Model created in COMSOL Multiphysics 6.4

Material Characteristics of a Laminated Composite Shell

Introduction

This model serves the purpose of validation and verification of the **Linear Elastic Material, Layered** model in the Shell interface.

Analyses of laminated composite shells are commonly based on one of three different theories (Ref. 1):

1. Equivalent Single Layer (ESL) Theory (2D)
   a. Classical Laminated Plate Theory (CLPT-ESL)
   b. First-Order Shear Deformation Laminated Plate Theory (FSDT-ESL)
   c. Higher-Order Shear Deformation Laminated Plate Theory
2. Three-Dimensional Elasticity Theory (3D)
   a. Traditional Three Dimensional Elasticity Theory
   b. Layerwise Theory
3. Multiple Model Methods (3D and 2D)

In COMSOL Multiphysics, composites are analyzed based on either Layerwise 3D elasticity theory through the Layered Shell interface or FSDT-ESL theory through the Shell interface.

The First-Order Shear Deformation Theory (FSDT-ESL) is implemented in the **Linear Elastic Material, Layered** model in the Shell interface available with the Composite Materials Module. This theory treats a heterogeneous laminated composite as a statically equivalent single layer. ESL theory reduces a 3D continuum problem to an equivalent 2D problem, thus reducing the size and computational time of the problem.

This example is a NAFEMS benchmark, described in *Benchmarks for Membrane and Bending Analysis of Laminated Shells, Part 2: Stiffness Matrix and Thermal Characteristics* (Ref. 2). Membrane and bending stiffness, flexibility matrices, midplane strains in case of unit loading, and the response to a unit change in temperature and unit temperature gradient are verified..

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

Model Definition

The geometry of the problem consists of eight square layers stacked above each other. The side length is 1 cm and each layer has a thickness of 0.1 mm, as shown in Figure 1. The laminate has $[0/60/-60/0]_s$ stacking sequence as shown in Figure 2.

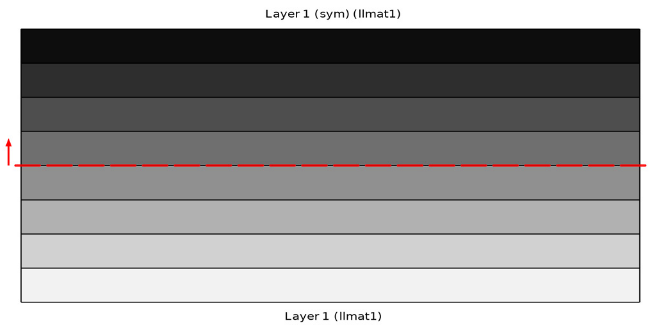

Figure 1: Cross-section view of laminated composite shell showing thickness (0.1mm) of each ply.

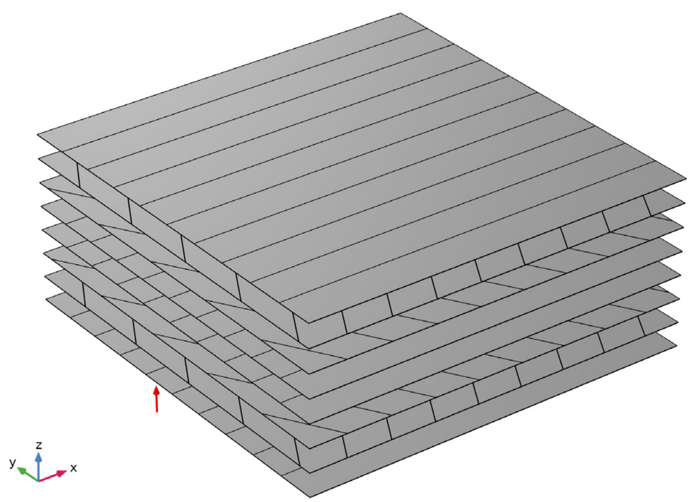

Figure 2: Stacking sequence $[0/60/-60/0]_s$ from bottom to top, showing fiber orientation in each ply.

MATERIAL PROPERTIES

The transversely isotropic material properties (Young's modulus, shear modulus, Poisson's ratio, and thermal expansion coefficients) are given in Table 1.

TABLE 1: MATERIAL PROPERTIES.

Material property	Value
$\{E_1,E_2\}$	{213,8.2} GPa
G_{12}	3.2 GPa
$\{\upsilon_{12},\upsilon_{23}\}$	{0.3,0}
$\{\alpha_1,\alpha_2,\alpha_3\}$	{1.3e-6,27e-6,27e-6} 1/K

All material properties are given in the layer coordinate system (local material directions of a layer), where the first axis is aligned with the fiber orientation.

BOUNDARY CONDITIONS

The constraints and loads applied on each node for unit loading and thermal loading are given in the tables below.

TABLE 2: NODE NUMBERS AND NODAL CONSTRAINTS.

Unit load and thermal cases **Node**	N_1	N_2	N_{12}	M_1	M_2	M_{12}	$\Delta T/ T'$
1	u, v, w, θ_x, θ_y, θ_z	u, v, w, θ_x, θ_y, θ_z	u, v, w, θ_x, θ_y, θ_z	u, v, w, θ_x, θ_y, θ_z	u, v, w, θ_x, θ_y, θ_z	u, v, w, θ_y, θ_z	u, v, w, θ_z
2	u, θ_z	θ_z	u, v, θ_z	u, θ_z	θ_z	w, θ_z	u, w, θ_z
3	θ_z	v, θ_z	u, θ_z	θ_z	v, θ_z	θ_z	w, θ_z
4	θ_z	θ_z	θ_z	θ_z	θ_z	v, θ_z	θ_z

TABLE 3: NODE NUMBERS AND NODAL POINT LOADS, MOMENTS.

Unit load cases **Node**	N_1	N_2	N_{12}	M_1	M_2	M_{12}
1	0	0	0	M_y=-5e-3	M_x=5e-3	0
2	0	F_y=5e-3	F_x=5e-3	M_y=-5e-3	M_x=-5e-3	0
3	F_x=5e-3	0	F_y=5e-3	M_y=5e-3	M_x=5e-3	F_z=2
4	F_x=5e-3	F_y=5e-3	F_x=5e-3 F_y=5e-3	M_y=5e-3	M_x=-5e-3	F_z=-2

Note that the way the benchmark is specified, it is assumed that a single first-order four-node element is used. This is the only case when such a specification can give a homogeneous strain state.

The node numbers specified in the benchmark are equal to the point numbers in the COMSOL Multiphysics geometry. The signs of the point loads and moments are adjusted to give positive unit loads and moments as specified in Ref. 2. The rotation around the z-axis, θ_z, is automatically constrained so it does not need to be considered. The unit change in temperature and unit temperature gradient is prescribed using a **Thermal Expansion** subnode of the **Linear Elastic Material, Layered**.

STUDY SETUP

The six different unit load cases and two thermal loading cases requires different boundary conditions and point loads as shown in Table 2 and Table 3. To solve all these cases in a single study, different load groups and constraint groups are created, and constrains and loads corresponding to a particular case are assigned to these groups according to Table 2 and Table 3. In the **Stationary** node in the study, appropriate load and constraint groups are selected for each load case.

Results and Discussion

The results presented in the benchmark (Ref. 2) are for the Classical Laminated Plate Theory (CLPT-ESL). The implementation of Linear Elastic Model, Layered model is however based on First Order Shear Deformation Theory (FSDT-ESL). The difference between both theories is that FSDT theory considers transverse shear stresses, but for the benchmark example with unit in-plane loading, the results should match between both theories because of the zero or negligible shear strains. Note that the bending strains are actually curvatures, but for consistency purpose they will be called as strains throughout this document.

The relation between in-plane forces and moments with midplane strains is represented by

$$\begin{bmatrix} N \\ M \end{bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa \end{bmatrix} \tag{1}$$

where A, B, and D are called *extensional stiffness matrix*, *bending-extensional stiffness matrix*, and *bending stiffness matrix*, respectively.

The same relation can be represented on flexibility form as

$$\begin{bmatrix} \varepsilon \\ \kappa \end{bmatrix} = \begin{bmatrix} a & b \\ b & d \end{bmatrix} \begin{bmatrix} N \\ M \end{bmatrix} \tag{2}$$

where a, b, and d are called *extensional flexibility matrix*, *bending-extensional flexibility matrix*, and *bending flexibility matrix*, respectively. The relationship between stiffness and flexibility matrices is

$$\begin{bmatrix} a & b \\ b & d \end{bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix}^{-1} \tag{3}$$

In the benchmark, the laminate flexibility matrix are computed in two ways, first directly from CLPT theory and then from midplane strains when unit loads and moments are applied. When unit in-plane forces N_{ij} and unit moments M_{ij} are applied, the midplane strains and curvature are the components of flexibility matrix. The midplane strains in the benchmark are computed numerically using commercial finite element software. Table 4 and Table 5 shows the flexibility matrix from the benchmark (Ref. 2).

TABLE 4: LAMINATE FLEXIBILITY MATRIX BASED ON CLASSICAL LAMINATE PLATE THEORY FROM BENCHMARK.

Strains	N_1	N_2	N_{12}	M_1	M_2	M_{12}
ε_1	1.131E-8	-3.697E-9	-4.42E-41	-1.59E-21	-3.8E-22	3.27E-21
ε_2	-3.69E-9	2.010E-8	1.16E-40	3.977E-22	-8.0E-21	-9.6E-22
γ_{12}	-4.42E-41	1.16E-40	5.593E-8	1.91E-21	-3.0E-21	-1.3E-20
κ_1	-1.59E-21	-3.8E-22	3.27E-21	1.807E-1	-5.85E-2	-1.42E-3
κ_2	3.977E-22	-8.0E-21	-9.6E-22	-5.85E-2	5.17E-1	-3.31E-1
κ_{12}	1.91E-21	-3.0E-21	-1.3E-20	-1.42E-3	-3.31E-1	1.470E0

TABLE 5: LAMINATE FLEXIBILITY MATRIX BASED ON MIDPLANE STRAINS FROM BENCHMARK.

Strains	N_1	N_2	N_{12}	M_1	M_2	M_{12}
ε_1	1.131E-8	-3.697E-9	-3.30E-24	0.00E0	0.00E0	0.00E0
ε_2	-3.69E-9	2.010E-8	1.44E-24	0.00E0	0.00E0	0.00E0
γ_{12}	-2.06E-25	-1.65E-24	5.593E-8	0.00E0	0.00E0	0.00E0
κ_1	0.00E0	0.00E0	0.00E0	1.807E-1	-5.85E-2	-1.42E-3
κ_2	0.00E0	0.00E0	0.00E0	-5.85E-2	5.17E-1	-3.31E-1
κ_{12}	0.00E0	0.00E0	0.00E0	-1.42E-3	-3.31E-1	1.470E0

The midplane strains for a unit change in temperature are given in the benchmark (Ref. 2).

TABLE 6: MIDPLANE STRAINS FOR UNIT CHANGE IN TEMPERATURE FROM BENCHMARK.

Strains	**Theory**	**Numerical FEM**
ε_1	1.9155E-6	1.9155E-6
ε_2	3.4566E-6	3.4566E-6
γ_{12}	0.00E0	-2.117E-22
κ_1	0.00E0	0.00E0
κ_2	0.00E0	0.00E0
κ_{12}	0.00E0	0.00E0

The midplane strains for a unit temperature gradient are given in the benchmark (Ref. 2).

TABLE 7: MIDPLANE STRAINS FOR UNIT TEMPERATURE GRADIENT FROM BENCHMARK.

Strains	**Theory**	**Numerical FEM**
ε_1	0.00E0	0.00E0
ε_2	0.00E0	0.00E0
γ_{12}	0.00E0	0.00E0
κ_1	1.7218E-6	1.7218E-6
κ_2	4.8753E-6	4.8753E-6
κ_{12}	-3.1156E-6	-3.1156E-6

In COMSOL Multiphysics, the stiffness (A, B, D) and flexibility matrices (a, b, d) are available as postprocessing variables. The midplane strains are computed for each unit load cases (six load cases). The strain vector then represents a column in the full flexibility matrix for a unit load case. For example, the membrane and bending midplane strains for unit N_1 forms a first column of full flexibility matrix. The full flexibility matrix based on First Order Shear Deformation Theory (FSDT-ESL) and midplane strains from COMSOL are shown in Table 8 and Table 9 below

TABLE 8: LAMINATE FLEXIBILITY MATRIX BASED ON FSDT THEORY FROM COMSOL.

Strains	N_1	N_2	N_{12}	M_1	M_2	M_{12}
ε_1	1.131E-8	-3.697E-9	2.27E-25	-4.79E-22	-2.3E-22	-1.0E-21
ε_2	-3.69E-9	2.010E-8	7.80E-24	-5.75E-23	1.40E-20	4.47E-20
γ_{12}	2.27E-25	7.80E-24	5.593E-8	-1.27E-21	3.94E-20	1.80E-20
κ_1	-4.79E-22	-2.3E-22	-1.0E-21	1.807E-1	-5.85E-2	-1.42E-3

TABLE 8: LAMINATE FLEXIBILITY MATRIX BASED ON FSDT THEORY FROM COMSOL.

Strains	N_1	N_2	N_{12}	M_1	M_2	M_{12}
κ_2	-5.75E-23	1.40E-20	4.47E-20	-5.85E-2	5.17E-1	-3.31E-1
κ_{12}	-1.27E-21	3.94E-20	1.80E-20	-1.42E-3	-3.31E-1	1.470E0

TABLE 9: LAMINATE FLEXIBILITY MATRIX BASED ON MIDPLANE STRAINS FROM COMSOL.

Strains	N_1	N_2	N_{12}	M_1	M_2	M_2
ε_1	1.131E-8	-3.697E-9	0.00E0	0.00E0	0.00E0	0.00E0
ε_2	-3.69E-9	2.010E-8	1.55E-23	0.00E0	0.00E0	0.00E0
γ_{12}	-6.41E-24	1.42E-23	5.593E-8	0.00E0	0.00E0	0.00E0
κ_1	0.00E0	0.00E0	0.00E0	1.807E-1	-5.85E-2	-1.42E-3
κ_2	0.00E0	0.00E0	0.00E0	-5.85E-2	5.17E-1	-3.31E-1
κ_{12}	0.00E0	0.00E0	0.00E0	-1.42E-3	-3.31E-1	1.470E0

The midplane strains for unit change in temperature and unit temperature gradient from COMSOL are

TABLE 10: MIDPLANE STRAINS FOR UNIT CHANGE IN TEMPERATURE FROM COMSOL.

Strains	COMSOL
ε_1	1.9155E-6
ε_2	3.4566E-6
γ_{12}	-1.645E-21
κ_1	-2.371E-19
κ_2	-1.2757E-17
κ_{12}	8.380E-18

TABLE 11: MIDPLANE STRAINS FOR UNIT TEMPERATURE GRADIENT FROM COMSOL.

Strains	COMSOL
ε_1	6.21611E-26
ε_2	-4.60298E-25
γ_{12}	-4.79904E-26
κ_1	1.72175E-6
κ_2	4.87532E-6
κ_{12}	-3.11557E-6

When comparing Table 4 to Table 8, Table 5 to Table 9, Table 6 to Table 10, and Table 7 to Table 11, an exact match can be found between the results of benchmark model and the results computed in COMSOL Multiphysics. The full flexibility matrix is found by inverting the full stiffness matrix, so the verification of the full flexibility matrix automatically verifies also the correctness of full stiffness matrix (not presented here but shown in the model).

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 **Layered Material Link** and **Layered Material Stack** have an option to transform given **Layered Material** into a symmetric or antisymmetric laminate. A repeated laminate can also be constructed using a transform option.
- 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.
- From a constitutive model point of view, you can either use the *Layerwise (LW)* theory based Layered Shell interface, or the *Equivalent Single Layer (ESL)* theory based **Linear Elastic Material, Layered** node in the Shell interface. The laminated composite presented in the current model is modeled using a **Linear Elastic Material, Layered** node in the Shell interface.

References

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

2. P. Hopkins, *Benchmarks for Membrane and Bending Analysis of Laminated Shells, Part 1: Stiffness Matrix and Thermal Characteristics,* NAFEMS, 2005.

Application Library path: `Composite_Materials_Module/Verification_Examples/laminated_shell_material_characteristics`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

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

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Load the material properties from a file.

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `laminated_shell_material_characteristics_parameters.txt`.

In order to apply unit in-plane forces and moments, as well as unit temperature difference and unit temperature gradient, create separate load and constraint groups.

Load Group for Unit N1

1. In the **Model Builder** window, right-click **Global Definitions** and choose **Load and Constraint Groups** > **Load Group**.
2. In the **Settings** window for **Load Group**, type `Load Group for Unit N1` in the **Label** text field.
3. Repeat this five times to get six load groups. Rename them as shown in the table below.

Name	**Label**
Load Group 2	`Load Group for Unit N2`
Load Group 3	`Load Group for Unit N12`
Load Group 4	`Load Group for Unit M1`
Load Group 5	`Load Group for Unit M2`
Load Group 6	`Load Group for Unit M12`

4. In the **Model Builder** window, under **Global Definitions** click **Load and Constraint Groups**.
5. Select all load groups and right click on **Group** to create a group.

Load Groups for Unit Loading

1. In the **Model Builder** window, under **Global Definitions** > **Load and Constraint Groups** click **Group 1**.
2. In the **Settings** window for **Group**, type `Load Groups for Unit Loading` in the **Label** text field.

Load Group for Unit Temperature Difference

1. In the **Model Builder** window, right-click **Load and Constraint Groups** and choose **Load Group**.
2. In the **Settings** window for **Load Group**, type `Load Group for Unit Temperature Difference` in the **Label** text field.

Load Group for Unit Temperature Gradient

1. Right-click **Load and Constraint Groups** and choose **Load Group**.
2. In the **Settings** window for **Load Group**, type `Load Group for Unit Temperature Gradient` in the **Label** text field.
3. Select thermal load groups and right click on **Group** to create a group.

Load Groups for Thermal Loading

1 In the **Model Builder** window, under **Global Definitions** > **Load and Constraint Groups** click **Group 2**.

2 In the **Settings** window for **Group**, type `Load Groups for Thermal Loading` in the **Label** text field.

Constraint Group for Unit N1, N2, N12, M1 and M2

1 In the **Model Builder** window, right-click **Load and Constraint Groups** and choose **Constraint Group**.

2 In the **Settings** window for **Constraint Group**, type `Constraint Group for Unit N1, N2, N12, M1 and M2` in the **Label** text field.

3 Repeat this four times to get required numbers of constraint groups. Rename them as shown in the table below.

Name	Label
Constraint Group 2	`Constraint Group for Unit N1 and M1`
Constraint Group 3	`Constraint Group for Unit N2 and M2`
Constraint Group 4	`Constraint Group for Unit N12`
Constraint Group 5	`Constraint Group for Unit M12`

Constraint Group for Unit N1 and M1

1 In the **Model Builder** window, click **Constraint Group for Unit N1 and M1**.

2 Select all constraint groups and right click on **Group** to create a group.

Constraint Groups for Unit Loading

1 In the **Model Builder** window, under **Global Definitions** > **Load and Constraint Groups** click **Group 3**.

2 In the **Settings** window for **Group**, type `Constraint Groups for Unit Loading` in the **Label** text field.

Constraint Group for Thermal Loading

1 In the **Model Builder** window, right-click **Load and Constraint Groups** and choose **Constraint Group**.

2 In the **Settings** window for **Constraint Group**, type `Constraint Group for Thermal Loading` in the **Label** text field.

Material 1 (mat1)

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

Now add a **Layered Material** node and assign appropriate thickness and rotation angles to each layer. The laminate is symmetric. It is sufficient to define only half the laminate in the **Layered Material** node. The transformation into a full laminate is performed through the layered material settings in the **Layered Material Link** node.

Layered Material: [0/60/-60/0]_s

1. Right-click **Materials** 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`	Material 1 (mat1)	0	0 rad	`th`	1

4. Click **Add** three times.
5. In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
`Layer 2`	Material 1 (mat1)	60	1.0472 rad	`th`	1
`Layer 3`	Material 1 (mat1)	-60	-1.0472 rad	`th`	1
`Layer 4`	Material 1 (mat1)	0	0 rad	`th`	1

6. In the **Label** text field, type `Layered Material: [0/60/-60/0]_s`.

GEOMETRY 1

Work Plane 1 (wp1)

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

Work Plane 1 (wp1) > Plane Geometry

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

Work Plane 1 (wp1) > Square 1 (sq1)

1. In the **Work Plane** toolbar, click **Square**.
2. In the **Settings** window for **Square**, locate the **Size** section.
3. In the **Side length** text field, type `1e-2`.

4 Click Build Selected.

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

MATERIALS

Layered Material Link 1 (llmat1)

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

The half laminate defined in the **Layered Material** node can be transformed into a full symmetric laminate using a transform option in the layered material settings.

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

3 From the **Transform** list, choose **Symmetric**.

4 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type `0.5`.

5 Locate the **Layered Material Settings** section. Click **Layer Cross-Section Preview** in the upper-right corner of the section.

6 Click **Layer Stack Preview** in the upper-right corner of the **Layered Material Settings** section.

The geometry is in an XY-plane in which the fibers are oriented with respect to the X direction. Hence set the first axis of the laminate coordinate system in the X direction.

DEFINITIONS (COMP1)

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**.

SHELL (SHELL)

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

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

3 Click **OK**.

Set the discretization for the displacement field to **Linear** in order to resemble the benchmark example.

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

5 In the **Settings** window for **Shell**, click to expand the **Discretization** section.

6 From the **Displacement field** list, choose **Linear**.

Linear Elastic Material, Layered 1

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

In order to study two thermal load cases separately, add two **Thermal Expansion** subfeatures and activate them in different load groups.

2 Select Boundary 1 only.

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

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

5 Select the **Transversely isotropic** checkbox.

Thermal Expansion for Unit Temperature Difference

1 In the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

2 In the **Settings** window for **Thermal Expansion**, type `Thermal Expansion for Unit Temperature Difference` in the **Label** text field.

3 Locate the **Model Input** section. From the T_{in} list, choose **User defined**. In the associated text field, type `293.15[K] +1[K]`.

4 In the **Physics** toolbar, click **Load Group** and choose **Load Groups for Thermal Loading**.

Linear Elastic Material, Layered 1

In the **Model Builder** window, click **Linear Elastic Material, Layered 1**.

Thermal Expansion for Unit Temperature Gradient

1 In the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

2 In the **Settings** window for **Thermal Expansion**, type `Thermal Expansion for Unit Temperature Gradient` in the **Label** text field.

3 Locate the **Thermal Bending** section. From the list, choose **Temperature gradient in thickness direction**.

4 In the T' text field, type `1[K/m]`.

5 In the **Physics** toolbar, click **Load Group** and choose **Load Group for Unit Temperature Gradient**.

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions** > **Materials** click **Material 1 (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}	`{E1, E2}`	Pa	Transversely isotropic
Poisson's ratio	{nuvector1, nuvector2}	`{nu12, nu23}`	1	Transversely isotropic
Shear modulus	{Gvector1}	`{G}`	N/m²	Transversely isotropic
Density	rho	`1`	kg/m³	Basic
Coefficient of thermal expansion	{alpha11, alpha22, alpha33} ; alphaij = 0	`{alpha1, alpha2, alpha3}`	1/K	Basic

Apply different constraints and point loads for different load cases as shown in Table 2 and Table 3. Attach appropriate load and constraint groups to these constraints and load features.

SHELL (SHELL)

Fixed Constraint for Unit N1, N2, N12, M1 and M2

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

2 In the **Settings** window for **Fixed Constraint**, type `Fixed Constraint for Unit N1, N2, N12, M1 and M2` in the **Label** text field.

3 Select Point 1 only.

4 In the **Physics** toolbar, click **Constraint Group** and choose **Constraint Groups for Unit Loading**.

Prescribed Displacement/Rotation for Unit N1 and M1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 In the **Settings** window for **Prescribed Displacement/Rotation**, type `Prescribed Displacement/Rotation for Unit N1 and M1` in the **Label** text field.

3 Select Point 2 only.

4 Locate the **Prescribed Displacement** section. From the **Displacement in x direction** list, choose **Prescribed**.

5 In the **Physics** toolbar, click **Constraint Group** and choose **Constraint Group for Unit N1 and M1**.

6 Duplicate the **Prescribed Displacement/Rotation 1** node nine times to get required numbers of nodal constraints. Label, selection, constraints, constraint groups should be as shown in table below.

Name	Label	Selection	DOFs to be constrained	Constraint group
Prescribed Displacement/ Rotation 2	`Prescribed Displacement/ Rotation for Unit N2 and M2`	3	`v`	`Constraint Group for N2 and M2`
Prescribed Displacement/ Rotation 3	`Prescribed Displacement/ Rotation for Unit N12 (1)`	2	`u, v`	`Constraint Group for N12`
Prescribed Displacement/ Rotation 4	`Prescribed Displacement/ Rotation for Unit N12 (2)`	3	`u`	`Constraint Group for N12`
Prescribed Displacement/ Rotation 5	`Prescribed Displacement/ Rotation for Unit M12 (1)`	1	`u, v, w, theta_y`	`Constraint Group for M12`
Prescribed Displacement/ Rotation 6	`Prescribed Displacement/ Rotation for Unit M12 (2)`	4	`v`	`Constraint Group for M12`
Prescribed Displacement/ Rotation 7	`Prescribed Displacement/ Rotation for Unit M12 (3)`	2	`w`	`Constraint Group for M12`

Name	Label	Selection	DOFs to be constrained	Constraint group
Prescribed Displacement/ Rotation 8	`Prescribed Displacement/ Rotation for Unit Change in Temperature (1)`	1	`u, v, w`	`Constraint Group for Thermal Loading`
Prescribed Displacement/ Rotation 9	`Prescribed Displacement/ Rotation for Unit Change in Temperature (2)`	3	`w`	`Constraint Group for Thermal Loading`
Prescribed Displacement/ Rotation 10	`Prescribed Displacement/ Rotation for Unit Change in Temperature (3)`	2	`u, w`	`Constraint Group for Thermal Loading`

Select all constraint features and right click on **Group** to create a group.

Constraints

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Shell (shell)** click **Group 4**.

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

Point Load for Unit N1

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

2 In the **Settings** window for **Point Load**, type `Point Load for Unit N1` in the **Label** text field.

3 Select Points 3 and 4 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{P}$ vector as

`Fi`	x
`0`	y
`0`	z

5 In the **Physics** toolbar, click **Load Group** and choose **Load Groups for Unit Loading**.

6 Duplicate the **Point Load 1** node nine times to get required numbers of nodal loads. Label, selection, loads, load groups should be as shown in the table below.

Name	**Label**	**Selection**	**Loads/Moments**	**Load group**
Point Load 2	`Point Load for Unit N2`	2, 4	`F_y = Fi`	`Load Group for N2`
Point Load 3	`Point Load for Unit N12 (1)`	2, 4	`F_x = Fi`	`Load Group for N12`
Point Load 4	`Point Load for Unit N12 (2)`	3, 4	`F_y = Fi`	`Load Group for N12`
Point Load 5	`Point Load for Unit M1 (1)`	1, 2	`M_y = -M`	`Load Group for M1`
Point Load 6	`Point Load for Unit M1 (2)`	3, 4	`M_y = M`	`Load Group for M1`
Point Load 7	`Point Load for Unit M2 (1)`	1, 3	`M_x = M`	`Load Group for M2`
Point Load 8	`Point Load for Unit M2 (2)`	2, 4	`M_x = -M`	`Load Group for unit M2`
Point Load 9	`Point Load for Unit M12 (1)`	3	`F_z = Fo`	`Load Group for Unit M12`
Point Load 10	`Point Load for Unit M12 (2)`	4	`F_z = -Fo`	`Load Group for Unit M12`

Select all load features and right click on **Group** to create a group.

Loads

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Shell (shell)** click **Group 5**.

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

MESH 1

Use a single quadrilateral element.

Mapped 1

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

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

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

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Edge Selection** section.

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

4 Locate the **Distribution** section. In the **Number of elements** text field, type 1.

5 Click **Build All**.

STUDY 1

Disable the default plots for this study.

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

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

3 Clear the **Generate default plots** checkbox.

Create eight different load cases each for six unit loads/moments and two for thermal loading. Assign appropriate load groups and constraint groups for each load cases as per Table 2 and Table 3.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Define load cases** checkbox.

4 Click **Add** seven times.

5 In the table, select the load cases and select appropriate load and constraint groups.

Load case	**Active load groups and constraint groups**
Load case for unit N1	lg1 cg1 cg2
Load case for unit N2	lg2 cg1 cg3
Load case for unit N12	lg3 cg1 cg4
Load case for unit M1	lg4 cg1 cg2
Load case for unit M2	lg5 cg1 cg3
Load case for unit M12	lg6 cg5
Load case for unit change in temperature	lg7 cg6
Load case for unit temperature gradient	lg8 cg6

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

In order to find the laminate flexibility and stiffness matrices, use **Point Matrix Evaluation** node under **Evaluation Group**.

RESULTS

Extensional Flexibility Matrix

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

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

3 From the **Parameter selection (Load case)** list, choose **First**.

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

5 In the **Label** text field, type `Extensional Flexibility Matrix`.

Point Matrix Evaluation 1

1 In the **Extensional Flexibility Matrix** toolbar, click **More Evaluations** and choose **Point Matrix Evaluation**.

2 Select Point 3 only.

3 In the **Settings** window for **Point Matrix Evaluation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Shell** > **Effective material properties** > **Extensional flexibility matrix** > **shell.Da - Extensional flexibility matrix - s²/kg**.

Extensional Flexibility Matrix

1 In the **Model Builder** window, click **Extensional Flexibility Matrix**.

2 In the **Extensional Flexibility Matrix** toolbar, click **Evaluate**.

EXTENSIONAL FLEXIBILITY MATRIX

1 Go to the **Extensional Flexibility Matrix** window.

2 To compute the remaining flexibility and stiffness matrices, duplicate the above node seven times and change the expression to compute `shell.Db`, `shell.Dd`, `shell.Das`, `shell.DA`, `shell.DB`, `shell.DD` and `shell.DAs`.

Select evaluation groups corresponding to the flexibility and stiffness matrices, and right click on **Group** to create a group.

RESULTS

Flexibility and Stiffness Matrices

1 In the **Model Builder** window, under **Results** click **Group 6**.

2 In the **Settings** window for **Group**, type `Flexibility and Stiffness Matrices` in the **Label** text field.

Use a **Point Evaluation** node under **Evaluation Group** to compute the midplane strains for each load case. The choice of point does not matter as the model gives a uniform strain field.

While evaluating the midplane strains corresponding to each load case, evaluate the corresponding in-plane force or moment postprocessing variable in order to verify the correctness of the applied forces and constraints.

Layered Material 1

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

Cut Point 3D 1

1 In the **Results** toolbar, click **Cut Point 3D**.
2 In the **Settings** window for **Cut Point 3D**, locate the **Data** section.
3 From the **Dataset** list, choose **Layered Material 1**.
4 Locate the **Point Data** section. In the **x** text field, type `1e-2`.
5 In the **y** text field, type `0`.
6 In the **z** text field, type `0`.

Midplane Strains for Unit N1

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Midplane Strains for Unit N1` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 3D 1**.
4 From the **Parameter selection (Load case)** list, choose **From list**.
5 In the **Load cases** list box, select **Load case for unit N1**.
6 Locate the **Transformation** section. Select the **Transpose** checkbox.
7 Locate the **Format** section. From the **Include parameters** list, choose **Off**.

Point Evaluation 1

1 Right-click **Midplane Strains for Unit N1** and choose **Point Evaluation**.
2 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`shell.eml11`	`1`	`Membrane part of strain tensor (local), 11 component`
`shell.eml22`	`1`	`Membrane part of strain tensor (local), 22 component`
`2*shell.eml12`	`1`	`Membrane part of strain tensor (local), 12 component`
`shell.ebl11`	`1/m`	`Bending part of strain tensor (local), 11 component`
`shell.ebl22`	`1/m`	`Bending part of strain tensor (local), 22 component`
`2*shell.ebl12`	`1/m`	`Bending part of strain tensor (local), 12 component`
`shell.Nl11`	`N/m`	`Local in-plane force, 11 component`

Midplane Strains for Unit N1

1. In the **Model Builder** window, click **Midplane Strains for Unit N1**.
2. In the **Midplane Strains for Unit N1** toolbar, click **Evaluate**.

MIDPLANE STRAINS FOR UNIT N1

1. Go to the **Midplane Strains for Unit N1** window.
2. To compute the midplane strains for remaining load cases, duplicate the above node seven times and change the load case in the **Data** section of corresponding **Evaluation Group** node. Replace the in-plane force/moment variable in the last row of table in **Point Evaluation** node with appropriate variable corresponding to the load cases.

Select evaluation groups corresponding to the midplane strains, and right click on **Group** to create a group.

RESULTS

Midplane Strains

1. In the **Model Builder** window, under **Results** click **Group 7**.
2. In the **Settings** window for **Group**, type `Midplane Strains` in the **Label** text field.