layers** checkbox.

Face Load, Pressure

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

2 In the **Settings** window for **Face Load**, type `Face Load, Pressure` in the **Label** text field.

3 Select Boundary 1 only.

4 Locate the **Interface Selection** section. From the **Apply to** list, choose **Top interface**.

5 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

6 In the *p* text field, type `p0*exp(-j*mn*atan2(Y,X))`.

SHELL 1 (MULTIPLE MODEL METHOD)

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

2 In the **Settings** window for **Shell**, type `Shell 1 (Multiple Model Method)` in the **Label** text field.

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

4 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

5 Click **OK**.

6 In the **Settings** window for **Shell**, click to expand the **Advanced Settings** section.

7 Clear the **Use MITC interpolation** checkbox.

Linear Elastic Material, Layered 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Linear Elastic Material, Layered**, locate the **Shell Properties** section.

4 Clear the **Use all layers** checkbox.

5 From the **Stack member** list, choose **Glass–Vinylester-1 [0/45/-45/90]_s (stlmat1.stllmat2)**.

6 Locate the **Linear Elastic Material** section. From the **Material symmetry** list, choose **Orthotropic**.

7 Select the **Transversely isotropic** checkbox.

8 Click to expand the **Shear Correction Factor** section. From the list, choose **User defined**.

Fixed Constraint 1

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

2 Select Edge 1 only.

3 In the **Model Builder** window, right-click **Shell 1 (Multiple Model Method) (shell)** and choose **Copy**.

SHELL 2 (MULTIPLE MODEL METHOD)

1 In the **Model Builder** window, right-click **Component 1 (comp1)** and choose **Paste Shell**.

2 In the **Messages from Paste** dialog, click **OK**.

3 In the **Settings** window for **Shell**, type `Shell 2 (Multiple Model Method)` in the **Label** text field.

Linear Elastic Material, Layered 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Shell 2 (Multiple Model Method) (shell2)** node, then click **Linear Elastic Material, Layered 1**.

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

3 From the **Stack member** list, choose **Glass–Vinylester-2 [0/45/-45/90]_s (stlmat1.stllmat4)**.

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}	{139e9, 9e9}	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	{0.32, 0.32}	1	Transversely isotropic
Shear modulus	Gvect1	5.5e9	N/m²	Transversely isotropic
Density	rho	1560	kg/m³	Basic

Material: Glass–Vinylester (mat2)

1 In the **Model Builder** window, click **Material: Glass–Vinylester (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	{Evect1, Evect2}	{41e9, 9e9}	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	{0.3, 0.3}	1	Transversely isotropic
Shear modulus	{Gvect1}	4.1e9	N/m²	Transversely isotropic
Density	rho	1890	kg/m³	Basic

Material: PVC Foam (mat3)

1 In the **Model Builder** window, click **Material: PVC Foam (mat3)**.

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}	{250e6, 250e6}	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	{0.35, 0.35}	1	Transversely isotropic
Shear modulus	Gvect1	92.6e6	N/m²	Transversely isotropic
Density	rho	200	kg/m³	Basic

MULTIPHYSICS

Layered Shell–Shell Connection 1 (lssh1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary** > **Layered Shell–Shell Connection**.

Layered Shell–Shell Connection 2 (lssh2)

1 Right-click **Layered Shell–Shell Connection 1 (lssh1)** and choose **Duplicate**.

2 In the **Settings** window for **Layered Shell–Shell Connection**, locate the **Coupled Interfaces** section.

3 From the **Shell** list, choose **Shell 2 (Multiple Model Method) (shell2)**.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY: EIGENFREQUENCY (MULTIPLE MODEL METHOD)

Switch off the generation of default plots in every study. We will use customized plots for comparison purposes.

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

2 In the **Settings** window for **Study**, type `Study: Eigenfrequency (Multiple Model Method)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Solution 1 (sol1)

In the **Study** toolbar, click **Show Default Solver**.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study: Eigenfrequency (Multiple Model Method)** > **Solver Configurations** > **Solution 1 (sol1)** node, then click **Eigenvalue Solver 1**.

2 In the **Settings** window for **Eigenvalue Solver**, locate the **Output** section.

3 From the **Scaling of eigenvectors** list, choose **RMS**.

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

RESULTS

Layered Material 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results** > **Datasets** and choose **More Datasets** > **Layered Material**.

Mode Shapes (Multiple Model Method)

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

2 In the **Settings** window for **3D Plot Group**, type `Mode Shapes (Multiple Model Method)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.

4 Locate the **Color Legend** section. Clear the **Show legends** checkbox.

5 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.

6 In the **Relative row padding** text field, type `1`.

7 In the **Relative column padding** text field, type `0.6`.

Surface 1

1 Right-click **Mode Shapes (Multiple Model Method)** and choose **Surface**.

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

3 From the **Color table** list, choose **AuroraAustralis**.

Deformation 1

Right-click **Surface 1** and choose **Deformation**.

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 **Plot Array** section.

3 From the **Array shape** list, choose **Square**.

4 From the **Order** list, choose **Column-major**.

Surface 2

1 Right-click **Surface 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type `shell.disp`.

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

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

6 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Deformation 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Deformation 1**.

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

3 In the **x-component** text field, type `u2`.

4 In the **y-component** text field, type `v2`.

5 In the **z-component** text field, type `w2`.

Surface 3

1 In the **Model Builder** window, under **Results** > **Mode Shapes (Multiple Model Method)** right-click **Surface 2** and choose **Duplicate**.

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

3 In the **Expression** text field, type `shell2.disp`.

Deformation 1

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Deformation 1**.

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

3 In the **x-component** text field, type `u3`.

4 In the **y-component** text field, type `v3`.

5 In the **z-component** text field, type `w3`.

Annotation 1

1 In the **Model Builder** window, right-click **Mode Shapes (Multiple Model Method)** and choose **Annotation**.

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

3 In the **Text** text field, type `freq = eval(freq,Hz,5) Hz`.

4 From the **Geometry level** list, choose **Global**.

5 Locate the **Position** section. In the **x** text field, type `0.7`.

6 In the **y** text field, type `-0.1`.

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

8 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Solution Array 1

In the **Model Builder** window, under **Results** > **Mode Shapes (Multiple Model Method)** > **Surface 1** right-click **Solution Array 1** and choose **Copy**.

Solution Array 1

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

Mode Shapes (Multiple Model Method)

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

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

3 In the **Mode Shapes (Multiple Model Method)** toolbar, click **Plot**.

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 the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `10`.

3 In the **Model Builder** window, click **Study 2**.

4 In the **Settings** window for **Study**, type `Study: Frequency (Multiple Model Method)` in the **Label** text field.

5 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

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

RESULTS

Layered Material 2

1 In the **Results** toolbar, click **More Datasets** and choose **Layered Material**.

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

3 From the **Dataset** list, choose **Study: Frequency (Multiple Model Method)/Solution 2 (sol2)**.

Stress

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

2 In the **Settings** window for **3D Plot Group**, type `Stress` in the **Label** text field.

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

4 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

5 In the **Stress** toolbar, click **Plot**.

This generates a dedicated view for the plot.

6 Clear the **Plot dataset edges** checkbox.

Surface 1

1 Right-click **Stress** and choose **Surface**.

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

3 In the **Expression** text field, type `abs(lshell.sllGp11)`.

4 From the **Unit** list, choose **GPa**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Deformation 1

Right-click **Surface 1** and choose **Deformation**.

Surface 2

1 In the **Model Builder** window, under **Results** > **Stress** right-click **Surface 1** and choose **Duplicate**.

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

3 From the **Title type** list, choose **None**.

4 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Deformation 1**.

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

3 In the **x-component** text field, type `u2`.

4 In the **y-component** text field, type `v2`.

5 In the **z-component** text field, type `w2`.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 In the **Expression** text field, type `abs(shell.sllGp11)`.

Surface 3

Right-click **Results** > **Stress** > **Surface 2** and choose **Duplicate**.

Deformation 1

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Deformation 1**.

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

3 In the **x-component** text field, type `u3`.

4 In the **y-component** text field, type `v3`.

5 In the **z-component** text field, type `w3`.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

3 In the **Expression** text field, type `abs(shell2.sllGp11)`.

Stress

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

2 In the **Stress** toolbar, click **Plot**.

Displacement, Slice

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

2 In the **Settings** window for **3D Plot Group**, type `Displacement, Slice` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Frequency (Multiple Model Method)/Solution 2 (sol2)**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 5**.

5 Clear the **Plot dataset edges** checkbox.

Layered Material Slice 1

1 In the **Displacement, Slice** toolbar, click **More Plots** and choose **Layered Material Slice**.

2 In the **Settings** window for **Layered Material Slice**, locate the **Through-Thickness Location** section.

3 From the **Location definition** list, choose **Relative**.

4 In the **Local z-coordinate [-1,1]** text field, type 1.

Displacement, Slice

1 In the **Model Builder** window, click **Displacement, Slice**.

2 In the **Displacement, Slice** toolbar, click **Plot**.

Modeling Instructions (Frequency and Eigenfrequency Analysis Using the Layerwise (LW) Theory)

This section describes how to model a composite blade using the **Layered Shell** interface based on the layerwise theory.

LAYERED SHELL (MULTIPLE MODEL METHOD) (LSHELL)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Layered Shell (Multiple Model Method) (lshell)** and choose **Copy**.

LAYERED SHELL (LAYERWISE THEORY)

1 In the **Model Builder** window, right-click **Component 1 (comp1)** and choose **Paste Layered Shell**.

2 In the **Messages from Paste** dialog, click **OK**.

3 In the **Settings** window for **Layered Shell**, type `Layered Shell (Layerwise Theory)` in the **Label** text field.

4 Locate the **Shell Properties** section. Select the **Use all layers** checkbox.

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** > **Eigenfrequency**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: EIGENFREQUENCY (LAYERWISE THEORY)

1 In the **Settings** window for **Study**, type `Study: Eigenfrequency (Layerwise Theory)` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study: Eigenfrequency (Layerwise Theory)** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Layered Shell (Multiple Model Method) (lshell)**, **Component 1 (comp1)** > **Shell 1 (Multiple Model Method) (shell)**, and **Component 1 (comp1)** > **Shell 2 (Multiple Model Method) (shell2)**.

5 Click **Disable in Model**.

6 In the tree, select **Component 1 (comp1)** > **Multiphysics** > **Layered Shell–Shell Connection 1 (lssh1)** and **Component 1 (comp1)** > **Multiphysics** > **Layered Shell–Shell Connection 2 (lssh2)**.

7 Click **Disable in Model**.

Solution 3 (sol3)

In the **Study** toolbar, click **Show Default Solver**.

Solution 3 (sol3)

1 In the **Model Builder** window, expand the **Study: Eigenfrequency (Layerwise Theory)** > **Solver Configurations** > **Solution 3 (sol3)** node, then click **Eigenvalue Solver 1**.

2 In the **Settings** window for **Eigenvalue Solver**, locate the **Output** section.

3 From the **Scaling of eigenvectors** list, choose **RMS**.

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

RESULTS

Layered Material 3

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Layered Material 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Eigenfrequency (Layerwise Theory)/Solution 3 (sol3)**.

Mode Shapes (Layerwise Theory)

1 In the **Model Builder** window, right-click **Mode Shapes (Multiple Model Method)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shapes (Layerwise Theory)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

4 In the **Model Builder** window, expand the **Mode Shapes (Layerwise Theory)** node.

Surface 2, Surface 3

1 In the **Model Builder** window, under **Results** > **Mode Shapes (Layerwise Theory)**, Ctrl-click to select **Surface 2** and **Surface 3**.

2 Right-click and choose **Delete**.

Surface 1

1 In the **Model Builder** window, under **Results** > **Mode Shapes (Layerwise Theory)** click **Surface 1**.

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

3 In the **Expression** text field, type `lshell2.disp`.

Deformation 1

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

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

3 In the **x-component** text field, type `u4`.

4 In the **y-component** text field, type `v4`.

5 In the **z-component** text field, type `w4`.

Mode Shapes (Layerwise Theory)

1 In the **Model Builder** window, under **Results** click **Mode Shapes (Layerwise Theory)**.

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

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

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

5 In the **Mode Shapes (Layerwise Theory)** toolbar, click **Plot**.

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 the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `10`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Layered Shell (Multiple Model Method) (lshell)**, **Component 1 (comp1)** > **Shell 1 (Multiple Model Method) (shell)**, and **Component 1 (comp1)** > **Shell 2 (Multiple Model Method) (shell2)**.

5 Click **Disable in Model**.

6 In the tree, select **Component 1 (comp1)** > **Multiphysics** > **Layered Shell–Shell Connection 1 (lssh1)** and **Component 1 (comp1)** > **Multiphysics** > **Layered Shell–Shell Connection 2 (lssh2)**.

7 Click **Disable in Model**.

8 In the **Model Builder** window, click **Study 4**.

9 In the **Settings** window for **Study**, type `Study: Frequency (Layerwise Theory)` in the **Label** text field.

10 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

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

RESULTS

Layered Material 4

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Layered Material 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Frequency (Layerwise Theory)/Solution 4 (sol4)**.

Stress

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

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

5 In the **Relative padding** text field, type `2`.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 Select the **Manual indexing** checkbox.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

3 Select the **Manual indexing** checkbox.

Surface 4

1 Right-click **Results** > **Stress** > **Surface 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Layered Material 4**.

4 Locate the **Expression** section. In the **Expression** text field, type `abs(lshell2.sllGp11)`.

5 Locate the **Plot Array** section. In the **Index** text field, type `1`.

Deformation 1

1 In the **Model Builder** window, expand the **Surface 4** node, then click **Deformation 1**.

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

3 In the **x-component** text field, type `u4`.

4 In the **y-component** text field, type `v4`.

5 In the **z-component** text field, type `w4`.

Stress

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

2 In the **Stress** toolbar, click **Plot**.

Displacement, Slice

1 In the **Model Builder** window, click **Displacement, Slice**.

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

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

5 In the **Relative padding** text field, type `2`.

Layered Material Slice 2

1 In the **Model Builder** window, under **Results** > **Displacement, Slice** right-click **Layered Material Slice 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Frequency (Layerwise Theory)/Solution 4 (sol4)**.

4 Locate the **Expression** section. In the **Expression** text field, type `lshell2.disp`.

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

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Layered Material Slice 1**.

7 In the **Displacement, Slice** toolbar, click **Plot**.

Modeling Instructions (Frequency and Eigenfrequency Analysis using the Equivalent Single Layer (ESL) Theory)

This section describes how to model a composite blade using the **Shell** interface based on the equivalent single layer (ESL) theory.

SHELL 1 (MULTIPLE MODEL METHOD) (SHELL)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Shell 1 (Multiple Model Method) (shell)** and choose **Copy**.

SHELL (ESL THEORY)

1 In the **Model Builder** window, right-click **Component 1 (comp1)** and choose **Paste Shell**.

2 In the **Messages from Paste** dialog, click **OK**.

3 In the **Settings** window for **Shell**, type `Shell (ESL Theory)` in the **Label** text field.

Linear Elastic Material, Layered 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Shell (ESL Theory) (shell3)** node, then click **Linear Elastic Material, Layered 1**.

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

3 Select the **Use all layers** checkbox.

Face Load, Pressure

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

2 In the **Settings** window for **Face Load**, type `Face Load, Pressure` in the **Label** text field.

3 Select Boundary 1 only.

4 Locate the **Through-Thickness Location** section. From the list, choose **Top surface**.

5 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

6 In the *p* text field, type `p0*exp(-j*mn*atan2(Y,X))`.

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** > **Eigenfrequency**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: EIGENFREQUENCY (ESL THEORY)

1 In the **Settings** window for **Study**, type `Study: Eigenfrequency (ESL Theory)` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study: Eigenfrequency (ESL Theory)** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Layered Shell (Multiple Model Method) (lshell)**, **Component 1 (comp1)** > **Shell 1 (Multiple Model Method) (shell)**, **Component 1 (comp1)** > **Shell 2 (Multiple Model Method) (shell2)**, and **Component 1 (comp1)** > **Layered Shell (Layerwise Theory) (lshell2)**.

5 Click **Disable in Model**.

6 In the tree, select **Component 1 (comp1)** > **Multiphysics** > **Layered Shell–Shell Connection 1 (lssh1)** and **Component 1 (comp1)** > **Multiphysics** > **Layered Shell–Shell Connection 2 (lssh2)**.

7 Click **Disable in Model**.

Solution 5 (sol5)

In the **Study** toolbar, click **Show Default Solver**.

Solution 5 (sol5)

1 In the **Model Builder** window, expand the **Study: Eigenfrequency (ESL Theory)** > **Solver Configurations** > **Solution 5 (sol5)** node, then click **Eigenvalue Solver 1**.

2 In the **Settings** window for **Eigenvalue Solver**, locate the **Output** section.

3 From the **Scaling of eigenvectors** list, choose **RMS**.

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

RESULTS

Layered Material 5

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Layered Material 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Eigenfrequency (ESL Theory)/Solution 5 (sol5)**.

Mode Shapes (ESL Theory)

1 In the **Model Builder** window, right-click **Mode Shapes (Layerwise Theory)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shapes (ESL Theory)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

Surface 1

1 In the **Model Builder** window, expand the **Mode Shapes (ESL Theory)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `shell3.disp`.

Deformation 1

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

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

3 In the **x-component** text field, type `u5`.

4 In the **y-component** text field, type `v5`.

5 In the **z-component** text field, type `w5`.

Mode Shapes (ESL Theory)

1 In the **Model Builder** window, under **Results** click **Mode Shapes (ESL Theory)**.

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

3 From the **Dataset** list, choose **Layered Material 5**.

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

5 In the **Mode Shapes (ESL Theory)** toolbar, click **Plot**.

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 the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 6

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `10`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Layered Shell (Multiple Model Method) (lshell)**, **Component 1 (comp1)** > **Shell 1 (Multiple Model Method) (shell)**, **Component 1 (comp1)** > **Shell 2 (Multiple Model Method) (shell2)**, and **Component 1 (comp1)** > **Layered Shell (Layerwise Theory) (lshell2)**.

5 Click **Disable in Model**.

6 In the tree, select **Component 1 (comp1)** > **Multiphysics** > **Layered Shell–Shell Connection 1 (lssh1)** and **Component 1 (comp1)** > **Multiphysics** > **Layered Shell–Shell Connection 2 (lssh2)**.

7 Click **Disable in Model**.

8 In the **Model Builder** window, click **Study 6**.

9 In the **Settings** window for **Study**, type `Study: Frequency (ESL Theory)` in the **Label** text field.

10 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

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

RESULTS

Layered Material 6

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Layered Material 4** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Frequency (ESL Theory)/Solution 6 (sol6)**.

Surface 5

1 In the **Model Builder** window, under **Results** > **Stress** right-click **Surface 4** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Layered Material 6**.

4 Locate the **Expression** section. In the **Expression** text field, type `abs(shell3.sllGp11)`.

5 Locate the **Plot Array** section. In the **Index** text field, type `2`.

Deformation 1

1 In the **Model Builder** window, expand the **Surface 5** node, then click **Deformation 1**.

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

3 In the **x-component** text field, type `u5`.

4 In the **y-component** text field, type `v5`.

5 In the **z-component** text field, type `w5`.

Stress

In the **Model Builder** window, under **Results** click **Stress**.

Table Annotation 1

1 In the **Stress** 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	z-coordinate	Annotation
-0.8	0.5	0	Multiple Model Method
-0.8	1.6	0	Layerwise Theory
-0.8	2.7	0	ESL Theory

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

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

7 In the **Stress** toolbar, click **Plot**.

Layered Material Slice 3

1 In the **Model Builder** window, under **Results** > **Displacement, Slice** right-click **Layered Material Slice 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Frequency (ESL Theory)/Solution 6 (sol6)**.

4 Locate the **Expression** section. In the **Expression** text field, type `shell3.disp`.

5 In the **Displacement, Slice** toolbar, click **Plot**.

Table Annotation 1

In the **Model Builder** window, under **Results** > **Stress** right-click **Table Annotation 1** and choose **Copy**.

Table Annotation 1

1 In the **Model Builder** window, right-click **Displacement, Slice** and choose **Paste Table Annotation**.

2 In the **Displacement, Slice** toolbar, click **Plot**.

Comparison: Stress

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

2 In the **Settings** window for **Evaluation Group**, type `Comparison: Stress` in the **Label** text field.

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

4 Click to expand the **Format** section. From the **Include parameters** list, choose **Off**.

Volume Maximum 1

1 Right-click **Comparison: Stress** and choose **Maximum** > **Volume Maximum**.

2 In the **Settings** window for **Volume Maximum**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(lshell.sllGp11)`	GPa	Absolute value of stress tensor component

Volume Maximum 2

1 Right-click **Volume Maximum 1** and choose **Duplicate**.

2 In the **Settings** window for **Volume Maximum**, locate the **Data** section.

3 From the **Dataset** list, choose **Layered Material 4**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`abs(lshell2.sllGp11)`	GPa	Absolute value of stress tensor component

Volume Maximum 3

1 Right-click **Volume Maximum 2** and choose **Duplicate**.

2 In the **Settings** window for **Volume Maximum**, locate the **Data** section.

3 From the **Dataset** list, choose **Layered Material 6**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`abs(shell3.sllGp11)`	GPa	Absolute value of stress tensor component

5 In the **Comparison: Stress** toolbar, click **Evaluate**.

Comparison: Maximum Displacement

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

2 In the **Settings** window for **Evaluation Group**, type `Comparison: Maximum Displacement` in the **Label** text field.

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

4 Locate the **Format** section. From the **Include parameters** list, choose **Off**.

Volume Maximum 1

1 Right-click **Comparison: Maximum Displacement** and choose **Maximum** > **Volume Maximum**.

2 In the **Settings** window for **Volume Maximum**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`lshell.disp`	m	Displacement magnitude

Volume Maximum 2

1 Right-click **Volume Maximum 1** and choose **Duplicate**.

2 In the **Settings** window for **Volume Maximum**, locate the **Data** section.

3 From the **Dataset** list, choose **Layered Material 4**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`lshell2.disp`	`m`	`Displacement magnitude`

Volume Maximum 3

1 Right-click **Volume Maximum 2** and choose **Duplicate**.

2 In the **Settings** window for **Volume Maximum**, locate the **Data** section.

3 From the **Dataset** list, choose **Layered Material 6**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`shell3.disp`	`m`	`Displacement magnitude`

5 In the **Comparison: Maximum Displacement** toolbar, click **Evaluate**.

Comparison: Eigenfrequency

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

2 In the **Settings** window for **Evaluation Group**, type `Comparison: Eigenfrequency` in the **Label** text field.

3 Locate the **Format** section. From the **Include parameters** list, choose **Off**.

Global Evaluation 1

1 Right-click **Comparison: Eigenfrequency** 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
`freq`	`Hz`	`Frequency`

Global Evaluation 2

1 Right-click **Global Evaluation 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Eigenfrequency (Layerwise Theory)/Solution 3 (sol3)**.

Global Evaluation 3

1 Right-click **Global Evaluation 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Eigenfrequency (ESL Theory)/Solution 5 (sol5)**.

4 In the **Comparison: Eigenfrequency** toolbar, click **Evaluate**.

In the first two studies, disable the newly added physics interfaces so that the studies can be run as originally configured.

STUDY: EIGENFREQUENCY (MULTIPLE MODEL METHOD)

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study: Eigenfrequency (Multiple Model Method)** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Layered Shell (Layerwise Theory) (lshell2)** and **Component 1 (comp1)** > **Shell (ESL Theory) (shell3)**.

5 Click **Disable in Model**.

STUDY: FREQUENCY (MULTIPLE MODEL METHOD)

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study: Frequency (Multiple Model Method)** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Layered Shell (Layerwise Theory) (lshell2)** and **Component 1 (comp1)** > **Shell (ESL Theory) (shell3)**.

5 Click **Disable in Model**.

STUDY: EIGENFREQUENCY (LAYERWISE THEORY)

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study: Eigenfrequency (Layerwise Theory)** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Shell (ESL Theory) (shell3)**.

4 Click **Disable in Model**.

STUDY: FREQUENCY (LAYERWISE THEORY)

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study: Frequency (Layerwise Theory)** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Shell (ESL Theory) (shell3)**.

4 Click **Disable in Model**.