Created in COMSOL Multiphysics 6.1

Analysis of a Composite Blade Using a Multiple Model Method

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 **Layered Linear Elastic Material** 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.

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 face of the blade. The load is entered as a normal component of the boundary load using the expression

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

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

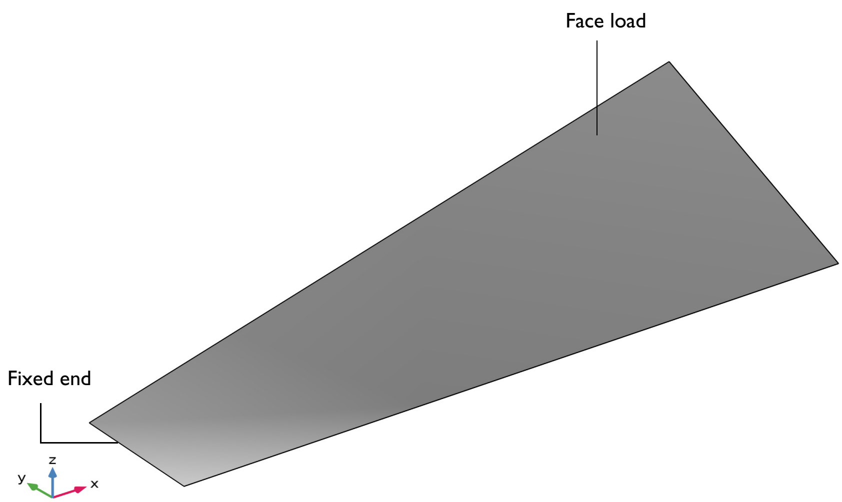

Figure 1: Geometry of the blade, with boundary conditions and loading.

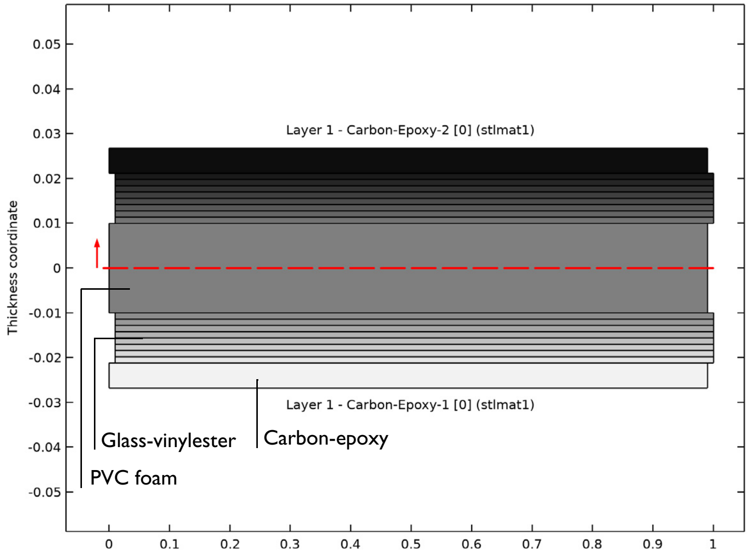

Figure 2: Through-thickness view of the laminate.

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 orthotropic material properties are given in Table 1.

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

Material property	**Value**
$\{E_1, E_2, E_3\}$	{139,9,9}(GPa)
$\{G_{12}, G_{23}, G_{13}\}$	{5.5,5.5,5.5}(GPa)
$\{\nu_{12}, \nu_{23}, \nu_{13}\}$	{0.32,0.32,0.32}

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 orthotropic material properties are given in Table 2.

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

Material property	**Value**
$\{E_1, E_2, E_3\}$	{41,9,9}(GPa)
$\{G_{12}, G_{23}, G_{13}\}$	{4.1,4.1,4.1}(GPa)
$\{\upsilon_{12}, \upsilon_{23}, \upsilon_{13}\}$	{0.3,0.3,0.3}

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.

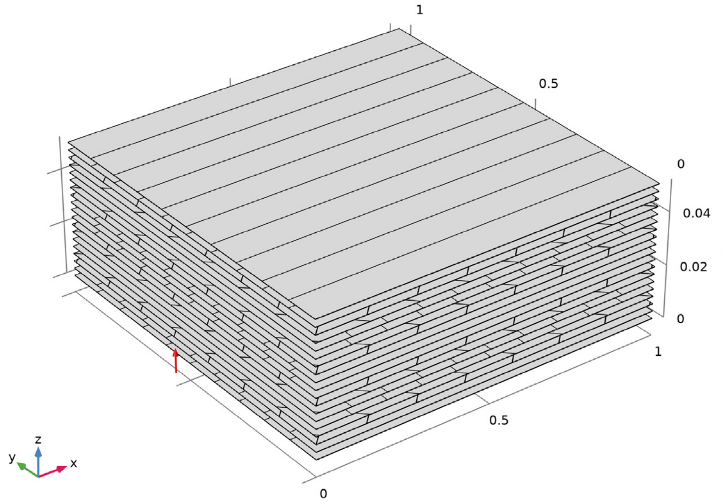

Figure 3: Stacking sequence for the laminate showing the fiber orientation of each layer, from bottom to top.

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

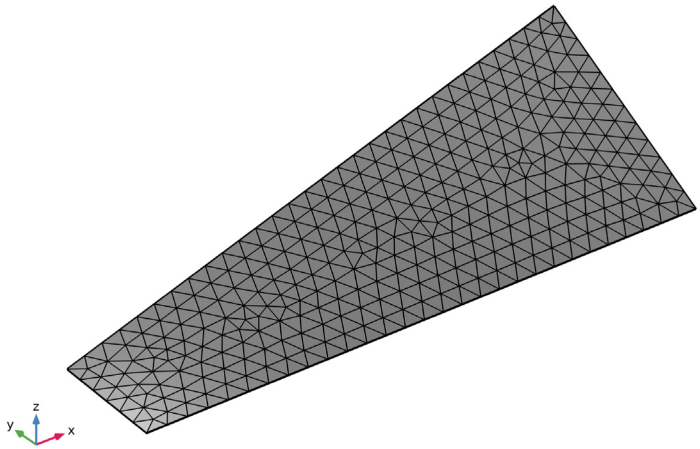

Figure 4: The mesh in the base selection.

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. However 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)
13.015	13.016	13.653
35.351	35.368	44.565
49.833	49.879	59.521

TABLE 5: COMPARISON OF EIGENFREQUENCIES.

Multiple model method (Hz)	Layerwise (Hz)	ESL (Hz)
101.02	101.05	113.32
137.81	137.89	179.14
181.37	181.62	265.99

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 is shown in Table 6.

TABLE 6: PERFORMANCE COMPARISON FOR THE EIGENFREQUENCY STUDIES.

Comparison	Multiple model method	Layerwise	ESL
Number of DOFs	167700	269100	7800
Solution time (s)	40	71	17
Relative solution time	0.56	1	0.23

FREQUENCY-DOMAIN ANALYSIS

The von Mises peak stress distribution for 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 (30% difference in peak value).

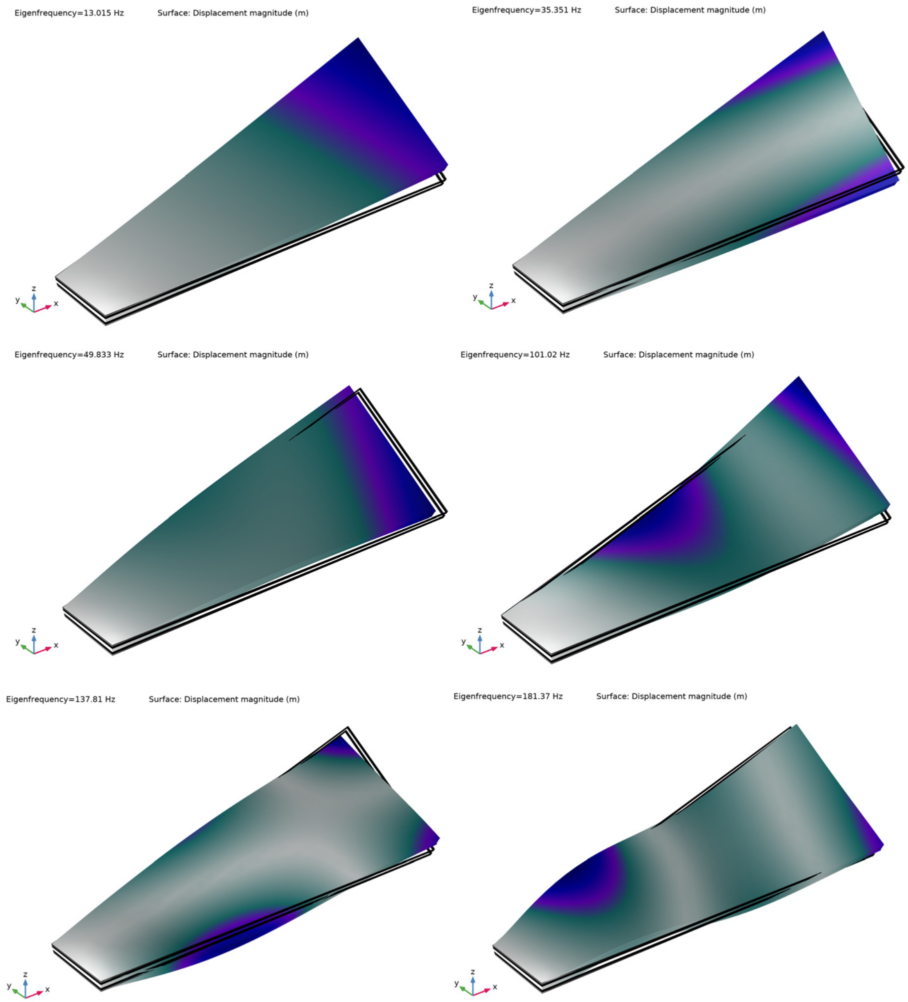

Figure 5: The first six mode shapes and corresponding eigenfrequencies of the composite blade, using the multiple model method.

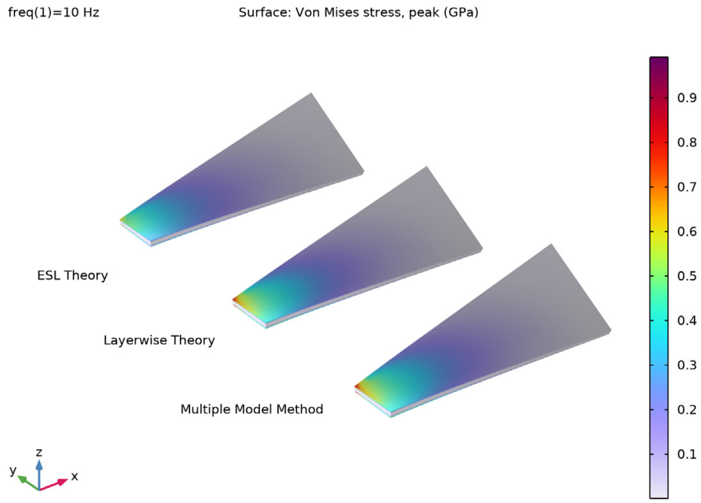

Figure 6: Peak von Mises stress distribution in the composite blade.

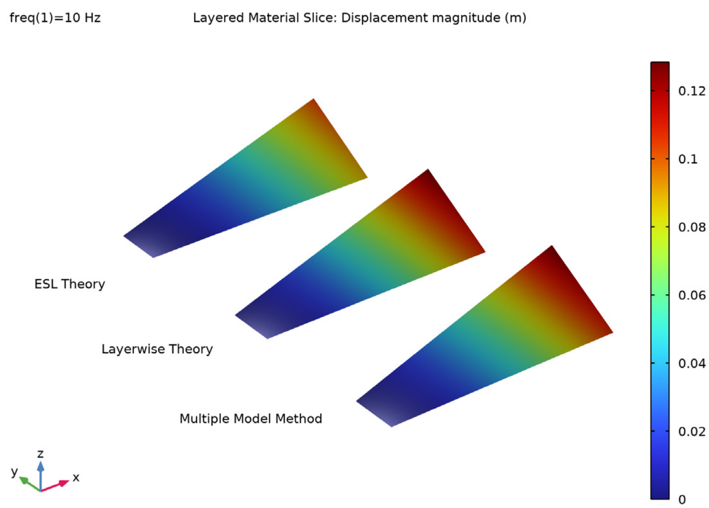

Figure 7: Displacement in the top layer of the composite blade.

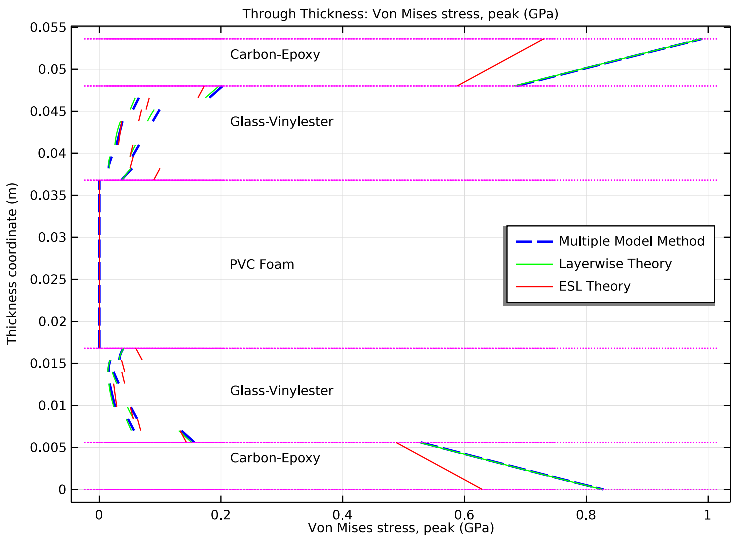

Figure 8: Through-thickness stress variation in von Mises peak stress at a particular location.

Figure 8 shows the distribution of peak von Mises stress in the through-thickness direction at a particular location. The results using the multiple model method and the layerwise theory are in close agreement, while the ESL theory produces significantly different results. This is expected, as the ESL theory is incapable of accurately computing interlaminar shear stresses in thick composites.

The performance of three methods with frequency-domain study is shown in Table 7.

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

Comparison	Multiple model method	Layerwise	ESL
Number of DOFs	167700	269100	7800
Solution time (s)	36	60	18
Relative solution time	0.6	1	0.3

Notes About the COMSOL Implementation

- To model composites you can use two approaches: You can use either the **Layered Shell** interface, that uses the layerwise theory, or the **Layered Linear Elastic Material** 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, however, so the **Layered Shell** interface can be 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,* Second Edition, CRC Press, 2004.

Application Library path: `Composite_Materials_Module/Tutorials/composite_blade_multiple_model_method`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

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

3 Click **Add**.

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

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>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	1e4[Pa]	10000 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 (deg)	Thickness	Mesh elements
Layer 1	Material: Carbon-Epoxy (mat1)	0.0	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 (deg)	Thickness	Mesh elements
Layer 1	Material: Glass-Vinylester (mat2)	0.0	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 (deg)	Thickness	Mesh elements
Layer 2	Material: Glass-Vinylester (mat2)	45	th	1
Layer 3	Material: Glass-Vinylester (mat2)	-45	th	1
Layer 4	Material: Glass-Vinylester (mat2)	90	th	1
Layer 5	Material: Glass-Vinylester (mat2)	90	th	1
Layer 6	Material: Glass-Vinylester (mat2)	-45	th	1
Layer 7	Material: Glass-Vinylester (mat2)	45	th	1
Layer 8	Material: Glass-Vinylester (mat2)	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.

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 (deg)	Thickness	Mesh elements
`Layer 1`	Material: PVC Foam (mat3)	`0.0`	`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 **zy-plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **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** check box.

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 **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.

You will now model the 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** check box.

4 Click **Clear All**.

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

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** check box.

Face Load, Top

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

2 In the **Settings** window for **Face Load**, type `Face Load, Top` 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. Specify the $\mathbf{F}_A$ vector as

0	x
0	y
-p0*exp(-j*mn*atan2(Y,X))	z

Face Load, Bottom

1 Right-click **Face Load, Top** and choose **Duplicate**.

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

3 Locate the **Interface Selection** section. From the **Apply to** list, choose **Bottom interface**.

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 box, in the tree, select the check box 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** check box.

Layered Linear Elastic Material 1

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

2 Select Boundary 1 only.

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

4 Clear the **Use all layers** check box.

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 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 box, click **OK**.

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

Layered Linear Elastic Material 1

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

2 In the **Settings** window for **Layered Linear Elastic Material**, 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	{Evector1, Evector2, Evector3}	`{139e9, 9e9, 9e9}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{0.32, 0.32, 0.32}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{5.5e9, 5.5e9, 5.5e9}`	N/m²	Orthotropic
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	{Evector1, Evector2, Evector3}	`{41e9, 9e9, 9e9}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{0.3, 0.3, 0.3}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{4.1e9, 4.1e9, 4.1e9}`	N/m²	Orthotropic
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	{Evector1, Evector2, Evector3}	`{250e6, 250e6, 250e6}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{0.35, 0.35, 0.35}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{92.6e6, 92.6e6, 92.6e6}`	N/m^2	Orthotropic
Density	rho	200	kg/m^3	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

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.

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

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

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build All**.

STUDY: EIGENFREQUENCY (MULTIPLE MODEL METHOD)

Switch off the generation of default plots in every study to avoid plot cluttering. 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** check box.

4 In the **Home** 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 Shape (Multiple Model Method)

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

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape (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** check box.

Surface 1

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

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

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Aurora>AuroraAustralis** in the tree.

5 Click **OK**.

Deformation 1

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

Surface 2

1 In the **Model Builder** window, under **Results>Mode Shape (Multiple Model Method)** 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**.

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 Shape (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`.

Mode Shape (Multiple Model Method)

1 In the **Model Builder** window, under **Results** click **Mode Shape (Multiple Model Method)**.
2 In the **Mode Shape (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 **Add Study** 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** check box.

6 In the **Home** 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)**.

Mises Peak Stress

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

2 In the **Settings** window for **3D Plot Group**, type `Mises Peak 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. Clear the **Plot dataset edges** check box.

Surface 1

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

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

3 In the **Expression** text field, type `lshell.mises_peak`.

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

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

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

7 Click **OK**.

Deformation 1

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

Surface 2

1 In the **Model Builder** window, under **Results>Mises Peak 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 `shell.mises_peak`.

Surface 3

Right-click **Results>Mises Peak 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 `shell2.mises_peak`.

Mises Peak Stress

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

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

Displacement, Slice

1 In the **Home** toolbar, click **Add Plot Group** and choose **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. Clear the **Plot dataset edges** check box.

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**.

You will now model the composite blade using the layerwise theory. To this end, you will use the **Layered Shell** interface.

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 box, 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** check box.

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 **Add Study** in the window toolbar.

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

STUDY 3

Step 1: Eigenfrequency

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

2 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Layered Shell (Multiple Model Method) (lshell)		Automatic (Stationary)
Shell 1 (Multiple Model Method) (shell)		Automatic (Stationary)
Shell 2 (Multiple Model Method) (shell2)		Automatic (Stationary)

3 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Layered Shell-Shell Connection 1 (lssh1)		Automatic (Stationary)
Layered Shell-Shell Connection 2 (lssh2)		Automatic (Stationary)

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

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

6 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

7 In the **Home** 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 Shape (Layerwise Theory)

1 In the **Model Builder** window, right-click **Mode Shape (Multiple Model Method)** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Mode Shape (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 Shape (Layerwise Theory)** node.

Surface 2, Surface 3

1 In the **Model Builder** window, under **Results>Mode Shape (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 Shape (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, expand the **Surface 1** 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`.

Mode Shape (Layerwise Theory)

1 In the **Model Builder** window, under **Results** click **Mode Shape (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 Shape (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 **Add Study** 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. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Layered Shell (Multiple Model Method) (lshell)		Automatic (Stationary)
Shell 1 (Multiple Model Method) (shell)		Automatic (Stationary)
Shell 2 (Multiple Model Method) (shell2)		Automatic (Stationary)

4 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Layered Shell-Shell Connection 1 (lssh1)		Automatic (Stationary)
Layered Shell-Shell Connection 2 (lssh2)		Automatic (Stationary)

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

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

7 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

8 In the **Home** 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)**.

Mises Peak Stress

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

2 In the **Settings** window for **3D Plot Group**, click to expand the **Plot Array** section.

3 Select the **Enable** check box.

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

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

Surface 2

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

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

3 Select the **Manual indexing** check box.

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** check box.

Surface 4

1 Right-click **Results>Mises Peak 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 `lshell2.mises_peak`.

5 Locate the **Plot Array** section. Select the **Manual indexing** check box.

6 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`.

Mises Peak Stress

1 In the **Model Builder** window, under **Results** click **Mises Peak Stress**.
2 In the **Mises Peak Stress** toolbar, click **Plot**.

Displacement, Slice

1 In the **Model Builder** window, click **Displacement, Slice**.
2 In the **Settings** window for **3D Plot Group**, click to expand the **Plot Array** section.
3 Select the **Enable** check box.
4 From the **Array axis** list, choose **y**.
5 In the **Relative padding** text field, type `1`.

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**.

You will now model the composite blade using the equivalent single layer theory. To this end, you will use the **Shell** interface.

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 box, click **OK**.

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

Layered Linear Elastic Material 1

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

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

3 Select the **Use all layers** check box.

Face Load, Top

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

2 In the **Settings** window for **Face Load**, type `Face Load, Top` 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. Specify the $\mathbf{F}_A$ vector as

0	x
0	y
-p0*exp(-j*mn*atan2(Y,X))	z

Face Load, Bottom

1 Right-click **Face Load, Top** and choose **Duplicate**.

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

3 Locate the **Through-Thickness Location** section. From the list, choose **Bottom surface**.

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 **Add Study** in the window toolbar.

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

STUDY 5

Step 1: Eigenfrequency

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

2 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Layered Shell (Multiple Model Method) (lshell)		Automatic (Stationary)
Shell 1 (Multiple Model Method) (shell)		Automatic (Stationary)
Shell 2 (Multiple Model Method) (shell2)		Automatic (Stationary)
Layered Shell (Layerwise Theory) (lshell2)		Automatic (Stationary)

3 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Layered Shell-Shell Connection 1 (lssh1)		Automatic (Stationary)
Layered Shell-Shell Connection 2 (lssh2)		Automatic (Stationary)

4 In the **Model Builder** window, click **Study 5**.

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

6 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

7 In the **Home** 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 Shape (ESL Theory)

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

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape (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 Shape (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, expand the **Surface 1** 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`.

Mode Shape (ESL Theory)

1 In the **Model Builder** window, under **Results** click **Mode Shape (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 Shape (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 **Add Study** 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. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Layered Shell (Multiple Model Method) (lshell)		Automatic (Stationary)
Shell 1 (Multiple Model Method) (shell)		Automatic (Stationary)
Shell 2 (Multiple Model Method) (shell2)		Automatic (Stationary)
Layered Shell (Layerwise Theory) (lshell2)		Automatic (Stationary)

4 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Layered Shell-Shell Connection 1 (lssh1)		Automatic (Stationary)
Layered Shell-Shell Connection 2 (lssh2)		Automatic (Stationary)

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

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

7 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

8 In the **Home** 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>Mises Peak 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 `shell3.mises_peak`.

5 Locate the **Plot Array** section. Select the **Manual indexing** check box.

6 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`.

Mises Peak Stress

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

Table Annotation 1

1 In the **Mises Peak 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** check box.

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

7 In the **Mises Peak 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>Mises Peak 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**.

Mises Peak Stress, Through Thickness

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

2 In the **Settings** window for **1D Plot Group**, type `Mises Peak Stress, Through Thickness` 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 **Legend** section. From the **Position** list, choose **Middle right**.

Through Thickness 1

1 In the **Mises Peak Stress, Through Thickness** toolbar, click **More Plots** and choose **Through Thickness**.

2 Select Point 2 only.

3 In the **Settings** window for **Through Thickness**, locate the **x-Axis Data** section.

4 In the **Expression** text field, type `lshell.mises_peak`.

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

6 Locate the **y-Axis Data** section. Find the **Interface positions** subsection. From the **Show interface positions** list, choose **Interfaces between layered materials**.

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

8 From the **Color** list, choose **Blue**.

9 From the **Width** list, choose **2**.

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

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

12 In the table, enter the following settings:

Legends
`Multiple Model Method`

Through Thickness 2

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

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

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

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

5 Locate the **Legends** section. Clear the **Show legends** check box.

Through Thickness 3

1 Right-click **Through Thickness 2** and choose **Duplicate**.

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

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

Through Thickness 4

1 In the **Model Builder** window, under **Results>Mises Peak Stress, Through Thickness** right-click **Through Thickness 1** and choose **Duplicate**.

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

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

4 Locate the **x-Axis Data** section. In the **Expression** text field, type `lshell2.mises_peak`.

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.

7 From the **Color** list, choose **Green**.

8 From the **Width** list, choose **1**.

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

Legends
`Layerwise Theory`

Through Thickness 5

1 Right-click **Through Thickness 4** and choose **Duplicate**.

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

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

4 Locate the **x-Axis Data** section. In the **Expression** text field, type `shell3.mises_peak`.

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

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

Legends
ESL Theory

Mises Peak Stress, Through Thickness

In the **Model Builder** window, click **Mises Peak Stress, Through Thickness**.

Table Annotation 1

1 In the **Mises Peak Stress, Through Thickness** toolbar, click **More Plots** and choose **Table Annotation**.

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

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

4 In the table, enter the following settings:

x-coordinate	**y-coordinate**	**Annotation**
0.2	0.005	Carbon-Epoxy
0.2	0.013	Glass-Vinylester
0.2	0.028	PVC Foam
0.2	0.045	Glass-Vinylester
0.2	0.053	Carbon-Epoxy

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

6 In the **Mises Peak Stress, Through Thickness** toolbar, click **Plot**.

Maximum Mises Peak Stress Comparison

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

2 In the **Settings** window for **Evaluation Group**, type `Maximum Mises Peak Stress Comparison` 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 **Maximum Mises Peak Stress Comparison** 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.mises_peak	GPa	Von Mises stress, peak

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.mises_peak	GPa	Von Mises stress, peak

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.mises_peak	GPa	Von Mises stress, peak

5 In the **Maximum Mises Peak Stress Comparison** toolbar, click **Evaluate**.

Maximum Displacement Comparison

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

2 In the **Settings** window for **Evaluation Group**, type `Maximum Displacement Comparison` 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 **Maximum Displacement Comparison** 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 **Maximum Displacement Comparison** toolbar, click **Evaluate**.

Eigenfrequency Comparison

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

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

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

Global Evaluation 1

1 Right-click **Eigenfrequency Comparison** 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 **Eigenfrequency Comparison** 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 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Layered Shell (Layerwise Theory) (lshell2)		Automatic (Stationary)
Shell (ESL Theory) (shell3)		Automatic (Stationary)

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 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Layered Shell (Layerwise Theory) (lshell2)		Automatic (Stationary)
Shell (ESL Theory) (shell3)		Automatic (Stationary)

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 table, enter the following settings:

Physics interface	Solve for	Equation form
Shell (ESL Theory) (shell3)		Automatic (Stationary)

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 table, enter the following settings:

Physics interface	Solve for	Equation form
Shell (ESL Theory) (shell3)		Automatic (Stationary)