Created in COMSOL Multiphysics 6.2

Micromechanical Model of a Piezoelectric Fiber Composite

Introduction

The monolithic piezoelectric materials found their use in many applications like sensors, actuators and transducers. The advancement of technology demands more advanced materials, which increases the usability and applications of composite piezoelectric materials. Compared to monolithic piezoelectric materials, composite piezoelectric material offers superior mechanical and thermal properties along with customized piezoelectric properties. These composite piezoelectric materials need complete set of constitutive properties in order to be analyzed numerically. A micromechanical analysis is one of the tools to obtain the homogenized electromechanical properties of piezoelectric composite materials.

The model is a benchmark example taken from the Ref. 1. In this example, a simplified micromechanical model of a unit cell with periodic boundary conditions is analyzed. The unit cell represents one type of piezocomposite from Ref. 1, which is a square-diagonal arrangement of fibers in a matrix. The homogenized electromechanical properties of the composite material are computed based on the individual properties of the fiber and matrix, fiber volume fraction, and the poling directions. The fibers in composite material is made of PZT-7A material, while the matrix is made of Barium Titanate. The fiber volume fraction is varied to check its effect on the homogenized properties. The poling directions of the matrix and fiber are parallel and aligned to the fiber axis.

Model Definition

The unit cell of cylindrical fibers embedded in a matrix is shown in Figure 1. The fiber radius is computed based on a parameterized fiber volume fraction.

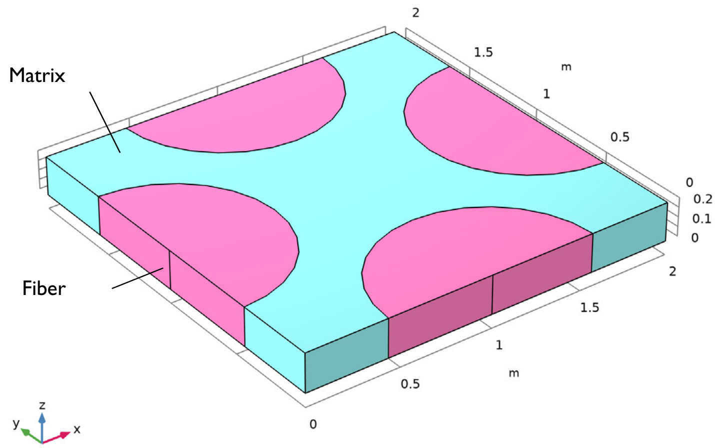

Figure 1: Geometry of the unit cell with a PZT-7A fibers in a Barium Titanate matrix.

Fiber and Matrix Properties

The fibers are made of PZT-7A material, and the matrix is made of Barium Titanate material, for which the built-in materials from the COMSOL Multiphysics material library are used.

The permittivity tensor at constant strain given in Ref. 1 for both materials is incorrect. It seems that the authors have instead used permittivity tensor at constant stress for the stress-charge formulation. The built-in materials in the COMSOL Multiphysics use correct permittivity or relative permittivity tensor, which in turn changes the result values of the homogenized permittivity tensor.

Results and Discussion

Figure 2 through Figure 6 show the homogenized elasticity tensor components for different fiber volume fractions. Some of the components increase linearly, while others decrease linearly with increase in the fiber volume fraction. The results very closely match those presented in Ref. 1 (see figure 4).

Figure 7 and Figure 8 show the homogenized piezoelectric coupling tensor components for different fiber volume fractions. Again, the results very closely match those presented in Ref. 1 (see figure 5). Figure 9 and Figure 10 show the homogenized permittivity tensor components for different fiber volume fractions. The homogenized permittivity tensor value varies with the fiber volume fraction in the same fashion as reported in Ref. 1, but the values do not match because the authors have used the permittivity tensor at constant stress for the fiber and matrix material.

Table 1, Table 2, and Table 3 show the nonzero components of the homogenized elasticity, homogenized piezoelectric coupling, and homogenized permittivity tensor, respectively. The values are for 70% fiber volume fraction. It is clear that the values presented in the reference paper closely match the values obtained with COMSOL Multiphysics. However, a match is not possible for the homogenized permittivity tensor because the authors used erroneous material inputs. In the current example, if one uses the permittivity tensor given by the authors then one gets exactly same homogenized permittivity tensor as presented in Ref. 1.

TABLE 1: HOMOGENIZED ELASTICITY TENSOR COMPONENTS.

Component	**Value from Ref. 1 in (GPa)**	**Value from COMSOL in (GPa)**
c_{11}	155	154.91
c_{12}	81	80.96
c_{13}	76.3	76.27
c_{22}	155	154.91
c_{23}	76.3	76.27
c_{33}	131.3	131.26
c_{44}	33.3	33.28
c_{55}	33.3	33.28
c_{66}	37.1	37.06

TABLE 2: HOMOGENIZED PIEZOELECTRIC COUPLING TENSOR COMPONENTS.

Component	**Value from Ref. 1 in (C/m^2)**	**Value from COMSOL in (C/m^2)**
e_{15}	10.9	10.97
e_{24}	10.9	10.97
e_{31}	-2.9	-2.94
e_{32}	-2.9	-2.94
e_{33}	11.8	11.80

TABLE 3: HOMOGENIZED PERMITTIVITY TENSOR COMPONENTS

Component	**Value from** Ref. 1 **in (nF/m)**	**Value from COMSOL in (nF/m)**
k_{11}	NA	5.762
k_{22}	NA	5.762
k_{33}	NA	4.753

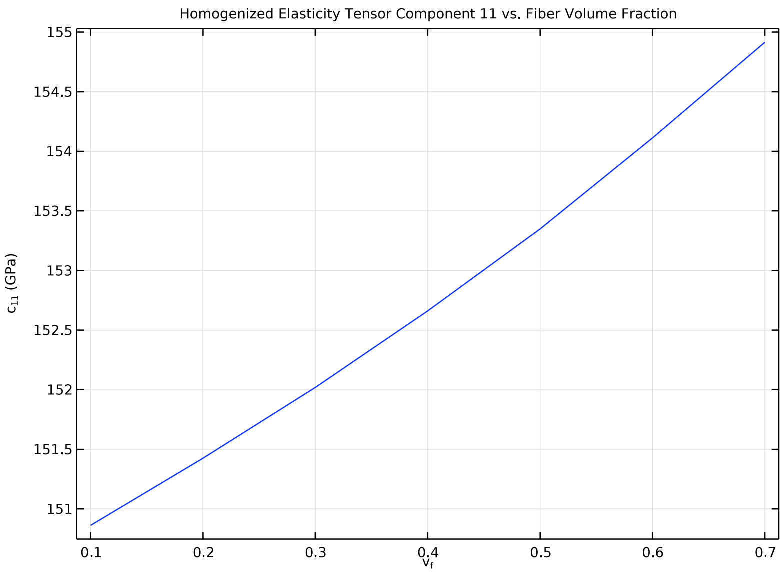

Figure 2: Homogenized elasticity tensor component 11 versus fiber volume fraction.

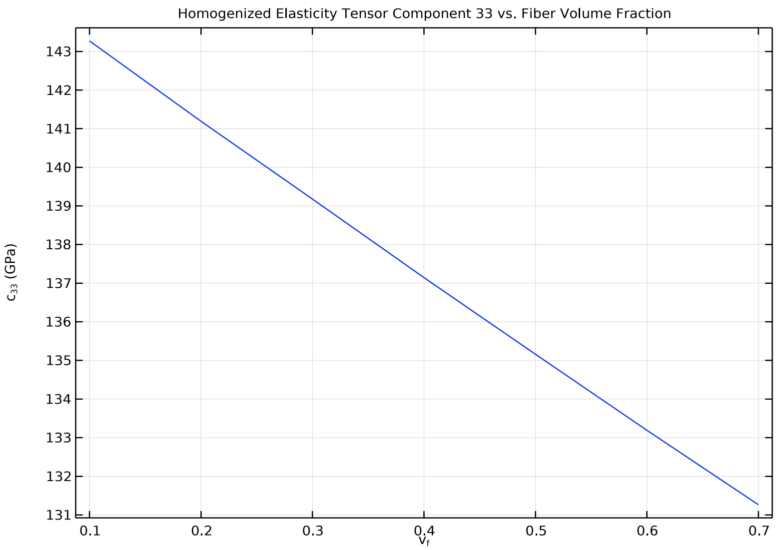

Figure 3: Homogenized elasticity tensor component 33 versus fiber volume fraction.

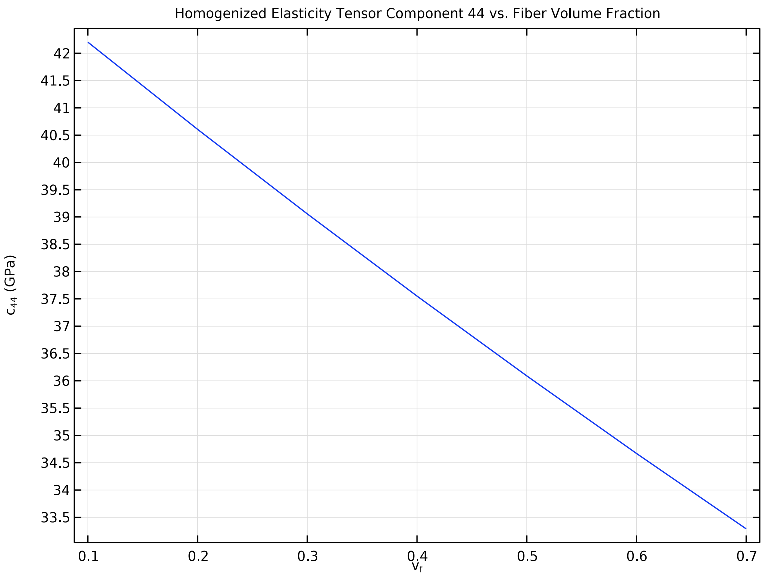

Figure 4: Homogenized elasticity tensor component 44 versus fiber volume fraction.

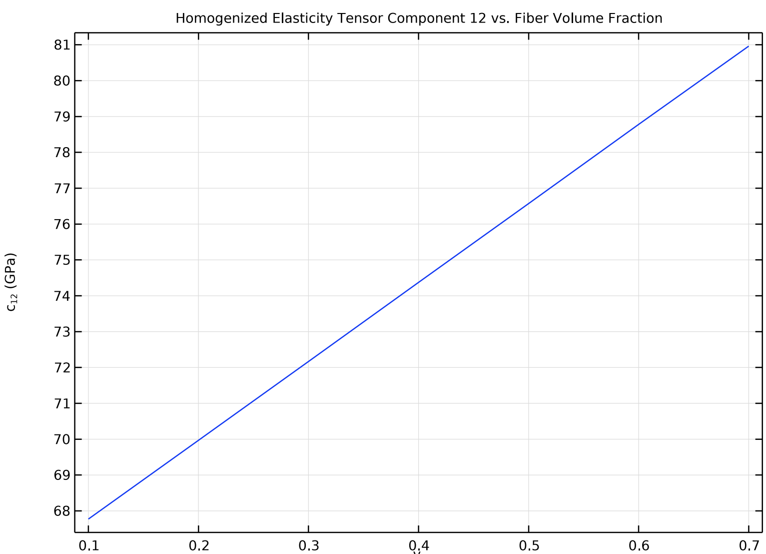

Figure 5: Homogenized elasticity tensor component 12 versus fiber volume fraction.

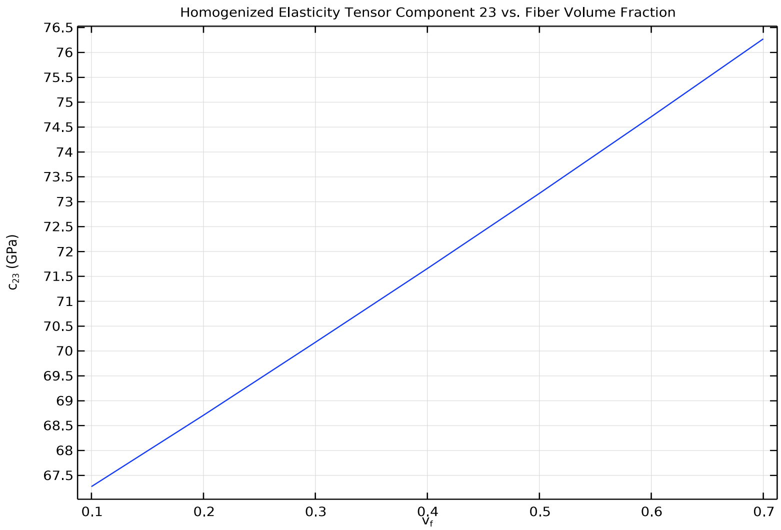

Figure 6: Homogenized elasticity tensor component 23 versus fiber volume fraction.

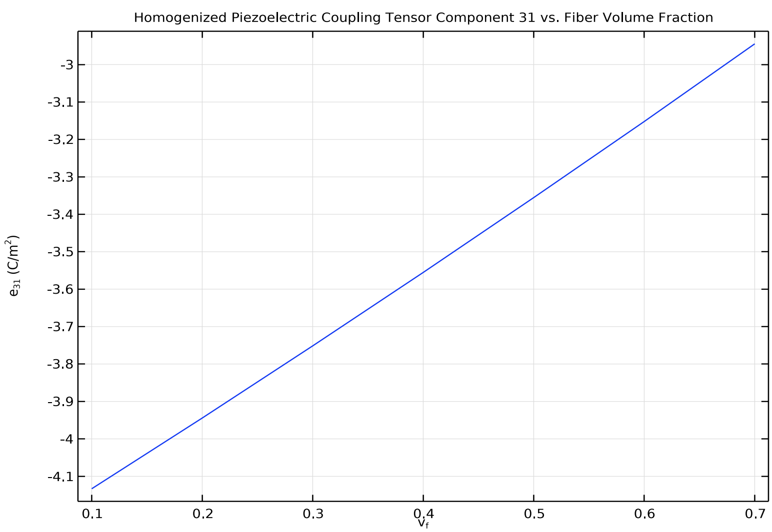

Figure 7: Homogenized piezoelectric coupling tensor component 31 versus fiber volume fraction.

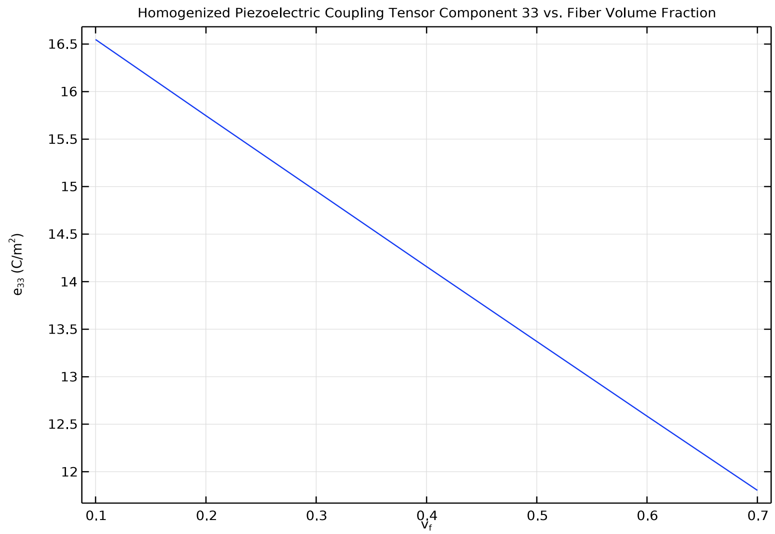

Figure 8: Homogenized piezoelectric coupling tensor component 33 versus fiber volume fraction.

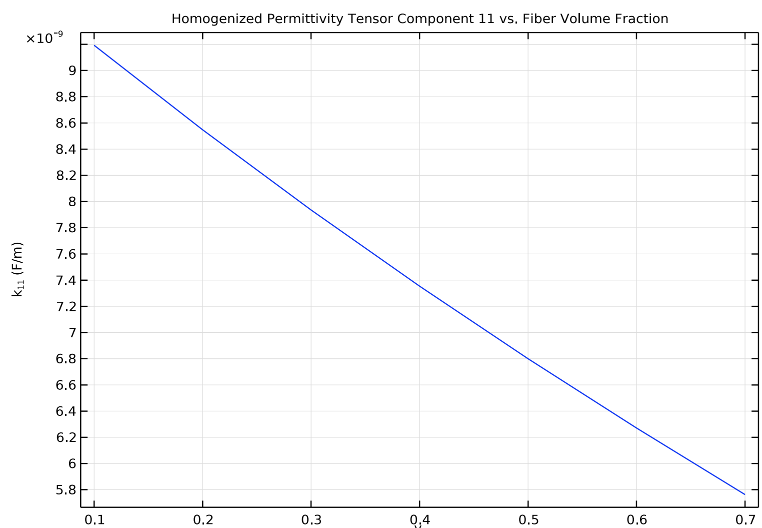

Figure 9: Homogenized permittivity tensor component 11 versus fiber volume fraction.

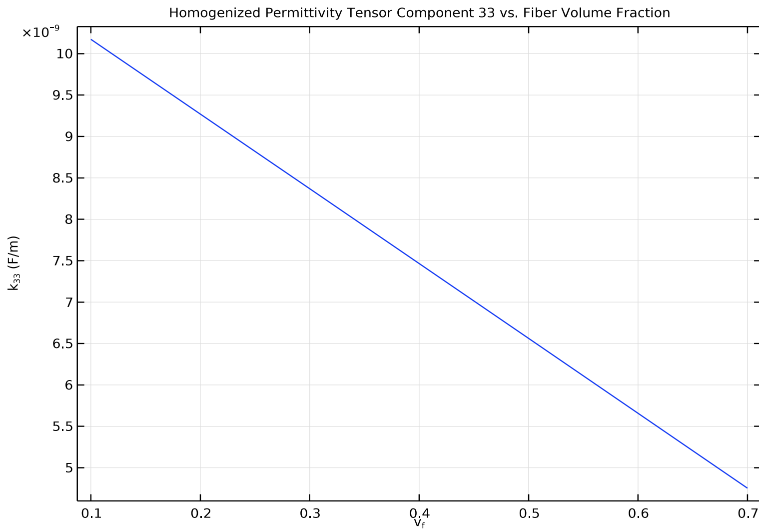

Figure 10: Homogenized permittivity tensor component 33 versus fiber volume fraction.

Notes About the COMSOL Implementation

- To get the homogenized elastic and piezoelectric coupling tensor, the **Cell Periodicity** node in the Solid Mechanics interface is used to apply periodic boundary conditions to the three pairs of faces of the unit cell.
- In order to get homogenized permittivity tensor, the periodic conditions are applied using **Pointwise Constraint** nodes in the Electrostatics interface. The **Piezoelectricity** multiphysics coupling is needed to couple the mechanical and electrical effects.
- 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 given options in the **Advanced** section of the feature.

Reference

1. R.K. Gupta and T.A. Venkatesh, “Electromechanical response of 1-3 piezoelectric composites: Effect of poling characteristics,” *J. Appl. Phys.*, vol. 98, no. 5, p. 054102, 2005.

Application Library path: `Structural_Mechanics_Module/Material_Models/micromechanical_model_of_a_piezoelectric_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>Electromagnetics–Structure Interaction>Piezoelectricity>Piezoelectricity, 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 In the table, enter the following settings:

Name	Expression	Value	Description
v_f	0.5	0.5	Fiber volume fraction
W	2[m]	2 m	Width of unit cell

Name	Expression	Value	Description
D	2[m]	2 m	Depth of unit cell
H	0.2[m]	0.2 m	Height of unit cell
V	W*D*H	0.8 m³	Volume of unit cell
V_f	v_f*V	0.4 m³	Volume of fibers
r_f	sqrt(V_f/(2*H*pi))	0.56419 m	Radius of cylindrical fibers

GEOMETRY 1

Block 1 (blk1)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Width** text field, type W.
4. In the **Depth** text field, type D.
5. In the **Height** text field, type H.
6. Click **Build Selected**.

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)>Circle 1 (c1)

1. In the **Work Plane** toolbar, click **Circle**.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type r_f.
4. In the **Sector angle** text field, type 180.
5. Locate the **Position** section. In the **yw** text field, type D/2.
6. Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.
7. Click **Build Selected**.

Work Plane 1 (wp1)>Circle 2 (c2)

1. In the **Work Plane** toolbar, click **Circle**.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type r_f.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **xw** text field, type `W`.

6 In the **yw** text field, type `D/2`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90`.

8 Click **Build Selected**.

Work Plane 1 (wp1)>Circle 3 (c3)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r_f`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **xw** text field, type `W/2`.

6 In the **yw** text field, type `D`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `180`.

8 Click **Build Selected**.

Work Plane 1 (wp1)>Circle 4 (c4)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r_f`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **xw** text field, type `W/2`.

6 Click **Build Selected**.

Fiber

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

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
H

4 In the **Label** text field, type `Fiber`.

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

6 From the **Color** list, choose **None** or — if you are running the cross-platform desktop — **Custom**. On the cross-platform desktop, click the **Color** button.

7 Click **Define custom colors**.

8 Set the RGB values to 255, 128, and 192, respectively.

9 Click **Add to custom colors**.

10 Click **Show color palette only** or **OK** on the cross-platform desktop.

11 Click **Build Selected**.

Matrix

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 In the **Settings** window for **Difference**, type `Matrix` in the **Label** text field.

3 Select the object **blk1** only.

4 Locate the **Difference** section. Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 From the **Objects to subtract** list, choose **Fiber**.

6 Select the **Keep objects to subtract** check box.

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

8 From the **Color** list, choose **None** or — if you are running the cross-platform desktop — **Custom**. On the cross-platform desktop, click the **Color** button.

9 Click **Define custom colors**.

10 Set the RGB values to 150, 240, and 240, respectively.

11 Click **Add to custom colors**.

12 Click **Show color palette only** or **OK** on the cross-platform desktop.

Form Union (fin)

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

Pair 1, Source

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 In the **Label** text field, type `Pair 1, Source`.

5 Locate the **Box Limits** section. In the **x maximum** text field, type `0`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Pair 1, Destination

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 In the **Label** text field, type `Pair 1, Destination`.

5 Locate the **Box Limits** section. In the **x minimum** text field, type `W`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Pair 2, Source

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 In the **Label** text field, type `Pair 2, Source`.

5 Locate the **Box Limits** section. In the **y maximum** text field, type `0`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Pair 2, Destination

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 In the **Label** text field, type `Pair 2, Destination`.

5 Locate the **Box Limits** section. In the **y minimum** text field, type `D`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Pair 3, Source

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 In the **Label** text field, type `Pair 3, Source`.

5 Locate the **Box Limits** section. In the **z maximum** text field, type `0`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Pair 3, Destination

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 In the **Label** text field, type `Pair 3, Destination`.

5 Locate the **Box Limits** section. In the **z minimum** text field, type `H`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Pair 1

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Pair 1, Source** and **Pair 1, Destination**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, type `Pair 1` in the **Label** text field.

Pair 2

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Pair 2, Source** and **Pair 2, Destination**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, type `Pair 2` in the **Label** text field.

Pair 3

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Pair 3, Source** and **Pair 3, Destination**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, type `Pair 3` in the **Label** text field.

ADD MATERIAL

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

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

3 In the tree, select **Piezoelectric>Barium Titanate (poled)**.

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

MATERIALS

Matrix: Barium Titanate (poled)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Barium Titanate (poled) (mat1)**.

2 In the **Settings** window for **Material**, type `Matrix: Barium Titanate (poled)` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Matrix**.

ADD MATERIAL

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

2 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-7A)**.

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

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

MATERIALS

Fiber: Lead Zirconate Titanate (PZT-7A)

1 In the **Settings** window for **Material**, type `Fiber: Lead Zirconate Titanate (PZT-7A)` in the **Label** text field.

2 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Fiber**.

SOLID MECHANICS (SOLID)

Cell Periodicity 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose the domain setting **More>Cell Periodicity**.
2. In the **Settings** window for **Cell Periodicity**, locate the **Cell Properties** section.
3. From the **Boundary conditions** list, choose **Average strain**.
4. From the **Calculate average properties** list, choose **Elasticity matrix, Voigt (XX, YY, ZZ, YZ, XZ, XY)**.

Boundary Pair 1

1. In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.
2. In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Pair 1**.
4. Right-click **Boundary Pair 1** and choose **Manual Destination Selection**.
5. Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 1, Destination**.

Cell Periodicity 1

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

Boundary Pair 2

1. In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.
2. In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Pair 2**.
4. Right-click **Boundary Pair 2** and choose **Manual Destination Selection**.
5. Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 2, Destination**.

Cell Periodicity 1

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

Boundary Pair 3

1. In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.
2. In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Pair 3**.
4. Right-click **Boundary Pair 3** and choose **Manual Destination Selection**.

5 Locate the **Destination Selection** section. From the **Selection** list, choose **Pair 3, Destination**.

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 1

To create a parametric study, use the options in the **Advanced** section of the feature. To see the section, activate **Advanced Physics** option from **Show** button.

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 1**.
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 Click **Automated Model Setup** in the upper-right corner of the **Cell Properties** section. From the menu, choose **Create Load Groups and Study**.

To compute homogenized permittivity tensor, three additional electric potential load cases are needed. Add three **Load Group** nodes under **Load Groups for Cell Periodicity** group.

GLOBAL DEFINITIONS

Load Group 1

1 In the **Model Builder** window, expand the **Global Definitions>Load and Constraint Groups** node.
2 Right-click **Load Groups for Cell Periodicity** and choose **Load Group**.

Load Group 2

Right-click **Load Groups for Cell Periodicity** and choose **Load Group**.

Load Group 3

Right-click **Load Groups for Cell Periodicity** and choose **Load Group**.

The periodic conditions in the **Electrostatics** interface need to be implemented using **Pointwise Constraint** features. To define the constraints, add three **General Extrusion** operators.

DEFINITIONS

General Extrusion 1 (genext1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **General Extrusion**.
2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Pair 1, Source**.
5 Click to expand the **Advanced** section. From the **Mesh search method** list, choose **Closest point**.
6 Right-click **General Extrusion 1 (genext1)** and choose **Duplicate**.

General Extrusion 2 (genext2)

1 In the **Model Builder** window, click **General Extrusion 2 (genext2)**.
2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.
3 From the **Selection** list, choose **Pair 2, Source**.
4 Right-click **General Extrusion 2 (genext2)** and choose **Duplicate**.

General Extrusion 3 (genext3)

1 In the **Model Builder** window, click **General Extrusion 3 (genext3)**.
2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.
3 From the **Selection** list, choose **Pair 3, Source**.
4 Click the **Show More Options** button in the **Model Builder** toolbar.
5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.
6 Click **OK**.

ELECTROSTATICS (ES)

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

Periodic Condition, Boundary Pair 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Pointwise Constraint**.

2 In the **Settings** window for **Pointwise Constraint**, type `Periodic Condition, Boundary Pair 1` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pair 1, Destination**.

4 Locate the **Pointwise Constraint** section. In the **Constraint expression** text field, type `V+group.lg1*1[V/m]*W-genext1(V)`.

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

Periodic Condition, Boundary Pair 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrostatics (es)** click **Periodic Condition, Boundary Pair 1.1**.

2 In the **Settings** window for **Pointwise Constraint**, type `Periodic Condition, Boundary Pair 2` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pair 2, Destination**.

4 Locate the **Pointwise Constraint** section. In the **Constraint expression** text field, type `V+group.lg2*1[V/m]*D-genext2(V)`.

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

Periodic Condition, Boundary Pair 3

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrostatics (es)** click **Periodic Condition, Boundary Pair 2.1**.

2 In the **Settings** window for **Pointwise Constraint**, type `Periodic Condition, Boundary Pair 3` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Pair 3, Destination**.

4 Locate the **Pointwise Constraint** section. In the **Constraint expression** text field, type `V+group.lg3*1[V/m]*H-genext3(V)`.

Ground 1

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

2 Select Point 1 only.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

Variables 1

1. In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.
3. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `micromechanical_model_of_a_piezoelectric_composite_variables.txt`.

MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
3. From the **Element size** list, choose **Extra fine**.
4. Click **Build All**.

Add three load cases corresponding to the three additional electric potential load cases.

CELL PERIODICITY STUDY

Step 1: Stationary

1. In the **Model Builder** window, expand the **Cell Periodicity Study** node, then click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, locate the **Study Extensions** section.
3. Click **Add**.
4. In the table, enter the following settings:

Load case	cp11	Weight	cp22	Weight	cp33	Weight	cp12	Weight	cp23	Weight	cp13	Weight	lg1	Weight	lg2	Weight	lg3	Weight
Load case 7		1.0		1.0		1.0		1.0		1.0		1.0	√	1.0		1.0		1.0

5. Click **Add**.

6 In the table, enter the following settings:

Load case	cp11	Weight	cp22	Weight	cp33	Weight	cp12	Weight	cp23	Weight	cp13	Weight	lg1	Weight	lg2	Weight	lg3	Weight
Load case 8		1.0		1.0		1.0		1.0		1.0		1.0		1.0	√	1.0		1.0

7 Click **Add**.

8 In the table, enter the following settings:

Load case	cp11	Weight	cp22	Weight	cp33	Weight	cp12	Weight	cp23	Weight	cp13	Weight	lg1	Weight	lg2	Weight	lg3	Weight
Load case 9		1.0		1.0		1.0		1.0		1.0		1.0		1.0		1.0	√	1.0

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

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

RESULTS

Homogenized Elasticity Tensor Component 11

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

2 In the **Settings** window for **1D Plot Group**, type `Homogenized Elasticity Tensor Component 11` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cell Periodicity Study/ 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 `Homogenized Elasticity Tensor Component 11 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<sub>f</sub>`.

9 Select the **y-axis label** check box. In the associated text field, type `c<sub>11</sub> (GPa)`.

10 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Homogenized Elasticity Tensor Component 11** 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	GPa	Elasticity matrix, 11-component

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. Duplicate or add this plot group five 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	Homogenized Elasticity Tensor Component 33	solid.cp1.D33
1D Plot Group 3	Homogenized Elasticity Tensor Component 44	solid.cp1.D44
1D Plot Group 4	Homogenized Elasticity Tensor Component 66	solid.cp1.D66
1D Plot Group 5	Homogenized Elasticity Tensor Component 12	solid.cp1.D12
1D Plot Group 6	Homogenized Elasticity Tensor Component 23	solid.cp1.D23

Homogenized Piezoelectric Coupling Tensor Component 31

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Homogenized Piezoelectric Coupling Tensor Component 31` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Cell Periodicity Study/ Solution 2 (solidcp1solp)**.
4. From the **Parameter selection (Load case)** list, choose **First**.
5. Locate the **Title** section. From the **Title type** list, choose **Manual**.
6. In the **Title** text area, type `Homogenized Piezoelectric Coupling Tensor Component 31 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<sub>f</sub>`.
9. Select the **y-axis label** check box. In the associated text field, type `e<sub>31</sub> (C/m<sup>2</sup>)`.
10. Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1. Right-click **Homogenized Piezoelectric Coupling Tensor Component 31** 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
`e31`	`C/m^2`	`Homogenized piezoelectric coupling tensor, 31 component`

4. Locate the **x-Axis Data** section. From the **Axis source data** list, choose **v_f**.
5. Duplicate or add this plot group two times in order to plot the remaining piezoelectric coupling 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 12	`Homogenized Piezoelectric Coupling Tensor Component 33`	`e33`
1D Plot Group 13	`Homogenized Piezoelectric Coupling Tensor Component 44`	`e15`

Homogenized Permittivity Tensor Component 11

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

2 In the **Settings** window for **1D Plot Group**, type `Homogenized Permittivity Tensor Component 11` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cell Periodicity Study/ Solution 2 (solidcp1solp)**.

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

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

6 In the **Title** text area, type `Homogenized Permittivity Tensor Component 11 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 k₁₁ (F/m).

10 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Homogenized Permittivity Tensor Component 11** 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
k11	F/m	Homogenized permittivity tensor, 11 component

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

5 Duplicate or add this plot group one time in order to plot the remaining permittivity 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 14	Homogenized Permittivity Tensor Component 33	k33

Homogenized Elasticity Tensor (70% Fiber Volume Fraction)

1 In the **Model Builder** window, under **Results** click **Material Properties (Cell Periodicity Study)**.

2 In the **Settings** window for **Evaluation Group**, type `Homogenized Elasticity Tensor (70% Fiber Volume Fraction)` in the **Label** text field.

Homogenized Piezoelectric Coupling Tensor (70% Fiber Volume Fraction)

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

2 In the **Settings** window for **Evaluation Group**, type `Homogenized Piezoelectric Coupling Tensor (70% Fiber Volume Fraction)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cell Periodicity Study/ Solution 2 (solidcp1solp)**.

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

5 From the **Parameter selection (Load case)** list, choose **Last**.

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

7 From the **Concatenation** list, choose **Vertical**.

Global Evaluation 1

1 Right-click **Homogenized Piezoelectric Coupling Tensor (70% Fiber Volume Fraction)** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
e11		Homogenized piezoelectric coupling tensor (C/m^2), 11 component
e12		Homogenized piezoelectric coupling tensor (C/m^2), 12 component
e13		Homogenized piezoelectric coupling tensor (C/m^2), 13 component
e14		Homogenized piezoelectric coupling tensor (C/m^2), 14 component
e15		Homogenized piezoelectric coupling tensor (C/m^2), 15 component

4 Right-click **Global Evaluation 1** and choose **Duplicate**.

Global Evaluation 2

1 In the **Model Builder** window, click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
e21		Homogenized piezoelectric coupling tensor (C/m^2), 21 component
e22		Homogenized piezoelectric coupling tensor (C/m^2), 22 component
e23		Homogenized piezoelectric coupling tensor (C/m^2), 23 component
e24		Homogenized piezoelectric coupling tensor (C/m^2), 24 component
e25		Homogenized piezoelectric coupling tensor (C/m^2), 25 component

4 Right-click **Global Evaluation 2** and choose **Duplicate**.

Global Evaluation 3

1 In the **Model Builder** window, click **Global Evaluation 3**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
e31		Homogenized piezoelectric coupling tensor (C/m^2), 31 component
e32		Homogenized piezoelectric coupling tensor (C/m^2), 32 component
e33		Homogenized piezoelectric coupling tensor (C/m^2), 33 component
e34		Homogenized piezoelectric coupling tensor (C/m^2), 34 component
e35		Homogenized piezoelectric coupling tensor (C/m^2), 35 component

4 In the **Homogenized Piezoelectric Coupling Tensor (70% Fiber Volume Fraction)** toolbar, click **Evaluate**.

Homogenized Permittivity Tensor (70% Fiber Volume Fraction)

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

2 In the **Settings** window for **Evaluation Group**, type `Homogenized Permittivity Tensor (70% Fiber Volume Fraction)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cell Periodicity Study/ Solution 2 (solidcp1solp)**.

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

5 From the **Parameter selection (Load case)** list, choose **Last**.

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

7 From the **Concatenation** list, choose **Vertical**.

Global Evaluation 1

1 Right-click **Homogenized Permittivity Tensor (70% Fiber Volume Fraction)** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
k11		Homogenized permittivity tensor (F/m), 11 component
k12		Homogenized permittivity tensor (F/m), 12 component
k13		Homogenized permittivity tensor (F/m), 13 component

4 Right-click **Global Evaluation 1** and choose **Duplicate**.

Global Evaluation 2

1 In the **Model Builder** window, click **Global Evaluation 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
k12		Homogenized permittivity tensor (F/m), 12 component
k22		Homogenized permittivity tensor (F/m), 22 component
k23		Homogenized permittivity tensor (F/m), 23 component

4 Right-click **Global Evaluation 2** and choose **Duplicate**.

Global Evaluation 3

1 In the **Model Builder** window, click **Global Evaluation 3**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
k13		Homogenized permittivity tensor (F/m), 13 component
k23		Homogenized permittivity tensor (F/m), 23 component
k33		Homogenized permittivity tensor (F/m), 33 component

4 In the **Homogenized Permittivity Tensor (70% Fiber Volume Fraction)** toolbar, click **Evaluate**.