Model created in COMSOL Multiphysics 6.4

Dome Tweeter with Composite Diaphragm — Eigenfrequency Analysis

Introduction

Fiber composites are widely used in industrial applications. Compared to more traditional metallic engineering materials, fiber composites often have superior specific stiffness and strength properties. Properties like toughness, stiffness, and weight can often be tailored to specific applications such as a diaphragm in loudspeakers.

Analyzing resonances is essential when designing loudspeakers. Resonances can come from various sources in a loudspeaker, the diaphragm being a significant one. Composite diaphragms can shift, dampen, and control the resonance, while also giving breakup symmetry modes which can improve the sound quality considerably.

Modeling individual fibers in every layer in the laminate is unfeasible. A simplified micromechanics model of a composite with specific microstructures is instead used to estimate the homogenized elastic properties of a single layer. In this example, the homogenized materials for two different composites along with a traditional material (titanium) are used in the eigenfrequency analysis of a diaphragm. Two approaches are used to model the diaphragm, namely the Layerwise (LW) theory and the Equivalent Single Layer (ESL) theory.

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

Model Definition

This model performs the following types of analyses. The model is divided into three parts:

- Micromechanical analysis of the composites
- Eigenfrequency analysis of the diaphragm using the Equivalent Single Layer theory
- Eigenfrequency analysis of the diaphragm using the Layerwise theory

Eigenfrequencies and mode shapes are computed and compared using both theories.

MICROMECHANICS ANALYSIS

In the first part, a micromechanical analysis of two different repeating unit cells (RUCs) is performed in order to obtain the homogenized material properties. Figure 1 shows the geometries of the RUCs. For both RUCs, the fiber volume fraction is in the range of 0.65–0.72.

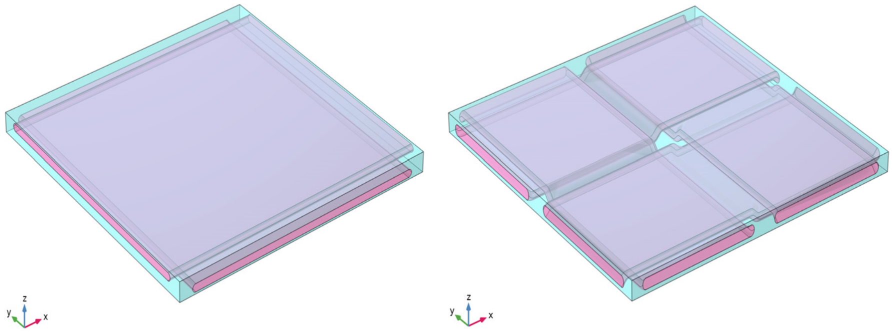

Figure 1: Unit cell geometries of a bidirectional noncrimp fiber composite (left) and a bidirectional spread-tow fiber composite (right).

Fiber and Resin Properties

A layer of the laminate is made of AS-4 carbon fiber and epoxy polymer. The carbon fiber is assumed to be transversely isotropic and the epoxy resin is assumed to be isotropic. Both materials are built-in materials in the **Composites** material library. The fiber and matrix material properties are given in Table 1 and Table 2, respectively.

TABLE 1: CARBON FIBER MATERIAL PROPERTIES.

Material Property	Value
$\{E_1, E_2\}$	{235,15} GPa
G_{12}	27 GPa
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.2, 0.0714}
ρ	1810 kg/m^3

TABLE 2: EPOXY RESIN MATERIAL PROPERTIES.

Material Property	Value
E	3.25 GPa
υ	0.265
ρ	1250 kg/m^3

Cell Periodicity

To perform a micromechanical analysis, a **Cell Periodicity** node in the Solid Mechanics interface is used. The **Cell Periodicity** node is used to apply periodic boundary conditions to the three pairs of faces of the unit cell.

To extract the homogenized elasticity matrix for a layer, the unit cell needs to be analyzed for six different load cases. The **Average Strain** periodicity type needs to be selected to obtain the homogenized elasticity matrix. This is automatically done with the help of the action buttons in the **Cell Periodicity** node, which has three action buttons in the toolbar of the section called **Periodicity Type**: **Create Load Groups and Study**, **Create Material by Reference**, and **Create Material by Value**. The action button **Create Load Groups and Study** generates six different load groups and a stationary study with six load cases. The action button **Create Material by Reference** generates a **Global Material** with an elasticity matrix corresponding to that of the homogenized material in terms of variables. The action button **Create Material by Value** generates a **Global Material** with an elasticity matrix corresponding to that of the homogenized material in terms of numbers after the study computed. The generated global material can be used to define the properties of individual layers in a composite laminate.

EIGENFREQUENCY ANALYSIS

Equivalent Single Layer (ESL) Theory

In the equivalent single layer (ESL) theory, the degrees of freedom are the displacements and rotations on the midplane of the laminate. From a constitutive equation point of view, this theory is similar to 3D shell elasticity. Through-thickness homogenized material properties of the laminate are used. It is therefore computationally less expensive than the layerwise theory. It can be used for the modeling of thin to moderately thick laminates with good accuracy.

Layerwise (LW) Theory

In the layerwise theory, the degrees of freedom are the displacements (u, v, w) available on the reference surface (or modeled surface) as well as in the through-thickness direction. From a constitutive equation point of view, this theory is similar to 3D solid elasticity. The layerwise theory is useful for modeling of thick composite laminates.

Material Properties

The material properties of two composite materials are obtained from the first analysis of the model. The material properties of titanium are presented in Table 3.

TABLE 3: TITANIUM MATERIAL PROPERTIES.

Material Property	**Value**
E	105 GPa
υ	0.33
ρ	4940 kg/m^3

Geometry and Boundary Conditions

Figure 2 shows the model geometry of a composite diaphragm with a radius of 65 mm and a thickness of 0.1 mm. The outer edges of the diaphragm are fixed.

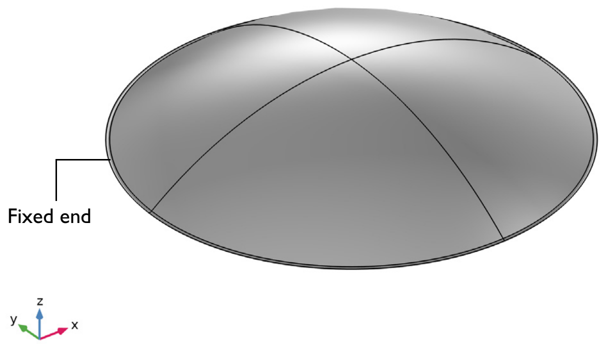

Figure 2: Geometry of the diaphragm showing boundary conditions.

Results and Discussion

In the first part of this model, a micromechanical analysis is carried out to get the homogenized materials for two composites.

In the second part of the model, six eigenmodes and eigenfrequencies of a diaphragm using titanium, composite material 1, and composite material 2 are presented. The first eigenmode of the diaphragm is of interest as its excitation by the driver frequency will determine the sound quality. The first eigenmode using three different materials with the **Shell** interface is presented in Figure 3. The first eigenmode of the diaphragm with the **Layered Shell** interface is presented in Figure 4. From both figures it is clear that the conventional material like titanium produces symmetric eigenmode, while the bidirectional composite materials breaks the symmetry. Furthermore, the eigenfrequencies of the composite materials are almost 10% to 25% higher than the titanium with 66% lower mass.

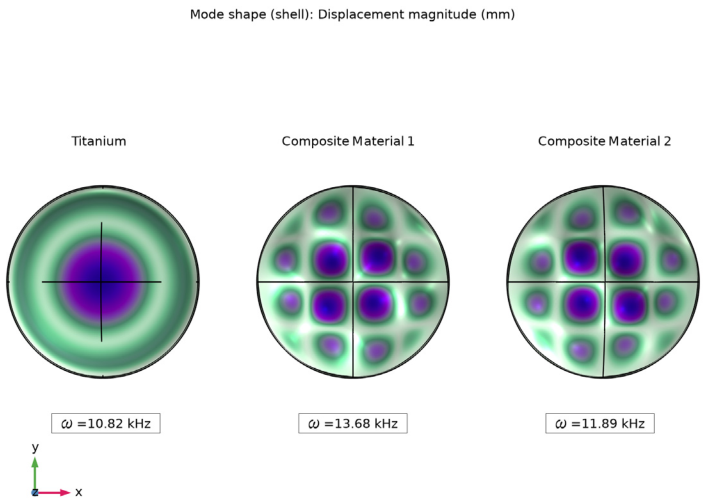

Figure 3: First mode shape of the diaphragm for titanium, composite material 1, and composite material 2 from left to right. The mode shapes are obtained using a Shell interface.

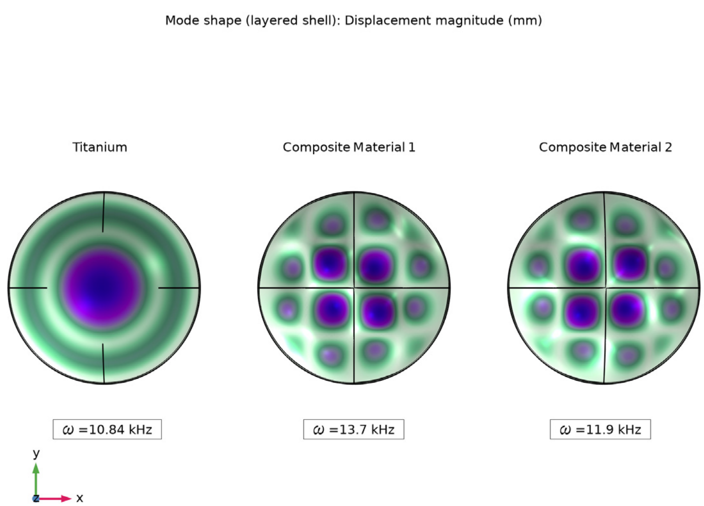

Figure 4: First mode shape of the diaphragm for titanium, composite material 1, and composite material 2 from left to right. The mode shapes are obtained using a Layered Shell interface.

Table 4 shows the first eigenfrequency with three different materials. The eigenfrequencies obtained using the LW and ESL theories match within 1%.

TABLE 4: COMPARISON OF EIGENFREQUENCIES.

Material	Eigenfrequencies from ESL theory (Hz)	Eigenfrequencies from layerwise theory (Hz)
Titanium	10823	10836
Composite Material 1	13682	13697
Composite Material 2	11890	11905

The mass of a diaphragm using three different materials is shown in Table 5. The composite materials are almost 66% lighter than the titanium.

TABLE 5: COMPARISON OF DIAPHRAGM MASS.

Titanium (gm)	Composite material 1 (gm)	Composite material 2 (gm)
2.8982	0.96852	0.95516

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 in the Layered Shell and Shell interfaces. 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.
- In order to run the analysis for various layered materials and compare the results, all the layered materials can be defined using a **Switch** node in **Global Materials**. This **Switch** node

can be selected in the **Layered Material Link** node and a **Material Sweep** node is added in the study.

- You can either use the *Layerwise (LW)* theory-based Layered Shell interface or the *Equivalent Single Layer (ESL)* theory-based **Linear Elastic Material, Layered** node in Shell interface.
- The built-in **Composites** material library contains data for fiber and matrix constituents as well as for unidirectional and bidirectional laminae.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/composite_dome_tweeter_eigen`

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** > **Solid Mechanics (solid)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

5 Click **Add**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`th`	`0.1[mm]`	1E-4 m	`Thickness of diaphragm`
`rd`	`65[mm]`	0.065 m	`Radius of diaphragm`

Modeling Instructions (Micromechanical analysis of the composites)

This section describes how to do a micromechanical analysis of the composites using the **Solid Mechanics** interface.

Add the geometry of each required repeating unit cell (RUC) from the built-in **Part Libraries**.

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts** > **Part Libraries**.

PART LIBRARIES

1 In the **Part Libraries** window, select **COMSOL Multiphysics** > **Unit Cells and RVEs** > **Fiber Composites** > **bidirectional_non_crimp_fiber** in the tree.

2 Click **Add to Model**.

GEOMETRY PARTS

In the **Model Builder** window, under **Global Definitions** right-click **Geometry Parts** and choose **Part Libraries**.

PART LIBRARIES

1 In the **Part Libraries** window, select **COMSOL Multiphysics** > **Unit Cells and RVEs** > **Fiber Composites** > **bidirectional_spread_tow_fiber** in the tree.

2 Click **Add to Model**.

GEOMETRY 1

RUC 1: Bidirectional Non-Crimp Fiber Composite

1 In the **Geometry** toolbar, click **Part Instance** and choose **Bidirectional Non-Crimp Fiber Composite**.

2 In the **Settings** window for **Part Instance**, type `RUC 1: Bidirectional Non-Crimp Fiber Composite` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
wfx	`0.9[mm]`	9E-4 m	Width of fiber strand in X direction
hfx	`0.024[mm]`	2.4E-5 m	Height of fiber strand in X direction
wfy	`0.9[mm]`	9E-4 m	Width of fiber strand in Y direction
hfy	`0.024[mm]`	2.4E-5 m	Height of fiber strand in Y direction
hm	`0.06[mm]`	6E-5 m	Cell height

4 Click **Build Selected**.

RUC 2: Bidirectional Spread-Tow Fiber Composite

1 In the **Geometry** toolbar, click **Part Instance** and choose **Bidirectional Spread-Tow Fiber Composite**.

2 In the **Settings** window for **Part Instance**, type `RUC 2: Bidirectional Spread-Tow Fiber Composite` in the **Label** text field.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
wfx	`0.42[mm]`	4.2E-4 m	Width of fiber strand in X direction
hfx	`0.024[mm]`	2.4E-5 m	Height of fiber strand in X direction
wfy	`0.42[mm]`	4.2E-4 m	Width of fiber strand in Y direction
hfy	`0.024[mm]`	2.4E-5 m	Height of fiber strand in Y direction
hm	`0.06[mm]`	6E-5 m	Cell height

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **ywi** text field, type `2[mm]`.

5 Click **Build All Objects**.

Disable the analysis of the geometry as the remaining small geometric details can be kept.

6 In the **Model Builder** window, click **Geometry 1**.

7 In the **Settings** window for **Geometry**, locate the **Cleanup** section.

8 Clear the **Automatic detection of small details** checkbox.

COMSOL Multiphysics is equipped with built-in material properties for a number of composite constituents. Select the materials needed from the **Composites** material folder in the built-in material library.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Composites > Fiber Constituents > AS-4 carbon fiber**.

3 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

AS-4 carbon fiber (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1) > Materials** click **AS-4 carbon fiber (mat1)**.

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

3 From the **Selection** list, choose **Fiber Strands (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.

AS-4 carbon fiber 1 (mat2)

1 Right-click **Component 1 (comp1) > Materials > AS-4 carbon fiber (mat1)** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Fiber Strands (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

ADD MATERIAL

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

2 In the tree, select **Composites > Matrix Constituents > Epoxy polymer**.

3 Right-click and choose **Add to Component 1 (comp1)**.

MATERIALS

Epoxy polymer (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1) > Materials** click **Epoxy polymer (mat3)**.

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

3 From the **Selection** list, choose **Matrix (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.

Epoxy polymer 1 (mat4)

1 Right-click **Component 1 (comp1)** > **Materials** > **Epoxy polymer (mat3)** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Matrix (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

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

AS-4 carbon fiber (mat1), Epoxy polymer (mat3)

Right-click and choose **Group**.

Materials for RUC 1

In the **Settings** window for **Group**, type `Materials for RUC 1` in the **Label** text field.

AS-4 carbon fiber 1 (mat2), Epoxy polymer 1 (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials**, Ctrl-click to select **AS-4 carbon fiber 1 (mat2)** and **Epoxy polymer 1 (mat4)**.

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

Materials for RUC 2

In the **Settings** window for **Group**, type `Materials for RUC 2` in the **Label** text field.

DEFINITIONS

Base Vector System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.

2 In the **Settings** window for **Base Vector System**, locate the **Base Vectors** section.

3 In the table, enter the following settings:

	x	**y**	**z**
x1	0	1	0
x2	-1	0	0

4 Find the **Simplifications** subsection. Select the **Assume orthonormal** checkbox.

SOLID MECHANICS: RUC 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics: RUC 1` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **All (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics: RUC 1 (solid)** click **Linear Elastic Material 1**.

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

3 From the **Material symmetry** list, choose **Orthotropic**.

Linear Elastic Material 2

1 Right-click **Component 1 (comp1)** > **Solid Mechanics: RUC 1 (solid)** > **Linear Elastic Material 1** and choose **Duplicate**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 7 only.

5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

Cell Periodicity 1

1 In the **Physics** toolbar, click **Domains** and choose **Cell Periodicity**.

2 In the **Settings** window for **Cell Periodicity**, locate the **Periodicity Settings** section.

3 From the **Boundary conditions** list, choose **Average strain**.

4 Locate the **Effective Properties** section. Select the **Compute density** checkbox.

5 Select the **Compute elasticity matrix, standard notation** checkbox.

6 Locate the **Periodicity Settings** section. In the λ_ε text field, type `0.01`.

Change the **Constraint** type to **Weak constraints** in order to obtain a smooth solution. To do this, activate **Advanced Physics Options**.

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

8 In the **Show More Options** dialog, select **Physics** > **Advanced Physics Options** in the tree.

9 In the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

10 Click **OK**.

11 In the **Settings** window for **Cell Periodicity**, click to expand the **Constraint Settings** section.

12 From the **Constraint** list, choose **Weak constraints**.

Boundary Pair 1

1 In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.
2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Pair 1 (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.
4 Right-click **Boundary Pair 1** and choose **Manual Destination Selection**.
5 Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 1, Destination (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.
6 Click to expand the **Constraint Settings** section. From the **Constraint** list, choose **Weak constraints**.

Cell Periodicity 1

In the **Model Builder** window, click **Cell Periodicity 1**.

Boundary Pair 2

1 In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.
2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Pair 2 (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.
4 Right-click **Boundary Pair 2** and choose **Manual Destination Selection**.
5 Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 2, Destination (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.
6 Locate the **Constraint Settings** section. From the **Constraint** list, choose **Weak constraints**.

Cell Periodicity 1

In the **Model Builder** window, click **Cell Periodicity 1**.

Boundary Pair 3

1 In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.
2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Pair 3 (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.
4 Right-click **Boundary Pair 3** and choose **Manual Destination Selection**.
5 Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 3, Destination (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.
6 Locate the **Constraint Settings** section. From the **Constraint** list, choose **Weak constraints**.

In the upper-right corner of the **Periodicity type** section, you find the buttons **Create Load Groups and Study**, **Create Material by Reference**, and **Create Material by Value**. When the **Average strain** option is selected for the computation of the density and elasticity matrix, you can automatically generate load groups, a study, and a material by clicking these buttons.

Cell Periodicity 1

1 In the **Model Builder** window, click **Cell Periodicity 1**.
2 In the **Settings** window for **Cell Periodicity**, click **Automated Model Setup** in the upper-right corner of the **Periodicity Settings** section. From the menu, choose **Create Load Groups and Study** to generate load groups and a study node.

SOLID MECHANICS: RUC 2

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics 2 (solid2)**.
2 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics: RUC 2` in the **Label** text field.
3 Locate the **Domain Selection** section. From the **Selection** list, choose **All (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics: RUC 2 (solid2)** click **Linear Elastic Material 1**.
2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.
3 From the **Material symmetry** list, choose **Orthotropic**.

Linear Elastic Material 2

1 Right-click **Component 1 (comp1)** > **Solid Mechanics: RUC 2 (solid2)** > **Linear Elastic Material 1** and choose **Duplicate**.
2 In the **Settings** window for **Linear Elastic Material**, locate the **Domain Selection** section.
3 Click **Clear Selection**.
4 Select Domains 6 and 8 only.
5 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

Cell Periodicity 1

1 In the **Physics** toolbar, click **Domains** and choose **Cell Periodicity**.
2 In the **Settings** window for **Cell Periodicity**, locate the **Periodicity Settings** section.
3 From the **Boundary conditions** list, choose **Average strain**.

4 Locate the **Effective Properties** section. Select the **Compute density** checkbox.

5 Select the **Compute elasticity matrix, standard notation** checkbox.

6 Locate the **Periodicity Settings** section. In the λ_ε text field, type `0.01`.

7 Locate the **Constraint Settings** section. From the **Constraint** list, choose **Weak constraints**.

Boundary Pair 1

1 In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.

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

3 From the **Selection** list, choose **Pair 1 (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

4 Right-click **Boundary Pair 1** and choose **Manual Destination Selection**.

5 Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 1, Destination (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

6 Locate the **Constraint Settings** section. From the **Constraint** list, choose **Weak constraints**.

Cell Periodicity 1

In the **Model Builder** window, click **Cell Periodicity 1**.

Boundary Pair 2

1 In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.

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

3 From the **Selection** list, choose **Pair 2 (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

4 Right-click **Boundary Pair 2** and choose **Manual Destination Selection**.

5 Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 2, Destination (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

6 Locate the **Constraint Settings** section. From the **Constraint** list, choose **Weak constraints**.

Cell Periodicity 1

In the **Model Builder** window, click **Cell Periodicity 1**.

Boundary Pair 3

1 In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.

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

3 From the **Selection** list, choose **Pair 3 (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

4 Right-click **Boundary Pair 3** and choose **Manual Destination Selection**.

5 Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 3, Destination (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

6 Locate the **Constraint Settings** section. From the **Constraint** list, choose **Weak constraints**.

Cell Periodicity 1

1 In the **Model Builder** window, click **Cell Periodicity 1**.

2 In the **Settings** window for **Cell Periodicity**, click **Automated Model Setup** in the upper-right corner of the **Periodicity Settings** section. From the menu, choose **Create Load Groups and Study** to generate load groups and a study node.

MESH 1

Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `1E-4`.

5 In the **Minimum element size** text field, type `2E-5`.

6 In the **Resolution of narrow regions** text field, type `2`.

7 Click **Build All**.

CELL PERIODICITY STUDY: RUC 1

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

2 In the **Settings** window for **Study**, type `Cell Periodicity Study: RUC 1` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Cell Periodicity Study: RUC 1** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, 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)** > **Solid Mechanics: RUC 2 (solid2)**.

5 Right-click and choose **Disable in Model**.

CELL PERIODICITY STUDY: RUC 2

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

2 In the **Settings** window for **Study**, type `Cell Periodicity Study: RUC 2` in the **Label** text field.

1 In the **Model Builder** window, under **Cell Periodicity Study: RUC 2** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, 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)** > **Solid Mechanics: RUC 1 (solid)**.

5 Right-click and choose **Disable in Model**.

CELL PERIODICITY STUDY: RUC 1

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

CELL PERIODICITY STUDY: RUC 2

Click **Compute**.

RESULTS

Stress (solid)

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

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

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **All (RUC 1: Bidirectional Non-Crimp Fiber Composite)**.

5 Select the **Apply to dataset edges** checkbox.

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

7 In the **Stress (solid)** toolbar, click **Plot**.

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

Stress (solid2)

1 In the **Model Builder** window, click **Stress (solid2)**.

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

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **All (RUC 2: Bidirectional Spread-Tow Fiber Composite)**.

5 Select the **Apply to dataset edges** checkbox.

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

7 In the **Stress (solid2)** toolbar, click **Plot**.

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

Before you do a macromechanical analysis of the composite structure, create homogenized materials from the **Cell Periodicity** features.

SOLID MECHANICS: RUC 1 (SOLID)

Cell Periodicity 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics: RUC 1 (solid)** click **Cell Periodicity 1**.
2 In the **Settings** window for **Cell Periodicity**, click **Automated Model Setup** in the upper-right corner of the **Periodicity Settings** section. From the menu, choose **Create Material by Value** to generate a global material node with computed density and elastic properties.

GLOBAL DEFINITIONS

Homogeneous Material: RUC 1

1 In the **Model Builder** window, expand the **Global Definitions** > **Materials** node, then click **Homogeneous Material (solidcp1mat)**.
2 In the **Settings** window for **Material**, type `Homogeneous Material: RUC 1` in the **Label** text field.

SOLID MECHANICS: RUC 2 (SOLID2)

Cell Periodicity 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics: RUC 2 (solid2)** click **Cell Periodicity 1**.
2 In the **Settings** window for **Cell Periodicity**, click **Automated Model Setup** in the upper-right corner of the **Periodicity Settings** section. From the menu, choose **Create Material by Value** to generate a global material node with computed density and elastic properties.

GLOBAL DEFINITIONS

Homogeneous Material: RUC 2

1 In the **Model Builder** window, under **Global Definitions** > **Materials** click **Homogeneous Material (solid2cp1mat)**.
2 In the **Settings** window for **Material**, type `Homogeneous Material: RUC 2` in the **Label** text field.

Modeling Instructions (Eigenfrequency Analysis of a Diaphragm)

This section describes how to do an eigenfrequency analysis of the diaphragm using the **Shell** and **Layered Shell** interfaces based on the homogenized material properties obtained from the previous section.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **3D**.

GEOMETRY 2

1 In the **Settings** window for **Geometry**, locate the **Units** section.

2 From the **Length unit** list, choose **mm**.

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**.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Circular Arc 1 (ca1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Center** section.

3 In the **yw** text field, type `-30`.

4 Locate the **Radius** section. In the **Radius** text field, type `rd`.

5 Locate the **Angles** section. In the **Start angle** text field, type `90`.

6 In the **End angle** text field, type `51.85535`.

7 Select the **Clockwise** checkbox.

Work Plane 1 (wp1) > Quadratic Bézier 1 (qb1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **xw** to `40.147180487314166`.

4 In row **1**, set **yw** to `21.119506051203402`.

5 In row **2**, set **xw** to `40.7`.

6 In row **2**, set **yw** to `20.680469611083915`.

7 In row **3**, set **xw** to 40.8.

8 In row **3**, set **yw** to 21.119506051203402.

Revolve 1 (rev1)

1 In the **Model Builder** window, right-click **Geometry 2** and choose **Revolve**.

2 In the **Settings** window for **Revolve**, click **Build All Objects**.

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

Add a **Material Switch** node to use three different materials: titanium and two composite materials.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Titanium beta-21S**.

3 Right-click and choose **Add to Global Materials**.

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

GLOBAL DEFINITIONS

Material Switch 1 (sw1)

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

Layered Material 1 (sw1.lmat1)

1 In the **Model Builder** window, right-click **Material Switch 1 (sw1)** and choose **Layered Material**.

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

3 In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
Layer 1	Titanium beta-21S (mat5)	0.0	0 rad	th	2

Layered Material 2 (sw1.lmat2)

1 Right-click **Layered Material 1 (sw1.lmat1)** and choose **Duplicate**.

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

3 In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
`Layer 1`	Homogeneous Material: RUC 1 (solidcp1mat)	`0.0`	0 rad	`th`	2

Layered Material 3 (sw1.lmat3)

1 Right-click **Layered Material 2 (sw1.lmat2)** and choose **Duplicate**.

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

3 In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
`Layer 1`	Homogeneous Material: RUC 2 (solid2cp1mat)	`0.0`	0 rad	`th`	2

MATERIALS

Layered Material Link 1 (llmat1)

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Layers** > **Layered Material Link**.

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

3 Click **Go to Source** for **Coordinate system**.

DEFINITIONS (COMP2)

Boundary System 3 (sys3)

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Definitions** click **Boundary System 3 (sys3)**.

2 In the **Settings** window for **Boundary System**, locate the **Settings** section.

3 Find the **Coordinate names** subsection. From the **Axis** list, choose **x**.

ADD PHYSICS

1 In the **Home** toolbar, click **Windows** and choose **Add Physics**.

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

3 In the tree, select **Structural Mechanics** > **Shell (shell)**.

4 Click the **Add to Component 2** button in the window toolbar.

5 In the tree, select **Structural Mechanics** > **Layered Shell (lshell)**.

6 Click the **Add to Component 2** button in the window toolbar.

7 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

SHELL (SHELL)

Linear Elastic Material, Layered 1

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

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

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

4 Locate the **Linear Elastic Material** section. From the **Material symmetry** list, choose **Anisotropic**.

Fixed Constraint 1

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

2 Select Edges 2, 3, 8, and 15 only.

LAYERED SHELL (LSHELL)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Layered Shell (lshell)** click **Linear Elastic Material 1**.

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

3 From the **Material symmetry** list, choose **Anisotropic**.

Fixed Constraint 1

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

2 Select Edges 2, 3, 8, and 15 only.

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 2 (comp2)** > **Mesh 2** click **Free Triangular 1**.

2 Select Boundaries 7 and 8 only.

3 In the **Settings** window for **Free Triangular**, click **Build Selected**.

Copy Face 1

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 Select Boundaries 7 and 8 only.

3 In the **Settings** window for **Copy Face**, locate the **Destination Boundaries** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 2 and 4 only.

6 Click **Build Selected**.

Copy Face 2

1 Right-click **Copy Face 1** and choose **Duplicate**.

2 In the **Settings** window for **Copy Face**, locate the **Source Boundaries** section.

3 Click to select the **Activate Selection** toggle button.

4 Select Boundaries 2, 4, 7, and 8 only.

5 Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.

6 Click **Clear Selection**.

7 Select Boundaries 1, 3, 5, and 6 only.

8 Click **Build All**.

ADD STUDY

1 In the **Study** 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 **Select Study** tree, select **General Studies** > **Eigenfrequency**.

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

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

EIGENFREQUENCY STUDY: SHELL

In the **Settings** window for **Study**, type `Eigenfrequency Study: Shell` in the **Label** text field.

Material Sweep

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

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

3 Click **Add**.

Step 1: Eigenfrequency

1 In the **Model Builder** window, click **Step 1: Eigenfrequency**.

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

3 In the **Solve for** column of the table, clear the checkbox for **Component 1 (comp1)**.

4 In the **Solve for** column of the table, under **Component 2 (comp2)**, clear the checkbox for **Layered Shell (lshell)**.

EIGENFREQUENCY STUDY: LAYERED SHELL

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

2 In the **Settings** window for **Study**, type `Eigenfrequency Study: Layered Shell` in the **Label** text field.

Material Sweep

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

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

3 Click **Add**.

Step 1: Eigenfrequency

1 In the **Model Builder** window, click **Step 1: Eigenfrequency**.

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

3 In the **Solve for** column of the table, clear the checkbox for **Component 1 (comp1)**.

4 In the **Solve for** column of the table, under **Component 2 (comp2)**, clear the checkbox for **Shell (shell)**.

EIGENFREQUENCY STUDY: SHELL

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

EIGENFREQUENCY STUDY: LAYERED SHELL

Click **Compute**.

RESULTS

Mode Shape (shell)

1 In the **Model Builder** window, under **Results** click **Mode Shape (shell)**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Material Switch 1** list, choose **Layered Material 1**.
4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
5 In the **Title** text area, type `Mode shape (shell): Displacement magnitude (mm)`.
6 Clear the **Parameter indicator** text field.
7 Locate the **Plot Settings** section. From the **View** list, choose **New view**.
8 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

Surface 2

1 In the **Model Builder** window, expand the **Mode Shape (shell)** node.
2 Right-click **Results** > **Mode Shape (shell)** > **Surface 1** and choose **Duplicate**.
3 In the **Settings** window for **Surface**, locate the **Data** section.
4 From the **Dataset** list, choose **Layered Material**.
5 From the **Material Switch 1** list, choose **Layered Material 2**.

Surface 3

1 Right-click **Surface 2** and choose **Duplicate**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Material Switch 1** list, choose **Layered Material 3**.
4 Click the **Go to XY View** button in the **Graphics** toolbar.
5 Click the **Show Grid** button in the **Graphics** toolbar.

Annotation 1

1 In the **Model Builder** window, right-click **Mode Shape (shell)** and choose **Annotation**.
2 In the **Settings** window for **Annotation**, locate the **Annotation** section.
3 In the **Text** text field, type `$\omega$ =eval(shell.freq, kHz, 4) kHz`.

4. Select the **LaTeX markup** checkbox.
5. From the **Geometry level** list, choose **Global**.
6. Locate the **Position** section. In the **y** text field, type `-rd`.
7. Locate the **Coloring and Style** section. Clear the **Show point** checkbox.
8. From the **Anchor point** list, choose **Lower middle**.
9. Select the **Show frame** checkbox.
10. Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Annotation 2

1. Right-click **Annotation 1** and choose **Duplicate**.
2. In the **Settings** window for **Annotation**, locate the **Data** section.
3. From the **Dataset** list, choose **Layered Material**.
4. From the **Material Switch 1** list, choose **Layered Material 2**.
5. Locate the **Plot Array** section. In the **Index** text field, type `1`.

Annotation 3

1. Right-click **Annotation 2** and choose **Duplicate**.
2. In the **Settings** window for **Annotation**, locate the **Data** section.
3. From the **Material Switch 1** list, choose **Layered Material 3**.
4. Locate the **Plot Array** section. In the **Index** text field, type `2`.

Mode Shape (shell)

In the **Model Builder** window, click **Mode Shape (shell)**.

Table Annotation 1

1. In the **Mode Shape (shell)** 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
`-0.05*rd`	`rd`	`0`	`Titanium`
`1.6*rd`	`rd`	`0`	`Composite Material 1`
`3.3*rd`	`rd`	`0`	`Composite Material 2`

5. Locate the **Coloring and Style** section. Clear the **Show point** checkbox.
6. From the **Anchor point** list, choose **Upper middle**.

Mode Shape (shell)

1 In the **Model Builder** window, click **Mode Shape (shell)**.

2 In the **Mode Shape (shell)** toolbar, click **Plot**.

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

Mode Shape (lshell)

1 In the **Model Builder** window, click **Mode Shape (lshell)**.

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

3 From the **Material Switch 1** list, choose **Layered Material 1**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Mode shape (layered shell): Displacement magnitude (mm)`.

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

7 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 9**.

8 Locate the **Plot Array** section. From the **Array type** list, choose **Linear**.

Surface 2

1 In the **Model Builder** window, expand the **Mode Shape (lshell)** node.

2 Right-click **Results** > **Mode Shape (lshell)** > **Surface 1** and choose **Duplicate**.

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

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

5 From the **Material Switch 1** list, choose **Layered Material 2**.

Surface 3

1 Right-click **Surface 2** and choose **Duplicate**.

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

3 From the **Material Switch 1** list, choose **Layered Material 3**.

Annotation 1

1 In the **Model Builder** window, right-click **Mode Shape (lshell)** and choose **Annotation**.

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

3 In the **Text** text field, type `$\omega$ =eval(lshell.freq, kHz, 4) kHz`.

4 Select the **LaTeX markup** checkbox.

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

6 Locate the **Position** section. In the **y** text field, type `-rd`.

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

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

9 Select the **Show frame** checkbox.

10 Locate the **Plot Array** section. Select the **Manual indexing** checkbox.

Annotation 2

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

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

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

4 From the **Material Switch 1** list, choose **Layered Material 2**.

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

Annotation 3

1 Right-click **Annotation 2** and choose **Duplicate**.

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

3 From the **Material Switch 1** list, choose **Layered Material 3**.

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

Mode Shape (lshell)

In the **Model Builder** window, click **Mode Shape (lshell)**.

Table Annotation 1

1 In the **Mode Shape (lshell)** 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
`-0.05*rd`	`rd`	`0`	`Titanium`
`1.6*rd`	`rd`	`0`	`Composite Material 1`
`3.3*rd`	`rd`	`0`	`Composite Material 2`

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

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

Mode Shape (lshell)

1 In the **Model Builder** window, click **Mode Shape (lshell)**.

2 In the **Mode Shape (lshell)** toolbar, click **Plot**.

Eigenfrequencies (Eigenfrequency Study: Shell)

1 In the **Model Builder** window, click **Eigenfrequencies (Eigenfrequency Study: Shell)**.

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

3 From the **Eigenfrequency selection** list, choose **First**.

4 In the **Eigenfrequencies (Eigenfrequency Study: Shell)** toolbar, click **Evaluate**.

Eigenfrequencies (Eigenfrequency Study: Layered Shell)

1 In the **Model Builder** window, click **Eigenfrequencies (Eigenfrequency Study: Layered Shell)**.

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

3 From the **Eigenfrequency selection** list, choose **First**.

4 In the **Eigenfrequencies (Eigenfrequency Study: Layered Shell)** toolbar, click **Evaluate**.

Mass of Diaphragm (Shell)

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

2 In the **Settings** window for **Evaluation Group**, type `Mass of Diaphragm (Shell)` in the **Label** text field.

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

4 From the **Eigenfrequency selection** list, choose **First**.

5 Click to expand the **Format** section.

Volume Integration 1

1 Right-click **Mass of Diaphragm (Shell)** and choose **Integration** > **Volume Integration**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`shell.rho`	`kg`	`Mass`

4 In the **Mass of Diaphragm (Shell)** toolbar, click **Evaluate**.

Mass of Diaphragm (Layered Shell)

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

2 In the **Settings** window for **Evaluation Group**, type `Mass of Diaphragm (Layered Shell)` in the **Label** text field.

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

4 From the **Eigenfrequency selection** list, choose **First**.

Volume Integration 1

1 Right-click **Mass of Diaphragm (Layered Shell)** and choose **Integration** > **Volume Integration**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`lshell.rho`	`kg`	`Mass`

4 In the **Mass of Diaphragm (Layered Shell)** toolbar, click **Evaluate**.

In the autogenerated cell periodicity studies, disable **Shell** and **Layered Shell**.

CELL PERIODICITY STUDY: RUC 1

Step 1: Stationary

1 In the **Model Builder** window, under **Cell Periodicity Study: RUC 1** click **Step 1: Stationary**.

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

3 In the tree, select **Component 2 (comp2)**.

4 Right-click and choose **Disable in Model**.

CELL PERIODICITY STUDY: RUC 2

1 In the **Model Builder** window, under **Cell Periodicity Study: RUC 2** click **Step 1: Stationary**.

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

3 In the tree, select **Component 2 (comp2)**.

4 Right-click and choose **Disable in Model**.