



Model created in COMSOL Multiphysics 6.4

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.

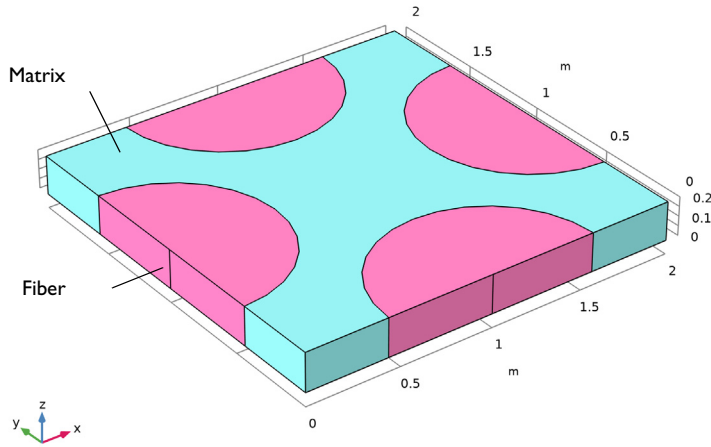


Figure 1: Geometry of the unit cell with a PZT-7A fibers in a Barium Titanate matrix.

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

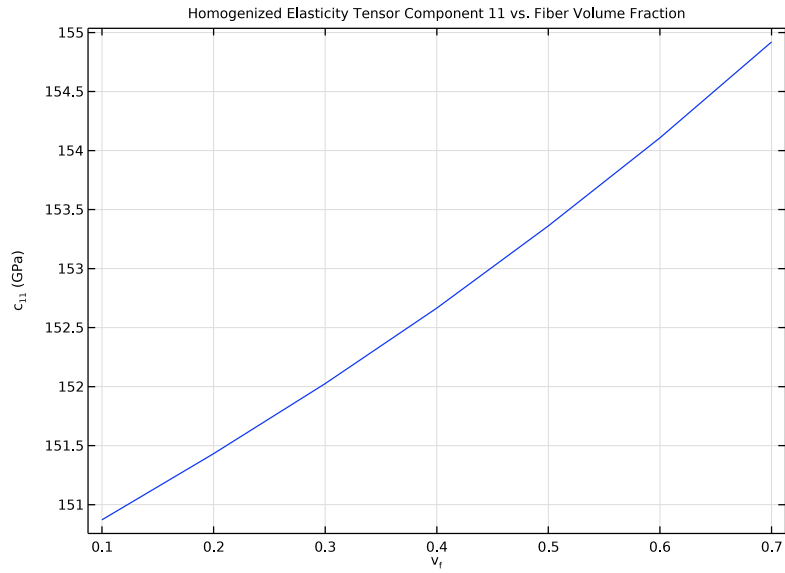


Figure 2: Homogenized elasticity tensor component 11 versus fiber volume fraction.

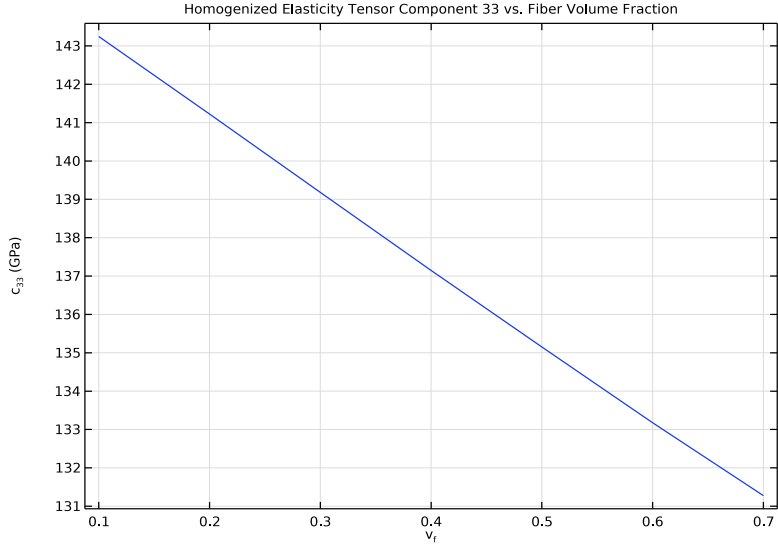


Figure 3: Homogenized elasticity tensor component 33 versus fiber volume fraction.

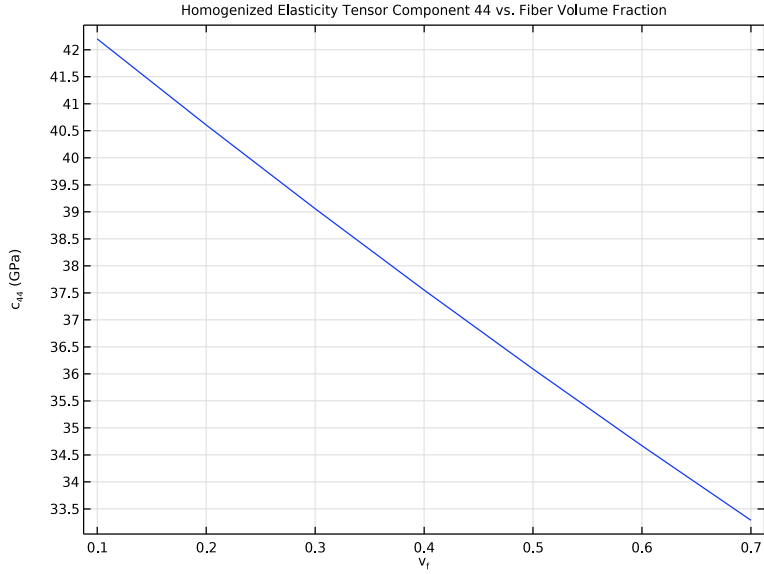


Figure 4: Homogenized elasticity tensor component 44 versus fiber volume fraction.

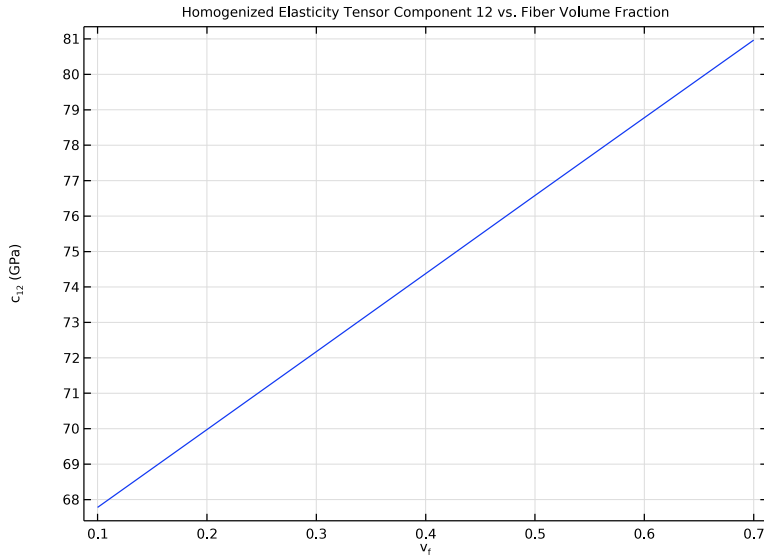


Figure 5: Homogenized elasticity tensor component 12 versus fiber volume fraction.

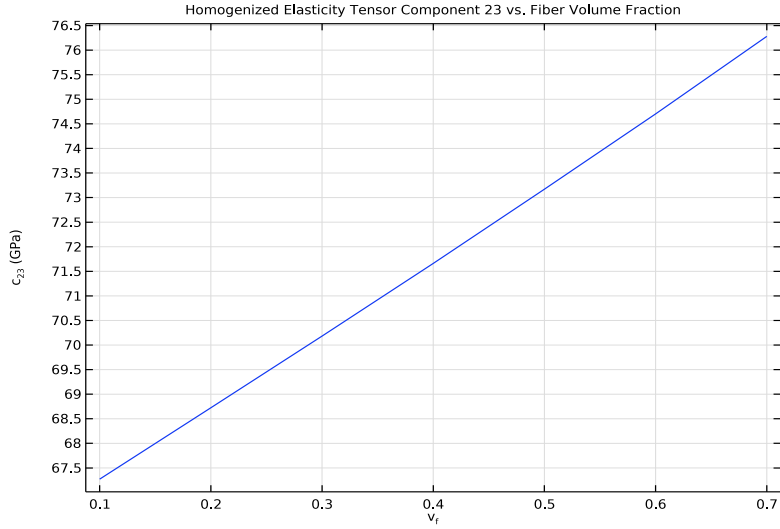


Figure 6: Homogenized elasticity tensor component 23 versus fiber volume fraction.

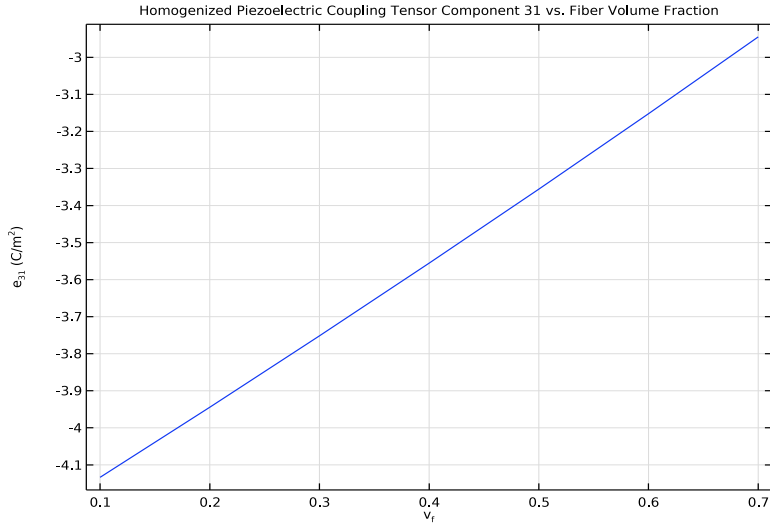


Figure 7: Homogenized piezoelectric coupling tensor component 31 versus fiber volume fraction.

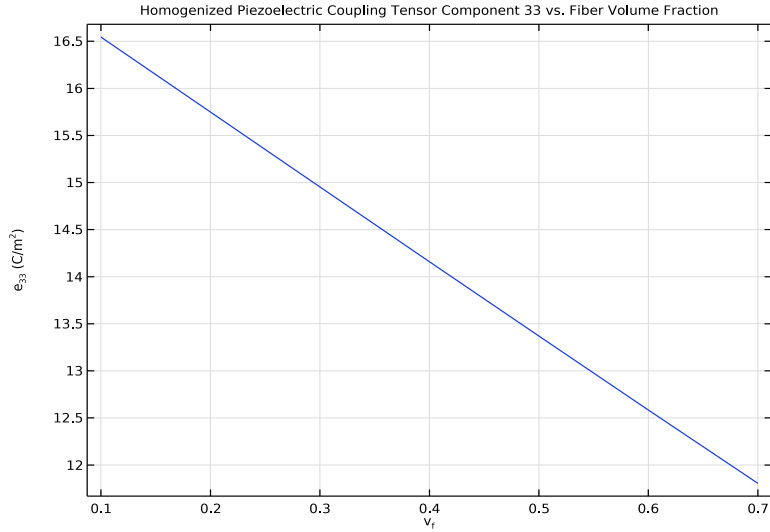


Figure 8: Homogenized piezoelectric coupling tensor component 33 versus fiber volume fraction.

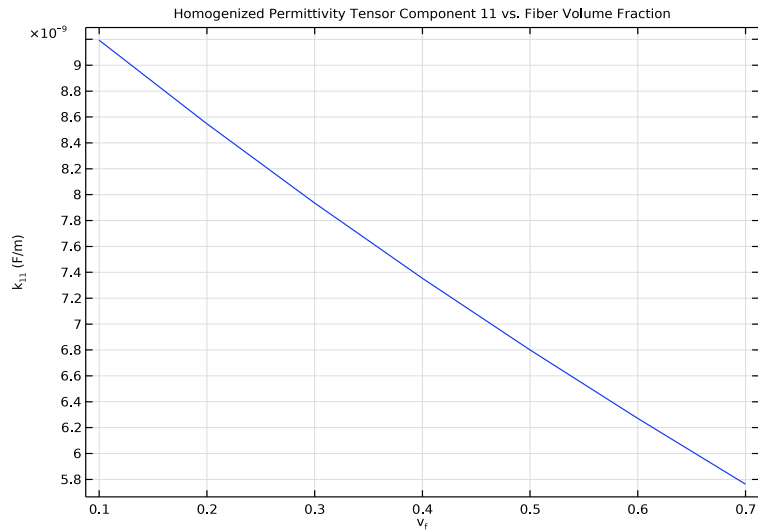


Figure 9: Homogenized permittivity tensor component 11 versus fiber volume fraction.

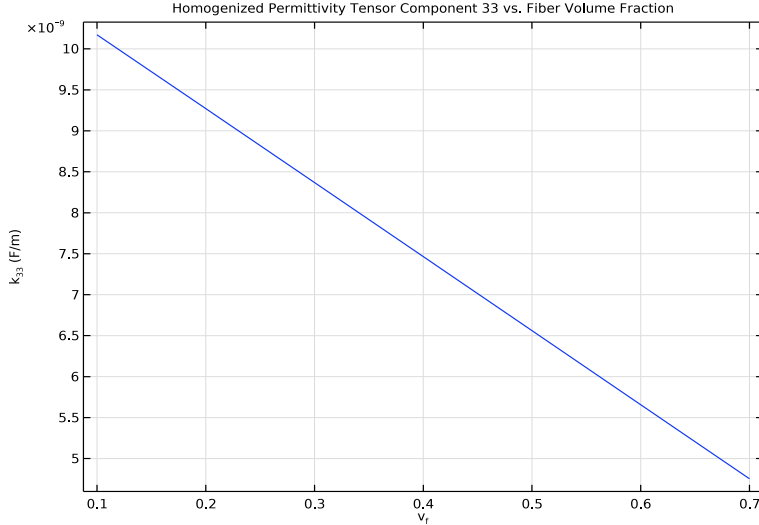


Figure 10: Homogenized permittivity tensor component 33 versus fiber volume fraction.

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 .
- 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 m ³	Volume of unit cell
V_f	v_f*V	0.4 m ³	Volume of fibers
r_f	sqrt(V_f/(2*H*pi))	0.56419 m	Radius of cylindrical fibers

GEOMETRY I

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** checkbox.

- 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** checkbox.
- 7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.
- 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, 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, 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, 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 I (comp1)**.

MATERIALS

Matrix: Barium Titanate (poled)

- 1 In the **Model Builder** window, under **Component I (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 I (comp1)**.
- 4 In the **Materials** 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 **Physics** toolbar, click  **Domains** and choose **Cell Periodicity**.

- 2 In the **Settings** window for **Cell Periodicity**, locate the **Periodicity Settings** section.
- 3 From the **Boundary conditions** list, choose **Average strain**.
- 4 Locate the **Effective Properties** section. Select the **Compute elasticity matrix**, **Voigt notation** checkbox.


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 selected 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 this section, activate the **Advanced Physics** option.

- 1 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 2 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics > Advanced Physics Options**.
- 3 Click **OK**.
- 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 **Periodicity Settings** section. From the menu, choose **Create Load Groups and Study**.

To compute the homogenized permittivity tensor, three additional electric potential load cases are needed. Add three **Load Group** nodes under the **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**.

General Extrusion 2 (genext2)


- 1 Right-click **General Extrusion 1 (genext1)** and choose **Duplicate**.
- 2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.
- 3 From the **Selection** list, choose **Pair 2, Source**.

General Extrusion 3 (genext3)

- 1 Right-click **General Extrusion 2 (genext2)** and choose **Duplicate**.
- 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, in the tree, select the checkbox for the node **Physics > Equation Contributions**.
- 6 Click **OK**.

ELECTROSTATIC (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 $V + \text{group.lg1} * 1 [\text{V/m}] * W - \text{genext1} (V)$.


Periodic Condition, Boundary Pair 2

- 1 Right-click **Periodic Condition, Boundary Pair 1** and choose **Duplicate**.
- 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 $V + \text{group.lg2} * 1 [V/m] * D - \text{genext2}(V)$.

Periodic Condition, Boundary Pair 3


- 1 Right-click **Periodic Condition, Boundary Pair 2** and choose **Duplicate**.
- 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 $V + \text{group.lg3} * 1 [V/m] * H - \text{genext3}(V)$.

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 **Finer**.

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** three times.

4 In the table, associate the new load cases with load groups as follows:


Load case	lg1	Weight	lg2	Weight	lg3	Weight
Load case 7	√	1.0		1.0		1.0
Load case 8		1.0	√	1.0		1.0
Load case 9		1.0		1.0	√	1.0

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

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

RESULTS

Homogenized Elasticity Tensor Component 11

1 In the **Results** toolbar, click  **ID Plot Group**.

2 In the **Settings** window for **ID 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** checkbox. In the associated text field, type v_f .

9 Select the **y-axis label** checkbox. In the associated text field, type c_{11} (GPa).

10 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global I

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.DVo11	GPa	Elasticity matrix, Voigt notation, 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 I** node are shown in the table below.

Name	Label/Title	Expressions in global node
ID Plot Group 2	Homogenized Elasticity Tensor Component 33	solid.cp1.DVo33
ID Plot Group 3	Homogenized Elasticity Tensor Component 44	solid.cp1.DVo44
ID Plot Group 4	Homogenized Elasticity Tensor Component 66	solid.cp1.DVo66
ID Plot Group 5	Homogenized Elasticity Tensor Component 12	solid.cp1.DVo12
ID Plot Group 6	Homogenized Elasticity Tensor Component 23	solid.cp1.DVo23

Homogenized Piezoelectric Coupling Tensor Component 31

1 In the **Results** toolbar, click  **ID Plot Group**.

2 In the **Settings** window for **ID 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** checkbox. In the associated text field, type v_f .
- 9 Select the **y-axis label** checkbox. In the associated text field, type e_{31} (C/m^2).
- 10 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global I


- 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 twice in order to plot the remaining piezoelectric coupling properties. The labels, titles, and expressions to be defined in the **Global I** node are shown in the table below.

Name	Label/Title	Expressions in global node
ID Plot Group 12	Homogenized Piezoelectric Coupling Tensor Component 33	e33
ID Plot Group 13	Homogenized Piezoelectric Coupling Tensor Component 44	e15

Homogenized Permittivity Tensor Component 11

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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** checkbox. In the associated text field, type v_f .
- 9 Select the **y-axis label** checkbox. In the associated text field, type k_{11} (F/m).
- 10 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global I

- 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 expressions to be defined in the **Global I** node are shown in the table below.

Name	Label/Title	Expressions in global node
ID 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 **Effective 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	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

Global Evaluation 2

- 1 Right-click **Global Evaluation 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	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


Global Evaluation 3

- 1 Right-click **Global Evaluation 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	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	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

Global Evaluation 2

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

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

3 In the table, enter the following settings:

Expression	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


Global Evaluation 3

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

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

3 In the table, enter the following settings:

Expression	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**.