Model created in COMSOL Multiphysics 6.4

Composite Piezoelectric 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.

Although you could analyze this problem using a 2D axisymmetric model, in order to illustrate the modeling principles for more complicated problems, this example uses a 3D geometry.

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 by 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.

128 GPa	68 GPa	66 GPa	0	0	0
	128 GPa	66 GPa	0	0	0
		110 GPa	0	0	0
			21 GPa	0	0
				21 GPa	0
					21 GPa

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	70.3 GPa	Young's modulus
nu	0.345	Poisson's ratio
rho	2690	Density

ADHESIVE MATERIAL PARAMETERS

PARAMETER	EXPRESSION/VALUE	DESCRIPTION
E	10 GPa	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.

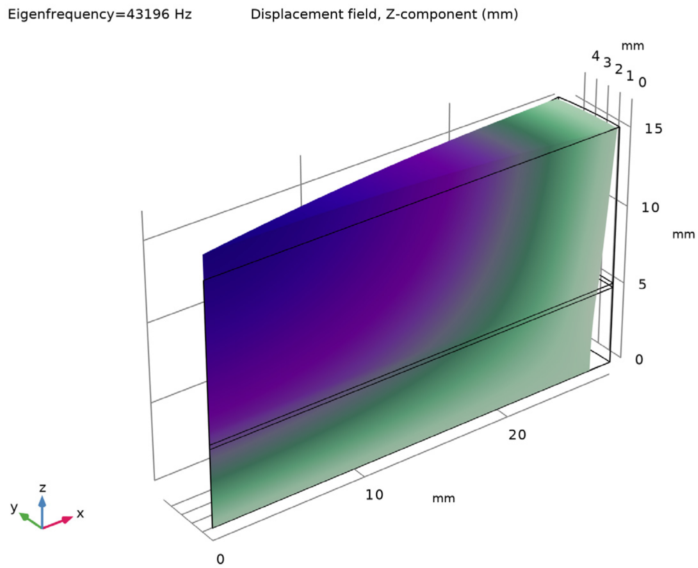


Figure 1: The lowest vibration eigenmode of the transducer.

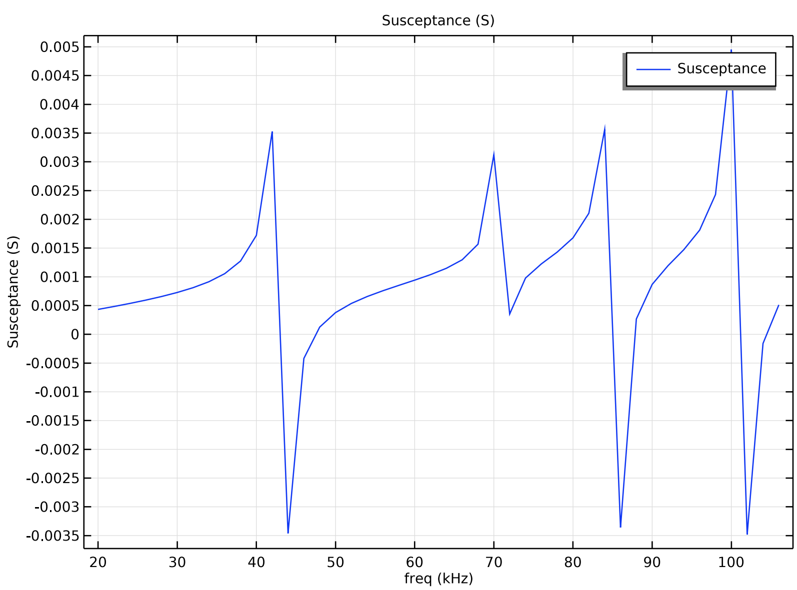


Figure 2: Input susceptance as a function of excitation frequency.

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.

Application Library path: `MEMS_Module/Piezoelectric_Devices/composite_transducer`

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 **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Eigenfrequency**.

6 Click **Done**.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

Work Plane 1 (wp1)

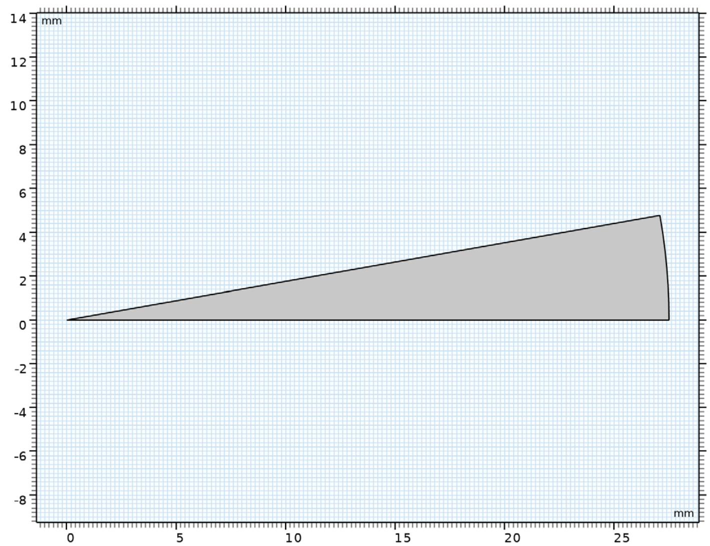
In the **Geometry** toolbar, click **Work Plane**.

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 `27.5`.
4. In the **Sector angle** text field, type `10`.
5. Click **Build Selected**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.



Extrude 1 (ext1)

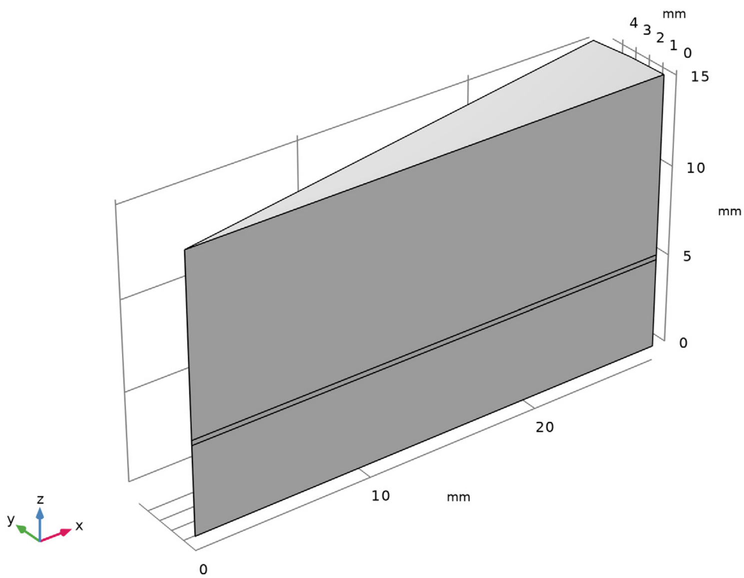
1. In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
2. In the **Settings** window for **Extrude**, locate the **Distances** section.
3. In the table, enter the following settings:

Distances (mm)
5
5.275
15.275

4. Click **Build All Objects**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

This completes the geometry modeling stage.



Before defining material properties, select the domains where each physics applies. Proceeding in this order enables to preselect required material properties during their definition.

SOLID MECHANICS (SOLID)

Piezoelectric Material 1

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

2 In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

ELECTROSTATICS (ES)

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

2 In the **Settings** window for **Electrostatics**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

Now materials can be defined.

MATERIALS

Nepec 6

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

2 In the **Settings** window for **Material**, type `Nepec 6` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domain 1 only.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
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	`{128[GPa], 68[GPa], 128[GPa], 66[GPa], 66[GPa], 110[GPa],0, 0,0, 21[GPa],0, 0,0,0, 21[GPa],0, 0,0,0,0, 21[GPa]}`	Pa	Stress-charge form
Coupling matrix	{eES11, eES21, eES31, eES12, eES22, eES32, eES13, eES23, eES33, eES14, eES24, eES34, eES15, eES25, eES35, eES16, eES26, eES36}	`{0,0,-6.1, 0,0,-6.1,0, 0,15.7,0,0, 0,0,0,0,0, 0,0}`	C/m²	Stress-charge form
Relative permittivity	epsilonrS_iso ; epsilonrSii = epsilonrS_iso, epsilonrSij = 0	`993.53`	1	Stress-charge form
Density	rho	7730	kg/m³	Basic

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 under **Component1**

> **Materials** > **Nepec 6** > **Stress-Charge** form in the Model builder and use the matrix input dialogs to enter the data as given in section NEPEC 6 Material Parameters.

Adhesive

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Adhesive` in the **Label** text field.
3. Select Domain 2 only.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`10[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.38`	1	Young's modulus and Poisson's ratio
Density	rho	`1700`	kg/m^3	Basic

Aluminum

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Aluminum` in the **Label** text field.
3. Select Domain 3 only.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`70.3[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.345`	1	Young's modulus and Poisson's ratio
Density	rho	`2690`	kg/m^3	Basic

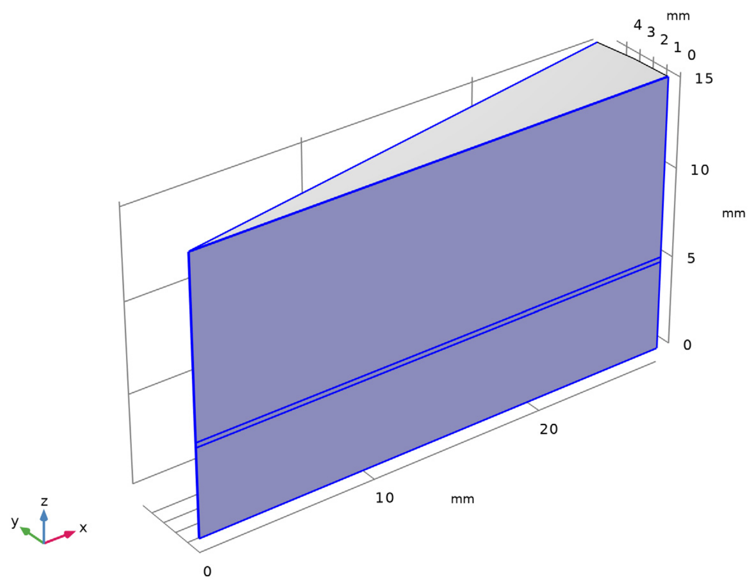
SOLID MECHANICS (SOLID)

Now apply the boundary conditions for each physics.

Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundaries 1–5, 7, and 8 only.

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



ELECTROSTATICS (ES)

Boundary Terminal 1

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

2 Select Boundary 6 only.

3 In the **Settings** window for **Boundary Terminal**, locate the **Terminal** section.

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

5 In the V_0 text 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 In the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundary 3 only.

DEFINITIONS

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

Variables 1

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

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

3 In the table, enter the following settings:

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

In the above expression, the factor 36 compensates for the fact that the total current at the **Terminal** is only computed for a 10 degree wedge of the full transducer. Moreover, the factor 1/2 accounts for the fact that only the upper half of the transducer is modeled because of symmetry in the *z* direction and hence only half of the actual voltage is applied. Since no damping is modeled, the real part of the admittance `es.Y11` will be zero. This is why it is suitable to evaluate only the imaginary part of the admittance, that is, the susceptance.

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**.

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 Select Boundary 3 only.

Distribution 1

1 Right-click **Free Triangular 1** and choose **Distribution**.

2 Select Edges 2 and 3 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

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

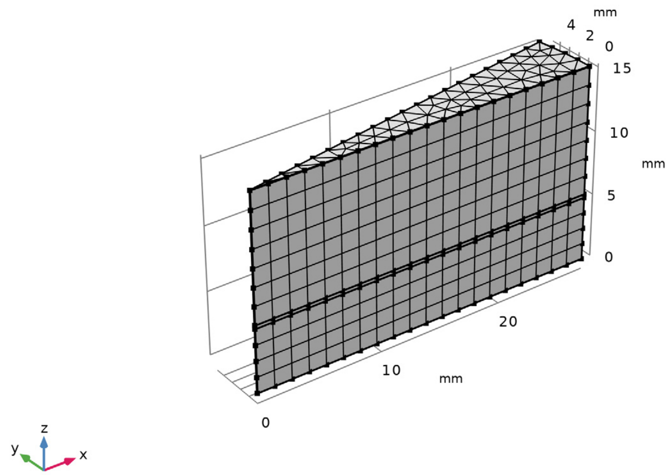
6 Click **Build Selected**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, click **Build All**.

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



STUDY 1

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Mode Shape (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Displacement** > **Displacement field - m** > **w - Displacement field, Z-component**.

3 In the **Mode Shape (solid)** toolbar, click **Plot**.

Compare the resulting plot to that in Figure 1.

Electric Potential (es)

In the **Model Builder** window, expand the **Electric Potential (es)** node.

Multislice 1, Streamline Multislice 1

1 In the **Model Builder** window, under **Results** > **Electric Potential (es)**, Ctrl-click to select **Multislice 1** and **Streamline Multislice 1**.

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

Surface 1

1 Right-click **Electric Potential (es)** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electrostatics** > **Electric** > **V - Electric potential - V**.

3 In the **Electric Potential (es)** toolbar, click **Plot**.

Next, add a separate study for the frequency sweep.

ADD STUDY

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

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

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

4 Click the **Add Study** button in the window toolbar.

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

STUDY 2

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 Click **Range**.

3 In the **Range** dialog, type `20[kHz]` in the **Start** text field.

4 In the **Stop** text field, type `106[kHz]`.

5 In the **Step** text field, type `2[kHz]`.

6 Click **Replace**.

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

RESULTS

Electric Potential (es) 1

In the **Model Builder** window, expand the **Electric Potential (es) 1** node.

Multislice 1, Streamline Multislice 1

1 In the **Model Builder** window, under **Results** > **Electric Potential (es) 1**, Ctrl-click to select **Multislice 1** and **Streamline Multislice 1**.

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

Surface 1

1 In the **Electric Potential (es) 1** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electrostatics** > **Electric** > **V - Electric potential - V**.

3 In the **Electric Potential (es) 1** toolbar, click **Plot**.

Displacement

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

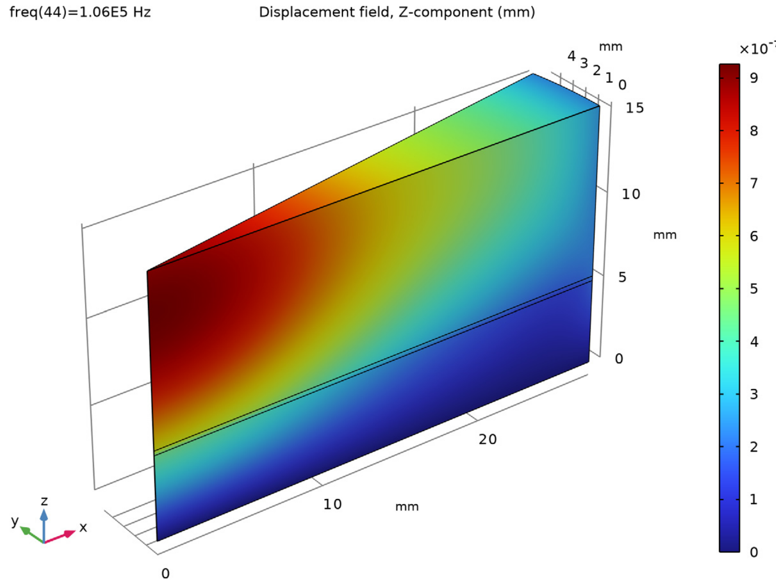
4 From the **Parameter value (freq (Hz))** list, choose **1.06E5**.

Surface 1

1 In the **Displacement** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Displacement** > **Displacement field - m** > **w - Displacement field, Z-component**.

3 In the **Displacement** toolbar, click **Plot**.



Susceptance

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

2 In the **Settings** window for **1D Plot Group**, type `Susceptance` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Global 1

1 Right-click **Susceptance** 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
B	S	Susceptance

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **kHz**.

5 In the **Susceptance** toolbar, click **Plot**.

Compare the result to that in Figure 2.