Composite Transducer

Introduction

This example shows how to set up a piezoelectric transducer model following the work of Y. Kagawa and T. Yamabuchi (Ref. 1). The composite piezoelectric ultrasonic transducer has a cylindrical geometry that consists of a piezoceramic (NEPEC 6) layer, two aluminum layers, and two adhesive layers. The layers are organized as follows: aluminum layer–adhesive layer–piezoceramic layer–adhesive layer–aluminum layer.

The system applies an AC potential on the electrode surfaces of both sides of the piezoceramic layer. The potential in this example has a peak value of 1 V in the frequency range 20 kHz to 106 kHz. The goal is to compute the susceptance (the imaginary part of the admittance) $Y = I/V$, where I is the total current and V is the potential, for a frequency range around the four lowest eigenfrequencies of the structure.

The first step finds the eigenmodes, and the second step runs a frequency sweep across an interval that encompasses the first four eigenfrequencies. Both analyses are fully coupled, and COMSOL Multiphysics assembles and solves both the electric and mechanical parts of the problem simultaneously.

The problem is axially symmetric, and you could analyze it using a 2D axisymmetric model. However, to illustrate the modeling principles for more complicated problems, this example is in 3D.

When creating the model geometry, you make use of the symmetry by first making a cut along a midplane perpendicular to the central axis and then cutting out a 10-degree wedge; doing so reduces memory requirements significantly.

Model Data

The model uses the following material data.

NEPEC 6 MATERIAL PARAMETERS

TABLE 1: ELASTICITY MATRIX c_E

12.8e10	6.8e10	6.6e10	0	0	0
	12.8e10	6.6e10	0	0	0
		11.0e10	0	0	0
			2.1e10	0	0
				2.1e10	0
					2.1e10

TABLE 2: COUPLING MATRIX e

0	0	0	0	0	0
0	0	0	0	0	0
-6.1	-6.1	15.7	0	0	0

TABLE 3: RELATIVE PERMITTIVITY ε_{rS}

993.53	0	0
	993.53	0
		993.53

ALUMINUM MATERIAL PARAMETERS

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	7.03e10	Young's modulus
nu	0.345	Poisson's ratio
rho	2690	Density

ADHESIVE MATERIAL PARAMETERS

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	1e10	Young's modulus
nu	0.38	Poisson's ratio
rho	1700	Density

Results and Discussion

Figure 1 shows the lowest vibration mode of the piezoelectric transducer while Figure 2 shows the transducer's input susceptance as a function of the excitation frequency.

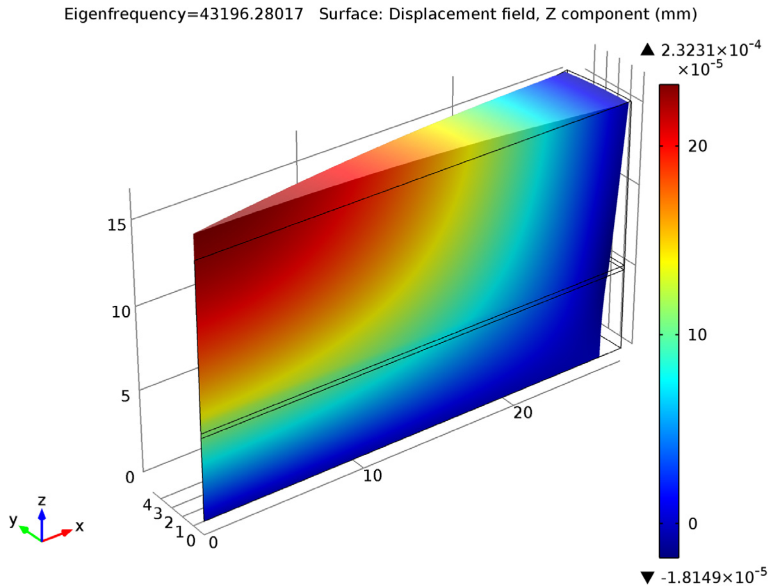


Figure 1: The lowest vibration eigenmode of the transducer.

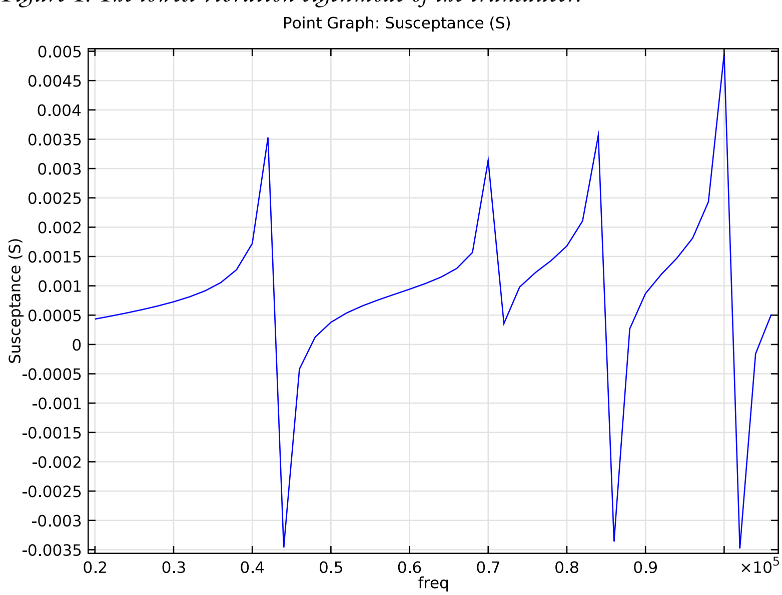


Figure 2: Input susceptance as a function of excitation frequency.

The result is in agreement with the work in Ref. 1. A small discrepancy close to the eigenfrequencies appears because the simulation uses no damping.

Reference

1. Y. Kagawa and T. Yamabuchi, "Finite Element Simulation of a Composite Piezoelectric Ultrasonic Transducer," *IEEE Transactions on Sonics and Ultrasonics*, vol. SU-26, no. 2, pp. 81–88, 1979.

Model Library path: `MEMS_Module/Piezoelectric_Devices/composite_transducer`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

2 In the **Select physics** tree, select **Structural Mechanics>Piezoelectric Devices (pzd)**.

3 Click the **Add** button.

4 Click the **Study** button.

5 In the tree, select **Preset Studies>Eigenfrequency**.

6 Click the **Done** button.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

4 On the **Geometry** toolbar, click **Work Plane**.

Circle 1

1 In the **Model Builder** window, under **Component 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Circle**.

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

3 In the **Radius** edit field, type 27.5.

4 Click the **Build Selected** button.

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

Bézier Polygon 1

1 Right-click **Plane Geometry** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **2**, set **xw** to 30.

5 Click the **Build Selected** button.

Rotate 1

1 On the **Work plane** toolbar, click **Rotate**.

2 Select the object **b1** only.

3 In the **Rotate** settings window, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Rotation Angle** section. In the **Rotation** edit field, type 10.

6 Click the **Build Selected** button.

Convert to Solid 1

1 On the **Work plane** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Click the **Select Box** button in the **Graphics** window and then press Ctrl+A to select all objects.

3 Click the **Build Selected** button.

Delete Entities 1

1 Right-click **Plane Geometry** and choose **Delete Entities**.

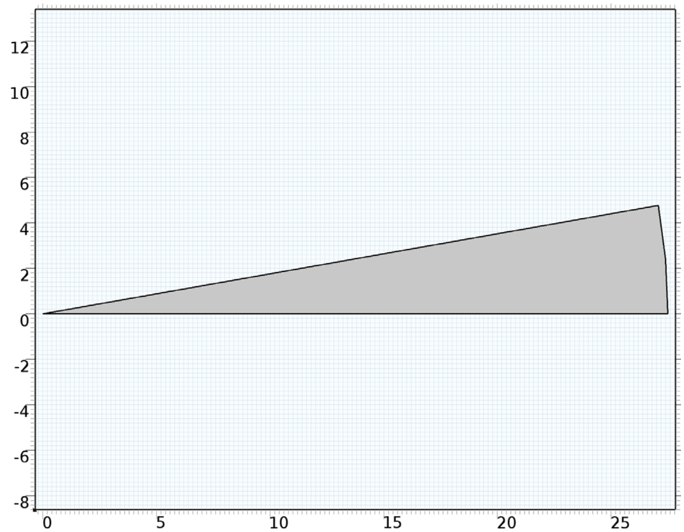
2 In the **Delete Entities** settings window, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select the larger of the two sectors.

5 Click the **Build Selected** button.

6 Click the **Zoom Extents** button on the Graphics toolbar.



Extrude 1

1 Close the **Work plane** toolbar. On the **Geometry** toolbar, click **Extrude**.

2 In the **Extrude** settings window, locate the **Distances from Plane** section.

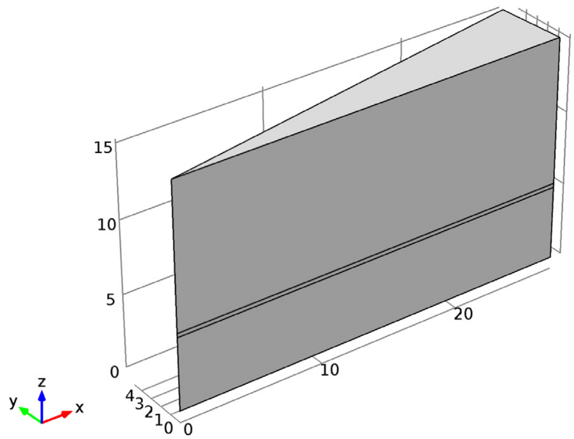
3 In the table, enter the following settings:

Distances (mm)
5
5.275
15.275

4 Click the **Build All Objects** button.

5 Click the **Go to Default 3D View** button on the Graphics toolbar.

This completes the geometry modeling stage.



PIEZOELECTRIC DEVICES

Linear Elastic Material 1

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

2 Select Domains 2 and 3 only.

MATERIALS

Material 1

1 In the **Model Builder** window, under **Component 1** right-click **Materials** and choose **New Material**.

2 Right-click **Material 1** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Nepec 6` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 1 only.

6 In the **Material** settings window, click to expand the **Material properties** section.

7 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Density**.

8 Click **Add to Material**.

9 In the **Material properties** tree, select **Piezoelectric Models>Stress-Charge Form**.

10 Click **Add to Material**.

Nepec 6

1 In the **Model Builder** window, expand the **Component 1>Materials>Nepec 6** node, then click **Basic**.

2 In the **Property Group** settings window, locate the **Output Properties and Model Inputs** section.

3 Find the **Output properties** subsection. In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Density	rho	7730	kg/m³	1x1

4 In the **Model Builder** window, under **Component 1>Materials>Nepec 6** click **Stress-charge form (StressCharge)**.

5 In the **Property Group** settings window, locate the **Output Properties and Model Inputs** section.

6 Find the **Output properties** subsection. In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Elasticity matrix (Ordering: xx, yy, zz, yz, xz, xy)	{cE11, cE12, cE22, cE13, cE23, cE33, cE14, cE24, cE34, cE44, cE15, cE25, cE35, cE45, cE55, cE16, cE26, cE36, cE46, cE56, cE66} ; cEij = cEji	`{12.8e10, 6.8e10, 12.8e10, 6.6e10, 6.6e10, 11.0e10, 0[Pa], 0[Pa], 0[Pa], 2.1e10, 0[Pa], 0[Pa], 0[Pa], 0[Pa], 2.1e10, 0[Pa], 0[Pa], 0[Pa], 0[Pa], 0[Pa], 2.1e10}`	Pa	6x6
Coupling matrix	{eES11, eES21, eES31, eES12, eES22, eES32, eES13, eES23, eES33, eES14, eES24, eES34, eES15, eES25, eES35, eES16, eES26, eES36}	`{0[C/m^2], 0[C/m^2], -6.1, 0[C/m^2], 0[C/m^2], -6.1, 0[C/m^2], 0[C/m^2], 15.7, 0[C/m^2], 0, 0[C/m^2], 0, 0[C/m^2], 0[C/m^2], 0[C/m^2], 0[C/m^2], 0[C/m^2]}`	C/m²	3x6
Relative permittivity	epsilonrS ; epsilonrSii = epsilonrS, epsilonrSij = 0	`993.53`	1	3x3

Alternatively, to define the symmetric elasticity matrix, cE, and the full coupling matrix, eES, you can click the **Edit** button below the Output properties table and use the matrix input dialogs to enter the data as given in section NEPEC 6 Material Parameters.

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **New Material**.

2 Right-click **Material 2** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Adhesive` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 2 only.

6 In the **Material** settings window, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	1e10	Pa	Basic
Poisson's ratio	nu	0.38	1	Basic
Density	rho	1700	kg/m³	Basic

Material 3

1 In the **Model Builder** window, right-click **Materials** and choose **New Material**.

2 Right-click **Material 3** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Aluminum` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 3 only.

6 In the **Material** settings window, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	7.03e10	Pa	Basic
Poisson's ratio	nu	0.345	1	Basic
Density	rho	2690	kg/m³	Basic

PIEZOELECTRIC DEVICES

Terminal 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Terminal**.

2 Select Boundary 6 only.

3 In the **Terminal** settings window, locate the **Terminal** section.

4 From the **Terminal type** list, choose **Voltage**.

5 In the V_0 edit field, type `0.5`.

This is half of the total peak voltage between the terminals, which accounts for modeling only the upper half of the transducer.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

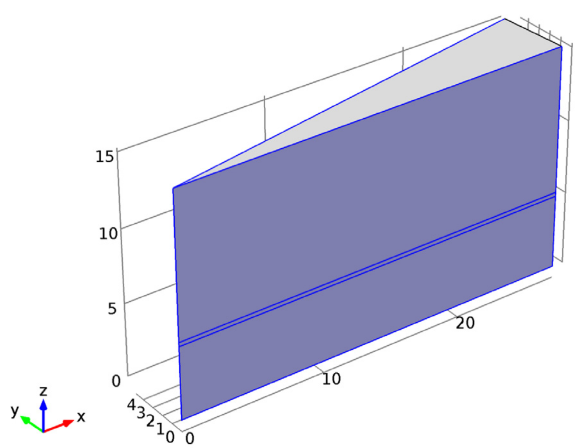
2 Select Boundary 3 only.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1–5, 7, and 8 only.

3 Click the **Go to Default 3D View** button on the Graphics toolbar.



DEFINITIONS

Before generating the mesh, define a variable for the susceptance.

Variables 1

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

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
B	imag(pzd.Y11)*36/2		Susceptance

In the above expression, the factor 36 compensates for the fact that the model is only a 10 degree wedge of the full transducer. Moreover, the factor 1/2 accounts for modeling only the upper half of the transducer because of symmetry in the z direction.

MESH 1

1 In the **Model Builder** window, under **Component 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Finer**.

Free Triangular 1

1 Right-click **Component 1>Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundary 3 only.

Distribution 1

1 Right-click **Component 1>Mesh 1>Free Triangular 1** and choose **Distribution**.

2 Select Edges 2 and 3 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

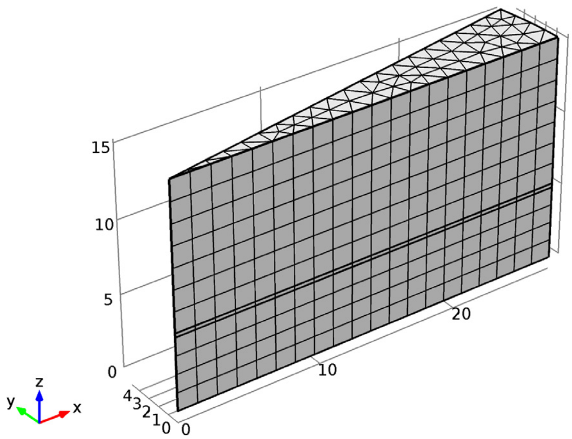
5 In the **Number of elements** edit field, type 20.

6 Click the **Build Selected** button.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Build Selected**.



STUDY 1

On the **Home** toolbar, click **Compute**.

RESULTS

Displacement (pzd)

1. In the **Model Builder** window, expand the **Displacement (pzd)** node, then click **Surface 1**.
2. In the **Surface** settings window, locate the **Expression** section.
3. Select **Piezoeelectric Devices (Solid Mechanics)>Displacement>Displacement field (Material)>Displacement field, Z component (w)** in the **Replace expression** menu accessible from the upper-right corner of the section.
4. On the **3D plot group** toolbar, click **Plot**.

Compare the resulting plot to that in Figure 1.

ROOT

Next, add a separate study for the frequency sweep.

1. On the **Home** toolbar, click **Add Study**.

ADD STUDY

1 Go to the **Add Study** window.
2 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
3 In the **Add study** window, click **Add Study**.

STUDY 2

Step 1: Frequency Domain

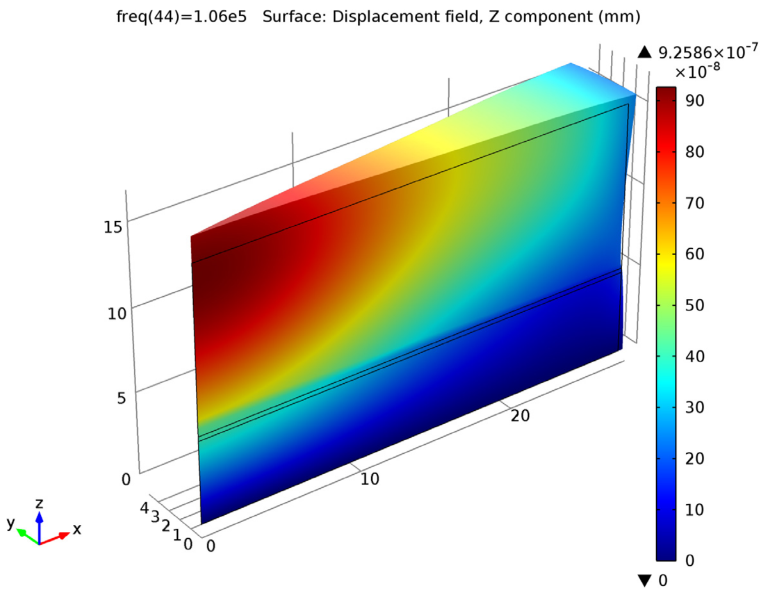
1 In the **Frequency Domain** settings window, locate the **Study Settings** section.
2 Click the **Range** button.
3 Go to the **Range** dialog box.
4 In the **Start** edit field, type `20e3`.
5 In the **Stop** edit field, type `106e3`.
6 In the **Step** edit field, type `2e3`.
7 Click the **Replace** button.
8 On the **Home** toolbar, click **Compute**.

RESULTS

Displacement (pzd) 1

1 In the **Model Builder** window, expand the **Displacement (pzd) 1** node, then click **Surface 1**.
2 In the **Surface** settings window, locate the **Expression** section.
3 Select **Piezoeelectric Devices (Solid Mechanics)>Displacement>Displacement field (Material)>Displacement field, Z component (w)** in the **Replace expression** menu accessible from the upper-right corner of the section.

4 On the **3D plot group** toolbar, click **Plot**.



1D Plot Group 5

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

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

3 From the **Data set** list, choose **Solution 2**.

4 On the **1D plot group** toolbar, click **Point Graph**.

5 Select Point 1 only.

6 In the **Point Graph** settings window, locate the **y-axis data** section.

7 Select **Definitions>Susceptance (B)** from the **Replace expression** menu accessible from the upper-right corner of the section. On the **1D plot group** toolbar, click **Plot**.

Compare the result to that in Figure 2.