Created in COMSOL Multiphysics 6.2

Micromechanical Model of a Fiber Composite

Introduction

The use of fiber-reinforced composites is increasing in various industries like automotive, aerospace, infrastructure, and many more. The accuracy of structural and thermal analyses relies on an accurate estimation of the 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. A repeating unit cell (RUC) is a material subvolume that can be repeated in space to form the complete microstructure of the material. Here, we consider a unit cell consisting of a single carbon fiber placed at the center of an epoxy matrix. The fiber volume fraction is varied between 0.1 and 0.7. The homogenized elastic and thermal properties of the composite material are computed based on the individual properties of fiber and matrix and compared against values obtained from different analytical models based on Rule of Mixture (ROM) analyses.

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 the matrix is shown in Figure 1. The fiber radius is computed based on the fiber volume fraction chosen.

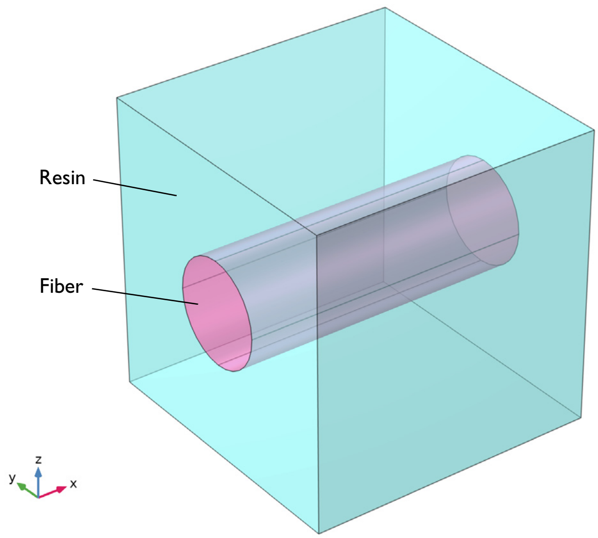

Figure 1: Geometry of the unit cell with a carbon fiber in an epoxy resin.

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
$\{\nu_{12f}, \nu_{23f}, \nu_{13f}\}$	{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
ν_m	0.35
ρ_m	1100 kg/m^3

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

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

Rule of Mixture (ROM)

For the analysis of the effective mechanical material properties, the following analytical ROM models are used in the comparison:

- Voigt–Reuss;
- Modified Voigt–Reuss;
- Chamis;
- Halpin–Tsai;
- Halpin–Tsai–Nielsen;
- Hashin–Rosen.

For analysis of the effective coefficient of thermal expansion, only the Voigt–Reuss and the Chamis models are used.

Results and Discussion

Figure 2 and Figure 3 show the comparison between the longitudinal and transversal homogenized Young's moduli computed from the RUC and different ROM models. The longitudinal Young's modulus matches quite closely. The transverse Young's modulus, however, differs more and more as the fiber volume fraction increases. For the in-plane homogenized shear modulus (Figure 4), the Modified Voigt–Reuss, Hashin–Tsai, and Hashin–Rosen models all give close results, while for small to medium volume fractions also the Hashin–Tsai–Nielsen model matches the numerical results closely.

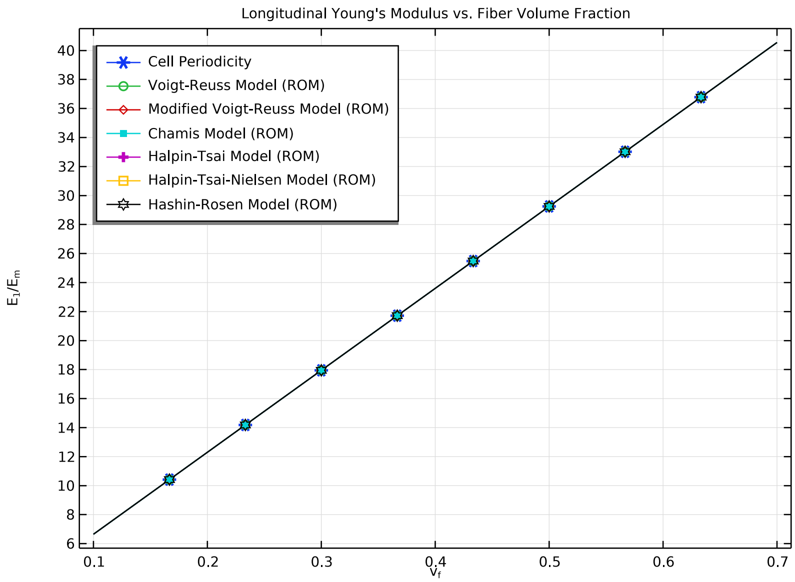

Figure 2: Longitudinal Young's modulus versus fiber volume fraction.

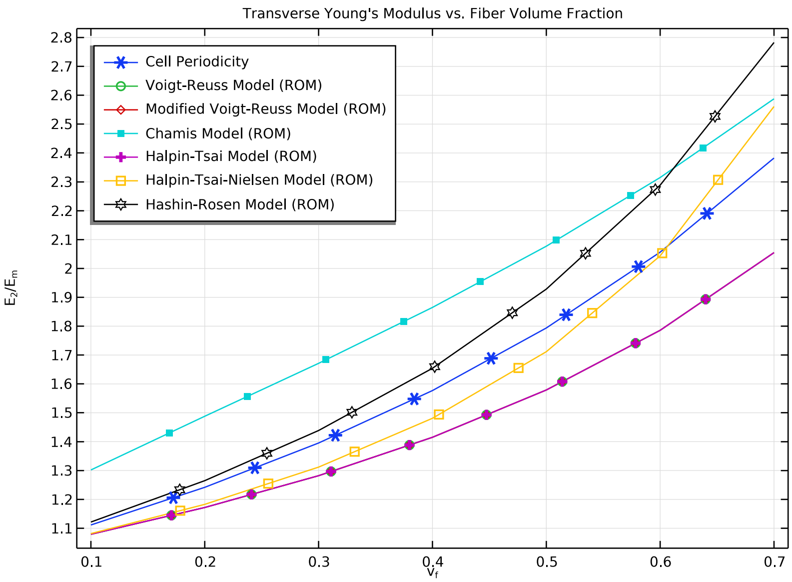

Figure 3: Transverse Young's modulus versus fiber volume fraction.

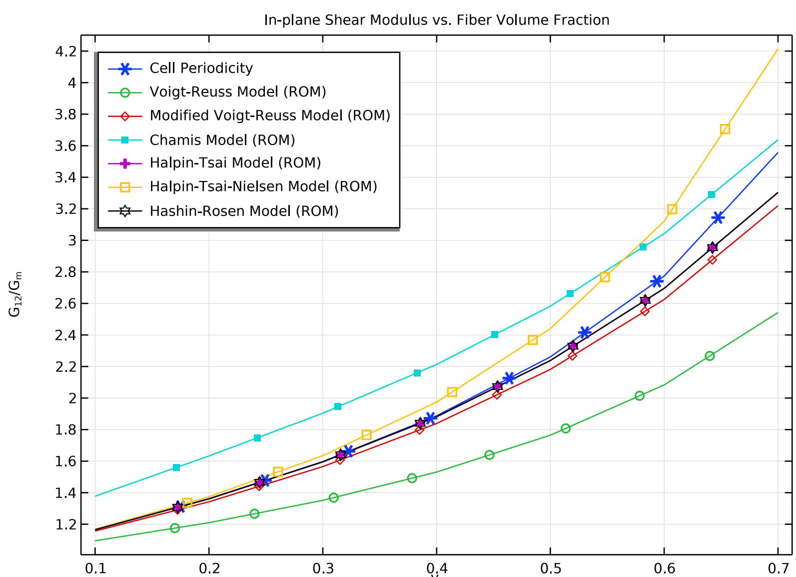

Figure 4: In-plane shear modulus versus fiber volume fraction.

The homogenized longitudinal and transverse coefficient of thermal expansion computed using the RUC and the different ROM models are shown in Figure 5 and Figure 6, respectively, for the case of 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 ROM predictions.

For nonzero Poisson's ratio, the homogenized longitudinal and transverse coefficient 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 transverse Young's modulus.

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

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

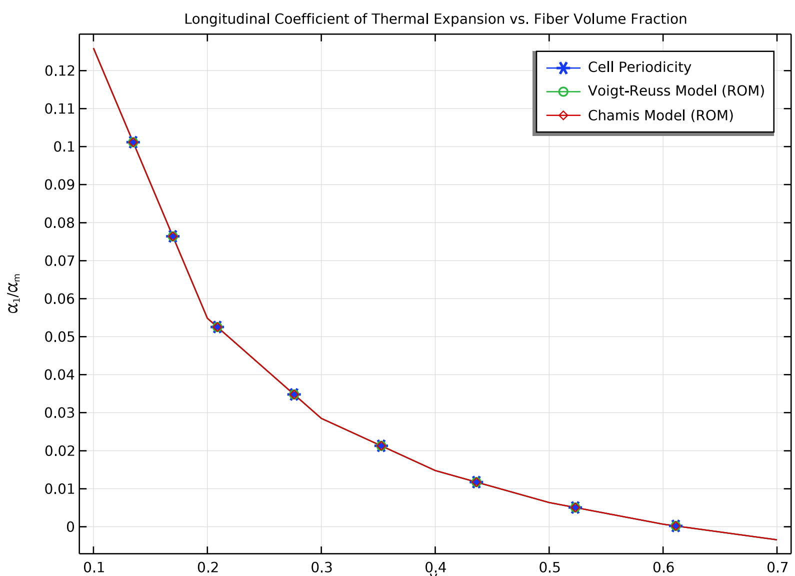

Figure 5: Longitudinal coefficient of thermal expansion versus fiber volume fraction.

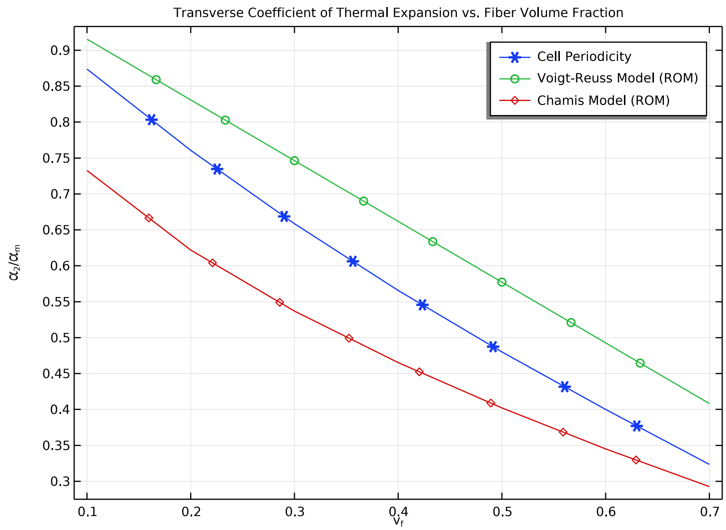

Figure 6: Transverse coefficient of thermal expansion versus fiber volume fraction.

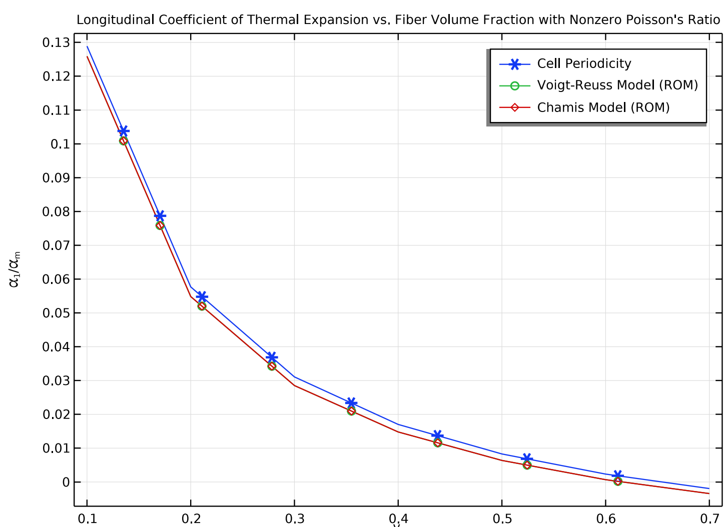

Figure 7: Longitudinal coefficient of thermal expansion versus fiber volume fraction with nonzero Poisson's ratio.

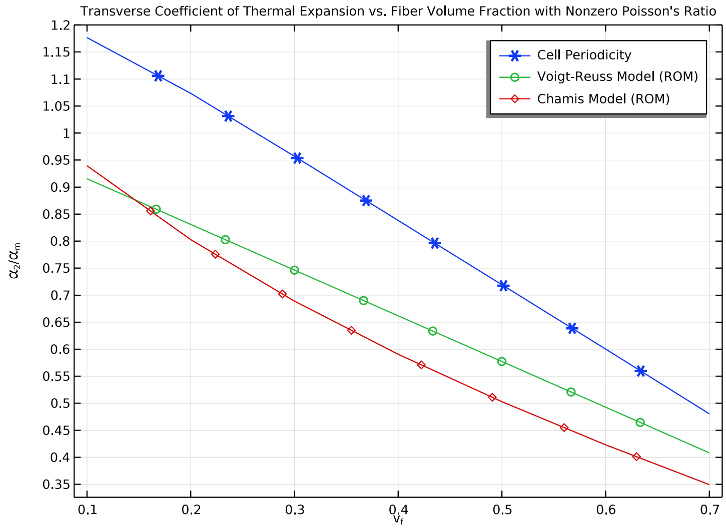

Figure 8: Transverse coefficient of thermal expansion versus fiber volume fraction with nonzero Poisson's ratio.

Notes About the COMSOL Implementation

- The **Effective Material** node can compute the effective properties of a heterogeneous material which has multiple constituents. Several composite-specific as well as general mixing rules are available depending on the material properties.
- 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 a unit cell.
- The **Cell Periodicity** node has three action buttons in the toolbar of the section called **Periodicity Type**: **Create Load Groups and Study**, **Create Material by Value**, and **Create Material by Reference**. The action button **Create Load Groups and Study** generates load groups and a stationary study with load cases. The action button **Create Material by Value** generates a **Global Material** with homogenized material properties, with material properties as numbers. The action button **Create Material by Reference** generates a **Global Material** with homogenized material properties, with material properties as variables. The action buttons are active depending on the choices in the **Periodicity Type** and **Calculate Average Properties** lists.

- The **Create Load Groups and Study** button does not generate a parametric study by default. In many situations, a parametric study is needed, and the homogenized elasticity matrix **D** needs to be based on the tag of the parametric solution. To do this use the settings in the **Advanced** section of the feature.
- 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 at Arlington, 2010.

Application Library path: `Structural_Mechanics_Module/Material_Models/micromechanical_model_of_a_fiber_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_fiber_composite_parameters.txt`.

GEOMETRY 1

Next, create a repeating unit cell (RUC) for a unidirectional fiber composite with square fiber packing. This RUC like many others can be found in the built-in **Part Libraries**.

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

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

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

4 Right-click **Component 1 (comp1)>Geometry 1** and choose **Add to Geometry**.

5 In the **Select Part Variant** dialog box, select **Specify fiber diameter** in the **Select part variant** list.

6 Click **OK**.

GEOMETRY 1

Unidirectional Fiber Composite, Square Packing 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Unidirectional Fiber Composite, Square Packing 1 (pi1)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
df	2*r_f	0.035682 m	Fiber diameter
wm	1	0.1 m	Cell width
dm	1	0.1 m	Cell depth
hm	1	0.1 m	Cell height

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

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 **Cell Properties** section. From the **Boundary conditions** list, choose **Average strain**.

4 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.

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

Boundary Pair 2

1 In the **Model Builder** window, click **Boundary Pair 2**.

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

3 Click **Clear Selection**.

4 Select Boundaries 2 and 10 only.

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

Boundary Pair 3

1 In the **Model Builder** window, click **Boundary Pair 3**.

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

To create a parametric study, use options in the feature's **Advanced** section. To see this section, activate advanced physics settings as follows.

1. Click the **Show More Options** button in the **Model Builder** toolbar.
2. In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
3. Click **OK**.
4. In the **Model Builder** window, click **Cell Periodicity for Elastic Properties**.
5. In the **Settings** window for **Cell Periodicity**, click to expand the **Advanced** section.
6. From the **Add parametric sweep** list, choose **Yes**.
7. In the **Parameters** table, enter the following settings:

Index	Parameter name	Parameter value list	Parameter unit
1	`v_f`	`range(0.1,0.1,0.7)`	`1`

8. Locate the **Cell Properties** section. Click **Create Load Groups and Study** in the upper-right corner of the section.
9. Right-click **Cell Periodicity for Elastic Properties** and choose **Duplicate**.

Cell Periodicity for Thermal Properties

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Cell Periodicity for Elastic Properties 1**.
2. In the **Settings** window for **Cell Periodicity**, type `Cell Periodicity for Thermal Properties` in the **Label** text field.
3. Locate the **Cell Properties** section. From the **Boundary conditions** list, choose **Free expansion**.
4. From the **Calculate average properties** list, choose **Coefficient of thermal expansion**.

MATERIALS

Material Link 1: Epoxy Resin

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Matrix (Unidirectional Fiber Composite, Square Packing 1)**.

4 Locate the **Link Settings** section. Click **Blank Material**.

5 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material Link 1: Epoxy Resin (matlnk1)**.

6 Click **Go to Material**.

GLOBAL DEFINITIONS

Material 1: Epoxy Resin

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material 1 (mat1)**.

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	{Evector1, Evector2, Evector3}	`{E_m, E_m, E_m}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{nu_m, nu_m, nu_m}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{G_m, G_m, G_m}`	N/m²	Orthotropic
Density	rho	`rho_m`	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`alpha_m`	1/K	Basic

MATERIALS

Material Link 2: Carbon Fiber

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Fiber (Unidirectional Fiber Composite, Square Packing 1)**.

4 Locate the **Link Settings** section. Click **Blank Material**.

5 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material Link 2: Carbon Fiber (matlnk2)**.

6 Click **Go to Material**.

GLOBAL DEFINITIONS

Material 2: Carbon Fiber

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material 2 (mat2)**.

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

3 Locate the **Material Contents** section. 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
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

The **Effective Material** node computes the effective material properties from the material properties of the individual constituents based on different analytical rule of mixture (ROM) models. Add an **Effective Material** node for the each of the following ROM models: Voigt-Reuss, Modified Voigt-Reuss, Chamis, Halpin-Tsai, Halpin-Tsai-Nielsen, and Hashin-Rosen.

Effective Material 1 (effmat1)

In the **Model Builder** window, right-click **Materials** and choose **More Materials>Effective Material**.

Constituent 1 (effmat1.const1)

1 In the **Settings** window for **Constituent**, locate the **Link Settings** section.

2 From the **Material** list, choose **Material 1: Epoxy Resin (mat1)**.

Constituent 2 (effmat1.const2)

1 In the **Model Builder** window, right-click **Effective Material 1 (effmat1)** and choose **Constituent**.

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

3 From the **Material** list, choose **Material 2: Carbon Fiber (mat2)**.

4 Locate the **Volume Fraction** section. In the V_f text field, type `v_f`.

Effective Material: Voigt-Reuss Model (ROM)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Effective Material 1 (effmat1)**.

2 In the **Settings** window for **Effective Material**, type `Effective Material: Voigt-Reuss Model (ROM)` in the **Label** text field.

3 Locate the **Material Content** section. Click to select row number 2 in the table.

4 Click **Edit Mixing Rule**.

5 In the **Edit Mixing Rule** dialog box, choose **Voigt-Reuss model** from the **Mixing rule** list.

6 Click **Next Row (Store Changes)**.

7 From the **Mixing rule** list, choose **Voigt-Reuss model**.

8 Click **Next Row (Store Changes)**.

9 From the **Mixing rule** list, choose **Voigt-Reuss model**.

10 Click **Next Row (Store Changes)**.

11 From the **Mixing rule** list, choose **Voigt-Reuss model**.

12 Click **OK**.

13 Right-click **Effective Material: Voigt-Reuss Model (ROM)** and choose **Duplicate**.

Effective Material: Modified Voigt-Reuss Model (ROM)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Effective Material: Voigt-Reuss Model (ROM) 1 (effmat2)**.

2 In the **Settings** window for **Effective Material**, type `Effective Material: Modified Voigt-Reuss Model (ROM)` in the **Label** text field.

3 Locate the **Material Content** section. Click to select row number 3 in the table.

4 In the **Edit Mixing Rule** dialog box, choose **Modified Voigt-Reuss model** from the **Mixing rule** list.

5 Click **Next Row (Store Changes)**.

6 From the **Mixing rule** list, choose **Modified Voigt-Reuss model**.

7 Click **Next Row (Store Changes)**.

8 From the **Mixing rule** list, choose **Modified Voigt-Reuss model**.

9 Click **OK**.

10 Right-click **Effective Material: Modified Voigt-Reuss Model (ROM)** and choose **Duplicate**.

Effective Material: Chamis Model (ROM)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Effective Material: Modified Voigt-Reuss Model (ROM) 1 (effmat3)**.

2 In the **Settings** window for **Effective Material**, type `Effective Material: Chamis Model (ROM)` in the **Label** text field.

3 Locate the **Material Content** section. Click to select row number 2 in the table.

4 In the **Edit Mixing Rule** dialog box, choose **Chamis model** from the **Mixing rule** list.

5 Click **Next Row (Store Changes)**.

6 From the **Mixing rule** list, choose **Chamis model**.

7 Click **Next Row (Store Changes)**.

8 From the **Mixing rule** list, choose **Chamis model**.

9 Click **Next Row (Store Changes)**.

10 From the **Mixing rule** list, choose **Chamis model**.

11 Click **OK**.

12 Right-click **Effective Material: Chamis Model (ROM)** and choose **Duplicate**.

Effective Material: Halpin-Tsai Model (ROM)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Effective Material: Chamis Model (ROM) 1 (effmat4)**.

2 In the **Settings** window for **Effective Material**, type `Effective Material: Halpin-Tsai Model (ROM)` in the **Label** text field.

3 Locate the **Material Content** section. Click to select row number 3 in the table.

4 In the **Edit Mixing Rule** dialog box, choose **Halpin-Tsai model** from the **Mixing rule** list.

5 Specify the **Reinforcement factor** vector as

inf	11
0	22
0	33

6 Click **Next Row (Store Changes)**.

7 From the **Mixing rule** list, choose **Halpin-Tsai model**.

8 Click **Next Row (Store Changes)**.

9 From the **Mixing rule** list, choose **Halpin-Tsai model**.

10 Click **OK**.

11 Right-click **Effective Material: Halpin-Tsai Model (ROM)** and choose **Duplicate**.

Effective Material: Halpin-Tsai-Nielsen Model (ROM)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Effective Material: Halpin-Tsai Model (ROM) 1 (effmat5)**.

2 In the **Settings** window for **Effective Material**, type `Effective Material: Halpin-Tsai-Nielsen Model (ROM)` in the **Label** text field.

3 Locate the **Material Content** section. Click to select row number 3 in the table.

4 In the **Edit Mixing Rule** dialog box, choose **Halpin-Tsai-Nielsen model** from the **Mixing rule** list.

5 Specify the **Reinforcement factor** vector as

inf	11
0	22
0	33

6 Click **Next Row (Store Changes)**.

7 From the **Mixing rule** list, choose **Halpin-Tsai-Nielsen model**.

8 Click **Next Row (Store Changes)**.

9 From the **Mixing rule** list, choose **Halpin-Tsai-Nielsen model**.

10 Click **Next Row (Store Changes)**.

11 Click **OK**.

12 Right-click **Effective Material: Halpin-Tsai-Nielsen Model (ROM)** and choose **Duplicate**.

Effective Material: Hashin-Rosen Model (ROM)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Effective Material: Halpin-Tsai-Nielsen Model (ROM) 1 (effmat6)**.

2 In the **Settings** window for **Effective Material**, type `Effective Material: Hashin-Rosen Model (ROM)` in the **Label** text field.

3 Locate the **Material Content** section. Click to select row number 3 in the table.

4 In the **Edit Mixing Rule** dialog box, choose **Hashin-Rosen model** from the **Mixing rule** list.

5 Click **Next Row (Store Changes)**.

6 From the **Mixing rule** list, choose **Hashin-Rosen model**.

7 Click **Next Row (Store Changes)**.

8 From the **Mixing rule** list, choose **Hashin-Rosen model**.

9 Click **Next Row (Store Changes)**.

10 Click **OK**.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** 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**.

CELL PERIODICITY STUDY FOR ELASTIC PROPERTIES

In the first study, the homogenized elastic properties are computed. Therefore, disable the **Thermal Expansion** and **Cell Periodicity for Thermal Properties** features.

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.

Step 1: Stationary

1 In the **Model Builder** window, expand the **Cell Periodicity Study for Elastic Properties** 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** 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)>Cell Periodicity for Thermal Properties**.

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

8 In the **Home** toolbar, click **Compute**.

ADD STUDY

Next, add a new study to compute the homogenized thermal properties. For this study, disable the **Cell Periodicity for Elastic Properties** feature.

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

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

5 In the **Home** 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**.

RESULTS

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

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/Solution 2 (solidcp1solp)**.

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.

8 Select the **x-axis label** check box. In the associated text field, type v_f.

9 Select the **y-axis label** check box. In the associated text field, type E₁/E_m.

10 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
solid.cp1.D11/E_m	1	Nondimensional longitudinal Young's modulus
effmat1.Orthotropic.Evector 1/E_m	1	Nondimensional longitudinal Young's modulus
effmat2.Orthotropic.Evector 1/E_m	1	Nondimensional longitudinal Young's modulus
effmat3.Orthotropic.Evector 1/E_m	1	Nondimensional longitudinal Young's modulus
effmat4.Orthotropic.Evector 1/E_m	1	Nondimensional longitudinal Young's modulus
effmat5.Orthotropic.Evector 1/E_m	1	Nondimensional longitudinal Young's modulus
effmat6.Orthotropic.Evector 1/E_m	1	Nondimensional longitudinal Young's modulus

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 From the **Positioning** list, choose **Interpolated**.

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

8 In the table, enter the following settings:

Legends
Cell Periodicity
Voigt-Reuss Model (ROM)
Modified Voigt-Reuss Model (ROM)
Chamis Model (ROM)
Halpin-Tsai Model (ROM)
Halpin-Tsai-Nielsen Model (ROM)
Hashin-Rosen Model (ROM)

Duplicate or add this plot group twice in order to plot the remaining elastic properties. Use appropriate labels, titles, and expressions.

Longitudinal Coefficient of Thermal Expansion 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**, locate the **Data** section.

3 From the **Dataset** list, choose **Cell Periodicity Study for Thermal Properties/ Parametric Solutions 1 (sol9)**.

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

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

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

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

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

9 Locate the **Plot Settings** section.

10 Select the **x-axis label** check box. In the associated text field, type v_f.

11 Select the **y-axis label** check box. In the associated text field, type \alpha₁ /\alpha_m.

Global 1

1 Right-click **Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction** and choose **Global**.

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	Nondimensional longitudinal coefficient of thermal expansion
effmat1.def.alpha11/alpha_m	1	Nondimensional longitudinal coefficient of thermal expansion
effmat3.def.alpha11/alpha_m	1	Nondimensional longitudinal coefficient of thermal expansion

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 From the **Positioning** list, choose **Interpolated**.

6 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Cell Periodicity`
`Voigt-Reuss Model (ROM)`
`Chamis Model (ROM)`

8 In the **Longitudinal Coefficient of Thermal Expansion vs. Fiber Volume Fraction** toolbar, click **Plot**.

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 Poisso n’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.