Micromechanical Model of a Composite

Introduction

The use of fiber reinforced composites is increasing in various industries like automotive, aerospace, infrastructure, and many more. The accuracy of structural or thermal analysis relies on an accurate estimation of mechanical and thermal properties of the composite material.

In this example, a simplified micromechanical model of a unit cell with periodic boundary conditions is analyzed. The unit cell is a representative volume element (RVE) which repeats itself to form the complete structure of the composite. In the current model, the unit cell is made of a single carbon fiber placed at center of an epoxy matrix. The fiber volume fraction is varied. The homogenized elastic and thermal properties of the composite material are computed based on the individual properties of fiber and matrix. A comparison is made against values obtained from the Rule of Mixture (ROM).

Model Definition

The composite is assumed to be made of carbon fibers unidirectionally embedded in epoxy resin. A representative unit cell having a cylindrical fiber located at the center of resin is shown in Figure 1. The fiber radius is computed based on a parameterized fiber volume fraction.

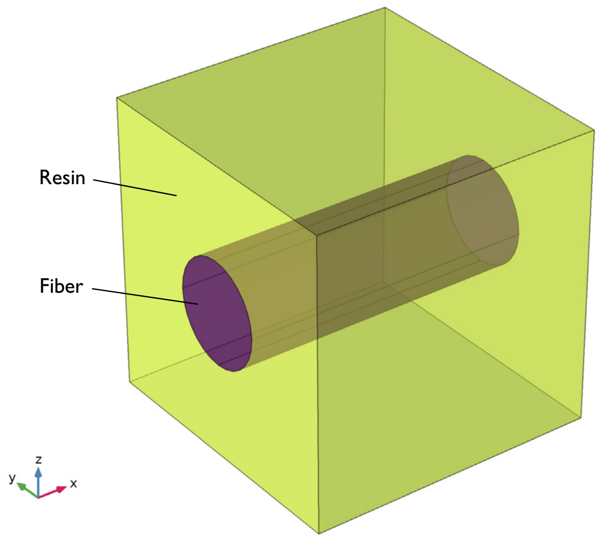

Figure 1: Geometry of the unit cell with a carbon fiber in an epoxy resin.

Fiber and Matrix Properties

The layers of the laminate are made of T300 carbon fiber and 914C epoxy. The carbon fiber is assumed to be transversely isotropic (modeled as orthotropic), and the epoxy resin is assumed to be isotropic. The material properties of fiber and resin are given in Table 1 and Table 2 respectively.

TABLE 1: CARBON FIBER MATERIAL PROPERTIES.

Material Property	**Value**
$\{E_{1f}, E_{2f}, E_{3f}\}$	{230,15,15} (GPa)
$\{G_{12f}, G_{23f}, G_{13f}\}$	{15,7,15} (GPa)
$\{\upsilon_{12f}, \upsilon_{23f}, \upsilon_{13f}\}$	{0.2,0.07,0.2}
ρ_f	1800 (kg/m^3)

TABLE 2: EPOXY RESIN MATERIAL PROPERTIES.

Material Property	**Value**
E_m	4 (GPa)
υ_m	0.35
ρ_m	1100 (kg/m^3)

For computing homogeneous elastic properties, the Poisson's ratios for the fiber and matrix materials are intentionally set to zero, in order to reduce the elements of elasticity matrix D_{11}, D_{22}, D_{33} to homogenized Young's modulus E_{11}, E_{22}, E_{33} respectively. In this way, results can be easily compared to the homogenized Young's moduli computed with the ROM.

The homogeneous thermal properties are computed with zero Poisson's ratio as well as with the ratios given in Table 1 and Table 2.

Rule of Mixture (ROM)

Based on the material properties from Table 1 and Table 2, the homogenized Young's moduli, the in-plane shear modulus, and the in-plane Poisson's ratio are computed from the ROM as (Ref. 1):

$$E_{11} = V_f E_{1f} + V_m E_m \tag{1}$$

$$E_{22} = E_{33} = \frac{E_{2f} E_m}{V_f E_m + V_m E_{2f}} \tag{2}$$

$$G_{12} = \frac{G_{12f} G_m}{V_f G_m + V_m G_{12f}} \tag{3}$$

$$\upsilon_{12} = V_f \upsilon_{12f} + V_m \upsilon_m \tag{4}$$

Here, V_f and V_m are the fiber and matrix volume fractions, respectively. The coefficients of thermal expansion for a composite in the fiber and transverse directions are calculated from the rule of mixture using:

$$\alpha_{11} = \frac{V_f \alpha_{1f} E_{1f} + V_m \alpha_m E_m}{V_f E_{1f} + V_m E_m} \tag{5}$$

$$\alpha_{22} = \alpha_{33} = (1 + \upsilon_m) V_m \alpha_m + \left(1 + \upsilon_{12f} \frac{\alpha_{1f}}{\alpha_{2f}}\right) V_f \alpha_{2f} - \upsilon_{12} \alpha_{11} \tag{6}$$

The above formulas are used to compute analytical values of the homogenized elastic and thermal properties for comparison with the RVE model.

Results and Discussion

Figure 2 and Figure 3 show the comparison between the longitudinal and transversal homogeneous Young's moduli computed from the RVE and the ROM. The longitudinal

Young's modulus matches quite closely. The transverse Young's modulus, however, differs more and more as fiber volume fraction increases. The same behavior is observed for the in-plane homogeneous shear modulus (Figure 4). This is not surprising, given that the transverse stresses are much more inhomogeneous than those along the fiber.

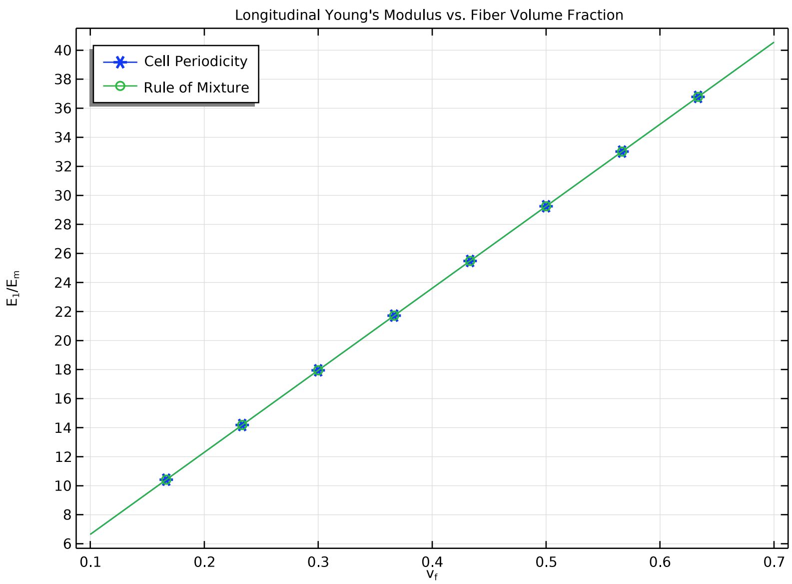

Figure 2: Longitudinal Young's modulus versus fiber volume fraction.

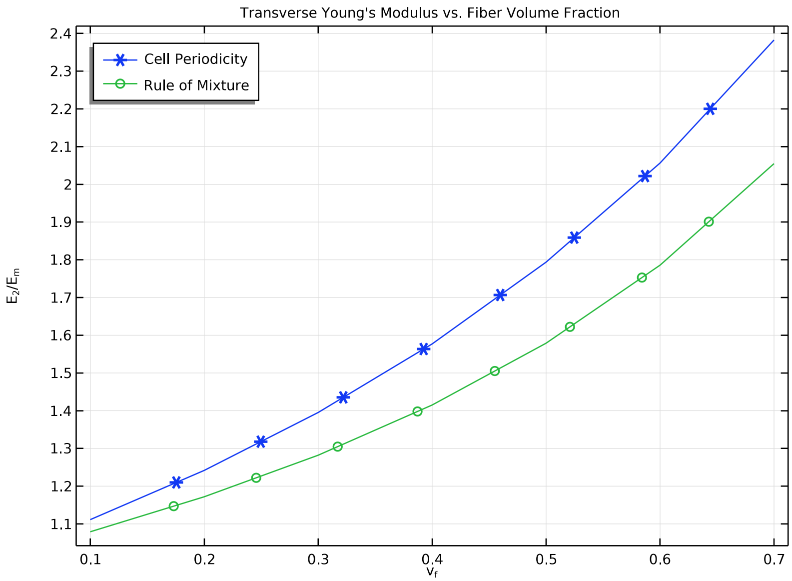

Figure 3: Transverse Young's modulus versus fiber volume fraction..

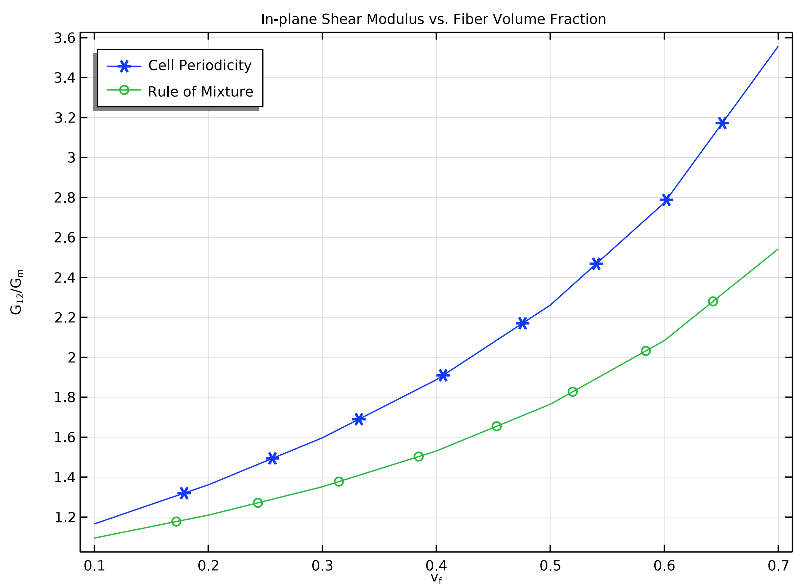

Figure 4: In-plane shear modulus versus fiber volume fraction.

The longitudinal and transverse homogeneous coefficients of thermal expansion computed using the RVE and the ROM are shown in the Figure 5 and Figure 6, respectively for zero Poisson's ratio. As expected, the longitudinal coefficient of thermal expansion matches exactly, while the numerically computed transverse coefficient of thermal expansion differs from the prediction by the ROM.

For nonzero Poisson's ratio, the homogeneous longitudinal and transverse coefficients of thermal expansion are shown in Figure 7 and Figure 8, respectively. Now, the numerically computed longitudinal coefficient of thermal expansion no longer matches the ROM values. The same behavior can also be expected for the longitudinal Young's modulus.

All figures indicate that the macromechanically computed longitudinal elastic and thermal properties match exactly with the values computed from the ROM when the constituent materials have zero Poisson's ratio. For nonzero Poisson's ratio, even longitudinal properties computed numerically and from the ROM differ.

The transverse/shear elastic and thermal properties do not match well with the values computed from the ROM. The difference increases with increasing fiber volume fraction.

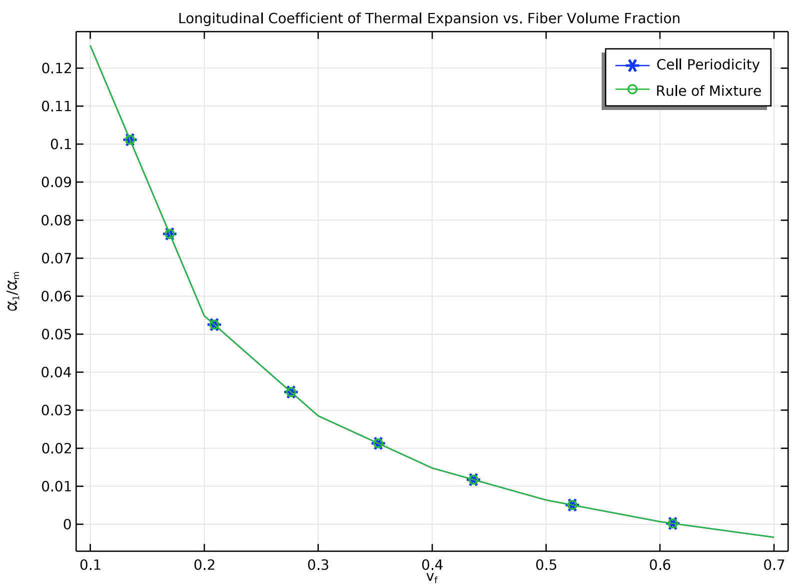

Figure 5: Longitudinal coefficient of thermal expansion versus fiber volume fraction.

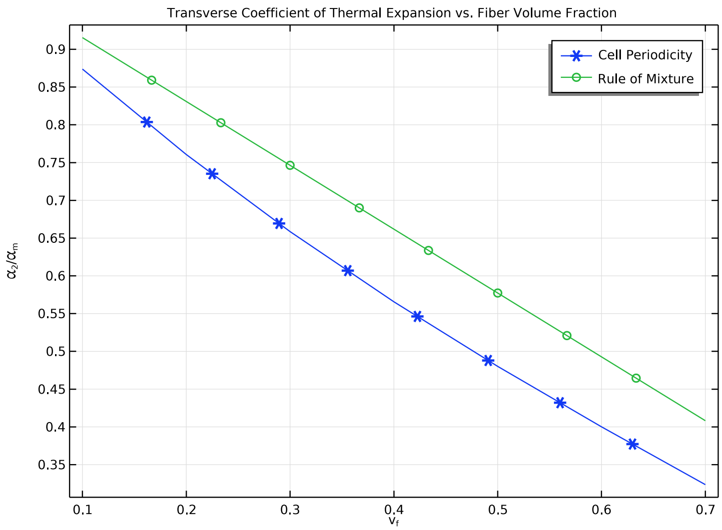

Figure 6: Transverse coefficient of thermal expansion versus fiber volume fraction.

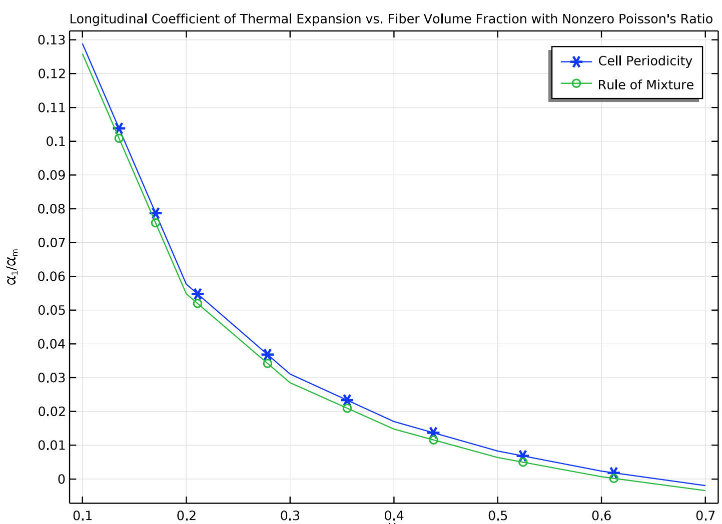

Figure 7: Longitudinal coefficient of thermal expansion versus fiber volume fraction with nonzero Poisson's ratio.

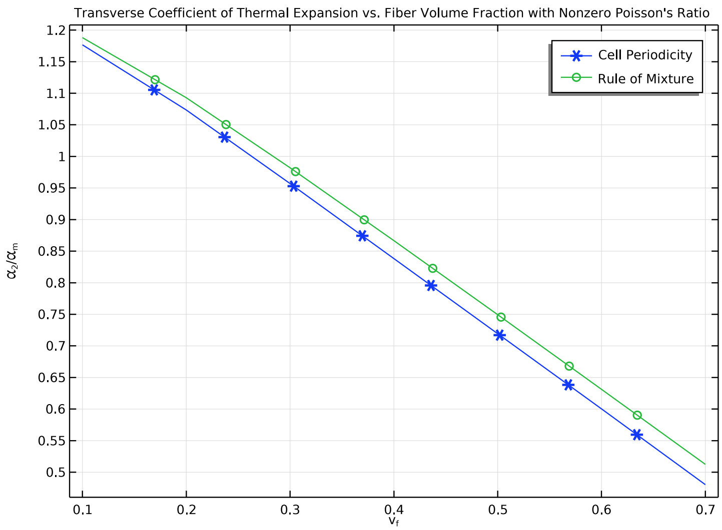

Figure 8: Transverse coefficient of thermal expansion versus fiber volume fraction with nonzero Poisson's ratio.

Notes About the COMSOL Implementation

- In order to perform a micromechanical analysis, the **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.
- In order to extract the homogenized elasticity matrix for a composite, the **Average strain** periodicity type needs to be chosen. The unit cell needs to be analyzed for six different load cases. This is automatically done through the **Cell Periodicity** node by clicking the **Create** button. This operation adds the required number of load cases, populates the average strain matrix with boolean variables, creates a global material, and creates a stationary study with preselected load groups. The created global material contains an elasticity matrix corresponding to that of the homogenized material. This material can be used to define the properties of individual layers in a composite laminate. If, by mistake, one of the automatically generated nodes is edited or deleted, you can click the **Create** button again to regenerate those nodes.
- The default computed homogenized elasticity matrix **D** is tied to the tag of the solution node of an automatically generated study. In this example, **D** is computed in a

parametric sweep. The elements of elasticity matrix must then be accessed using customized expressions as the tag of the parametric solution node is different.

- In order to extract the homogenized coefficient of thermal expansions, the **Free Expansion** option with **Coefficient of thermal expansion** is used.

Reference

1. N. Srisuk, *A Micromechanics Model of Thermal Expansion Coefficient in Fiber Reinforced Composites*, Master Thesis-The University of Texas st Arlington, 2010.

Application Library path: `Structural_Mechanics_Module/Material_Models/micromechanical_model_of_a_composite`

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 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 Click **Load from File**.

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

GEOMETRY 1

Block: Resin

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Block: Resin` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `1`.

4 In the **Depth** text field, type `1`.

5 In the **Height** text field, type `1`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Color** list, choose **Color 4**.

On Windows, click the 4th color in the first row of the palette.

Cylinder: Fiber

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, type `Cylinder: Fiber` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_f`.

4 In the **Height** text field, type `1`.

5 Locate the **Position** section. In the **y** text field, type `1/2`.

6 In the **z** text field, type `1/2`.

7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 From the **Color** list, choose **Color 1**.

On Windows, click the 1st color in the first row of the palette.

10 In the **Geometry** toolbar, click **Build All**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Thermal Expansion 1

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

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `21[degC]`.

Linear Elastic Material 2

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

2 Select Domain 2 only.

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

4 From the **Solid model** list, choose **Orthotropic**.

Thermal Expansion 1

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

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `21[degC]`.

Cell Periodicity for Elastic Properties

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

2 In the **Settings** window for **Cell Periodicity**, type `Cell Periodicity for Elastic Properties` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

4 Locate the **Periodicity Type** section. From the list, choose **Average strain**.

5 From the **Calculate average properties** list, choose **Elasticity matrix, Standard (XX, YY, ZZ, XY, YZ, XZ)**.

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 Click **Clear Selection**.

4 Select Boundaries 1, 5, 11, and 12 only.

Boundary Pair 2

1 Right-click **Boundary Pair 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundaries 2 and 10 only.

Boundary Pair 3

1 Right-click **Boundary Pair 2** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundaries 3 and 4 only.

With the **Average strain** option in the **Cell Periodicity** feature, appropriate load groups, a study and a material with computed elastic properties can be generated automatically. To create load groups and a study node, click the **Create Load Groups and Study** button in the section toolbar.

Cell Periodicity for Elastic Properties

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

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

3 Click **Create Load Groups and Study** in the upper-right corner of the section.

Cell Periodicity for Thermal Properties

1 Right-click **Cell Periodicity for Elastic Properties** and choose **Duplicate**.

2 In the **Settings** window for **Cell Periodicity**, type `Cell Periodicity for Thermal Properties` in the **Label** text field.

3 Locate the **Periodicity Type** section. From the list, choose **Free expansion**.

4 From the **Calculate average properties** list, choose **Coefficient of thermal expansion**.

MATERIALS

Material 1: Epoxy Resin

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

2 In the **Settings** window for **Material**, type `Material 1: Epoxy Resin` in the **Label** text field.

3 Select Domain 1 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	`E_m`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`nu_m`	1	Young's modulus and Poisson's ratio
Density	rho	`rho_m`	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`alpha_m`	1/K	Basic

Material 2: Carbon Fiber

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Material 2: Carbon Fiber` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. Click **Paste Selection**.
4. In the **Paste Selection** dialog box, type 2 in the **Selection** text field.
5. Click **OK**.
6. In the **Settings** window for **Material**, locate the **Material Contents** section.
7. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	{Evector1, Evector2, Evector3}	`{E1_f, E2_f, E2_f}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{nu12_f, nu23_f, nu12_f}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{G12_f, G23_f, G12_f}`	N/m²	Orthotropic

Property	Variable	Value	Unit	Property group
Density	rho	rho_f	kg/m³	Basic
Coefficient of thermal expansion	{alpha11, alpha22, alpha33} ; alphaij = 0	{alpha1_f, alpha2_f, alpha2_f}	1/K	Basic

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.
2 Select Boundaries 1 and 5 only.
3 In the **Settings** window for **Free Triangular**, click **Build Selected**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.
2 In the **Settings** window for **Swept**, click **Build Selected**.

For this study, disable the thermal expansion nodes and the cell periodicity feature for thermal expansion.

CELL PERIODICITY STUDY FOR ELASTIC PROPERTIES

1 In the **Model Builder** window, click **Cell Periodicity Study**.
2 In the **Settings** window for **Study**, type `Cell Periodicity Study for Elastic Properties` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 Click **Add**.
4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
v_f (Fiber volume fraction)	range(0.1,0.1,0.7)	

Step 1: Stationary

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

4 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)> Linear Elastic Material 1>Thermal Expansion 1**.

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

6 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)> Linear Elastic Material 2>Thermal Expansion 1**.

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

8 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)> Cell Periodicity for Thermal Properties**.

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

The default computed homogenized elasticity matrix `D` is tied to the tag of the solution node of the automatically generated study. Hence, if one needs to compute `D` matrix with a parametric sweep, the elements of the elasticity matrix need to be computed using customized expressions, as the tag of the parametric solution node is different.

Create a **Variable** node and enter the following customized expressions to compute the elements of the `D` matrix for postprocessing purpose.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
D11	withsol('sol1',solid.cp1.sigmatmp1, setval(loadcase,1),setval(v_f, root.v_f))		Elasticity matrix, 11 component
D22	withsol('sol1',solid.cp1.sigmatmp2, setval(loadcase,2),setval(v_f, root.v_f))		Elasticity matrix, 22 component
D44	withsol('sol1',solid.cp1.sigmatmp4, setval(loadcase,4),setval(v_f, root.v_f))		Elasticity matrix, 44 component

CELL PERIODICITY STUDY FOR ELASTIC PROPERTIES

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

Add a separate study to compute the homogeneous thermal properties. For this study, disable the cell periodicity feature for elastic properties.

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>Stationary**.

4 Right-click and choose **Add Study**.

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

CELL PERIODICITY STUDY FOR THERMAL PROPERTIES

In the **Settings** window for **Study**, type `Cell Periodicity Study for Thermal Properties` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

This study computes the homogenized thermal properties with zero and nonzero Poisson's ratios. Therefore, use a parametric sweep for the parameter `para` along with `v_f`.

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

3 From the **Sweep type** list, choose **All combinations**.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Nondimensional parameter)	0 1	

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
v_f (Fiber volume fraction)	range(0.1,0.1,0.7)	

Step 1: Stationary

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

4 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)> Cell Periodicity for Elastic Properties**.

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

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

When plotting the computed elasticity matrix elements in 1D plot groups, the load case in the parameter selection is irrelevant.

RESULTS

Longitudinal Young's Modulus vs. Fiber Volume Fraction

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

2 In the **Settings** window for **1D Plot Group**, type Longitudinal Young's Modulus vs. Fiber Volume Fraction in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cell Periodicity Study for Elastic Properties/Parametric Solutions 1 (sol1)**.

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

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type Longitudinal Young's Modulus vs. Fiber Volume Fraction.

7 Locate the **Plot Settings** section. Select the **x-axis label** check box.

8 In the associated text field, type v_f.

9 Select the **y-axis label** check box.

10 In the associated text field, type E₁/E_m.

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

Global 1

1 Right-click **Longitudinal Young's Modulus vs. Fiber Volume Fraction** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
D11/E_m	1	
E1/E_m	1	

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **v_f**.

5 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

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

7 In the table, enter the following settings:

Legends
Cell Periodicity
Rule of Mixture

8 Duplicate or add this plot group two times in order to plot the remaining elastic properties. The labels, titles, and the expressions to be defined in the **Global 1** node are shown in the table below.

Name	Label/Title	Expressions in global node
1D Plot Group 2	Transverse Young's Modulus vs. Fiber Volume Fraction	D22/E_m, E2/E_m
1D Plot Group 3	In-plane Shear Modulus vs. Fiber Volume Fraction	D44/G12_m, G12/G12_m

Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction

1 In the **Model Builder** window, right-click **Longitudinal Young's Modulus vs. Fiber Volume Fraction** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cell Periodicity Study for Thermal Properties/ Parametric Solutions 2 (sol10)**.

4 From the **Parameter selection (para)** list, choose **From list**.

5 In the **Parameter values (para)** list, select **0**.

6 From the **Parameter selection (v_f)** list, choose **All**.

7 In the **Label** text field, type Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction.

8 Locate the **Title** section. In the **Title** text area, type Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction.

9 Locate the **Plot Settings** section. In the **y-axis label** text field, type \alpha₁/\alpha_m.

10 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Global 1

1. In the **Model Builder** window, expand the **Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction** node, then click **Global 1**.
2. In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Cell periodicity>Coefficient of thermal expansion (material and geometry frames) - 1/K>solid.cp2.alphaXX - Coefficient of thermal expansion, XX component**.
3. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.cp2.alphaXX/alpha_m`	1	
`alpha1/alpha_m`	1	

4. Locate the **x-Axis Data** section. From the **Axis source data** list, choose **v_f**.
5. In the **Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction** toolbar, click **Plot**.
6. Duplicate or add this plot group three times in order to plot the remaining thermal properties. The labels, titles, parameter values, and expressions to be defined in the **Global 1** node are shown in the table below. The parameter value of `para` needs to be changed in the **Data** section of the corresponding 1D plot group.

Name	Label/Title	parameter value	Expressions in global node
1D Plot Group 5	`Transverse Coefficient of Thermal Expansion vs. Fiber Volume Fraction`	`para = 0`	`solid.cp2.alphaYY/alpha_m, alpha2/alpha_m`

Name	Label/Title	parameter value	Expressions in global node
1D Plot Group 6	Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction with Nonzero Poisson's Ratio	para = 1	solid.cp2.alphaXX/alpha_m, alpha1/alpha_m
1D Plot Group 7	Transverse Coefficient of Thermal Expansion vs. Fiber Volume Fraction with Nonzero Poisson's Ratio	para = 1	solid.cp2.alphaYY/alpha_m, alpha2/alpha_m

You can finally group the plots for better readability.

In-plane Shear Modulus vs. Fiber Volume Fraction, Longitudinal Young's Modulus vs. Fiber Volume Fraction, Transverse Young's Modulus vs. Fiber Volume Fraction

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Longitudinal Young's Modulus vs. Fiber Volume Fraction**, **Transverse Young's Modulus vs. Fiber Volume Fraction**, and **In-plane Shear Modulus vs. Fiber Volume Fraction**.

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

Elastic Properties

In the **Settings** window for **Group**, type `Elastic Properties` in the **Label** text field.

Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction, Transverse Coefficient of Thermal Expansion vs. Fiber Volume Fraction

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction** and **Transverse Coefficient of Thermal Expansion vs. Fiber Volume Fraction**.

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

Thermal Expansion Properties, Zero Poisson's Ratio

In the **Settings** window for **Group**, type `Thermal Expansion Properties, Zero Poisson's Ratio` in the **Label** text field.

Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction with Nonzero Poisson's Ratio, Transverse Coefficient of Thermal Expansion vs. Fiber Volume Fraction with Nonzero Poisson's Ratio

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction with Nonzero Poisson's Ratio** and **Transverse Coefficient of Thermal Expansion vs. Fiber Volume Fraction with Nonzero Poisson's Ratio**.

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

Thermal Expansion Properties, Nonzero Poisson's Ratio

In the **Settings** window for **Group**, type `Thermal Expansion Properties, Nonzero Poisson's Ratio` in the **Label** text field.