Piezoacoustic Transducer

Introduction

A piezoelectric transducer can be used either to transform an electric current to an acoustic pressure field or, the opposite, to produce an electric current from an acoustic field. These devices are generally useful for applications that require the generation of sound in air and liquids. Examples of such applications include phased array microphones, ultrasound equipment, inkjet droplet actuators, drug discovery, sonar transducers, bioimaging, and acousto-biotherapeutics.

Model Definition

In a phased-array microphone, the piezoelectric crystal plate fits into the structure through a series of stacked layers that are divided into rows. The space between these layers is referred to as the *kerf*, and the rows are repeated with a periodicity, or *pitch*.

This model simulates a single crystal plate in such a structure. The element is rotationally symmetric, making it possible to set up the model in axially symmetric 2D.

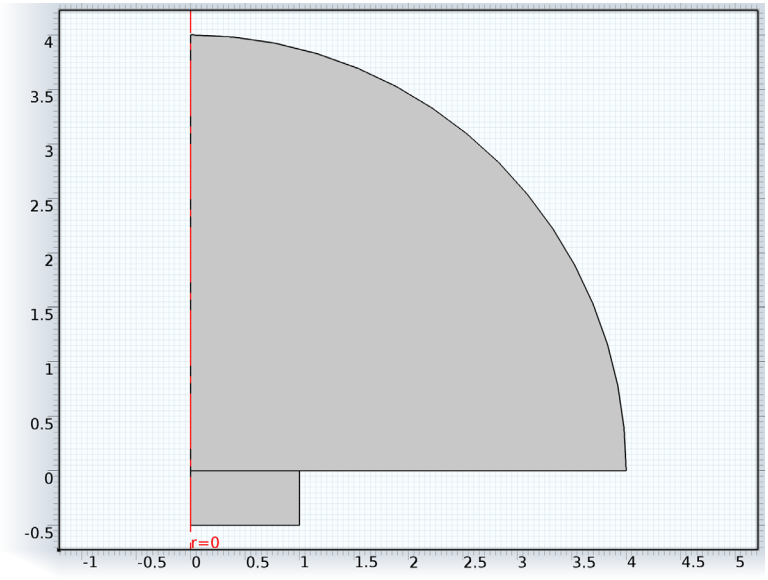


Figure 1: The model geometry.

In the air domain, the wave equation describes the pressure distribution:

$$\frac{1}{\rho_0 c_s^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) = Q \tag{1}$$

For this model, assume that the pressure varies harmonically in time as

$$p(\mathbf{x}, t) = p(\mathbf{x})e^{i\omega t}$$

Hence Equation 1 simplifies to

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{\omega^2 p}{\rho_0 c_s^2} = Q \tag{2}$$

Because there are no sources present, Equation 2 simplifies further to

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p)\right) - \frac{\omega^2 p}{\rho_0 c_s^2} = 0$$

The piezoelectric domain is made of the crystal PZT5-H, which is a common material in piezoelectric transducers. The structural analysis is also time harmonic although, for historical reasons, in structural-mechanics terminology it is a frequency response analysis.

The frequency is set to 200 kHz, which is in the ultrasonic range (dolphins and bats, for example, communicate in the range of 20 Hz to 150 kHz, while humans can only hear frequencies in the range 20 Hz to 20 kHz).

BOUNDARY CONDITIONS

An AC electric potential of 100 V is applied to the upper part of the transducer, while the bottom part is grounded. At the interface between the air and solid domain, the boundary condition for the acoustics interface is that the pressure is equal to the normal acceleration of the solid domain

$$n \cdot \left(\frac{1}{\rho_0}(\nabla p)\right) = a_n$$

where a_n is the normal acceleration.

This drives the pressure in the air domain. The solid domain is on the other hand subjected to the acoustic pressure changes in the air domain. Because of the high

 ©2012 COMSOL

voltage applied to the transducer, this load is probably negligible in comparison. Yet because the model is in 2D, it is possible to include this load and solve the full model simultaneously on any computer.

Results and Discussion

Figure 2 shows the pressure distribution in the air domain.

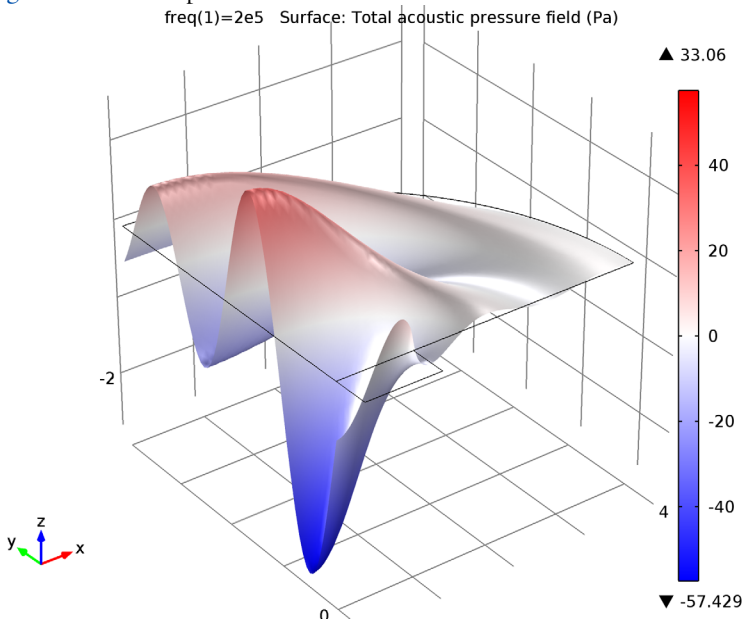


Figure 2: Surface and height plot of the pressure distribution.

Figure 3 shows the pressure distribution along the air-solid interface. The acoustic pressure load is small in comparison to the electrical load, which is plotted in Figure 4.

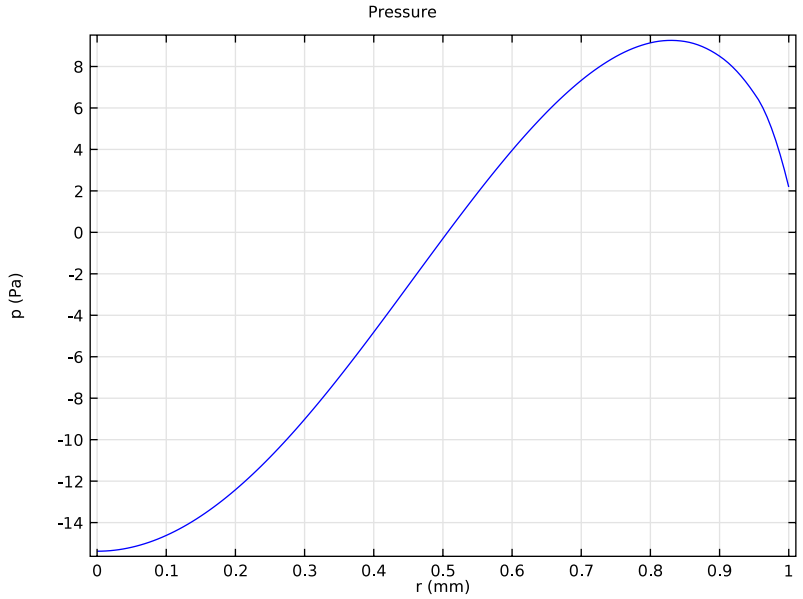


Figure 3: Acoustic pressure at the air-solid interface.

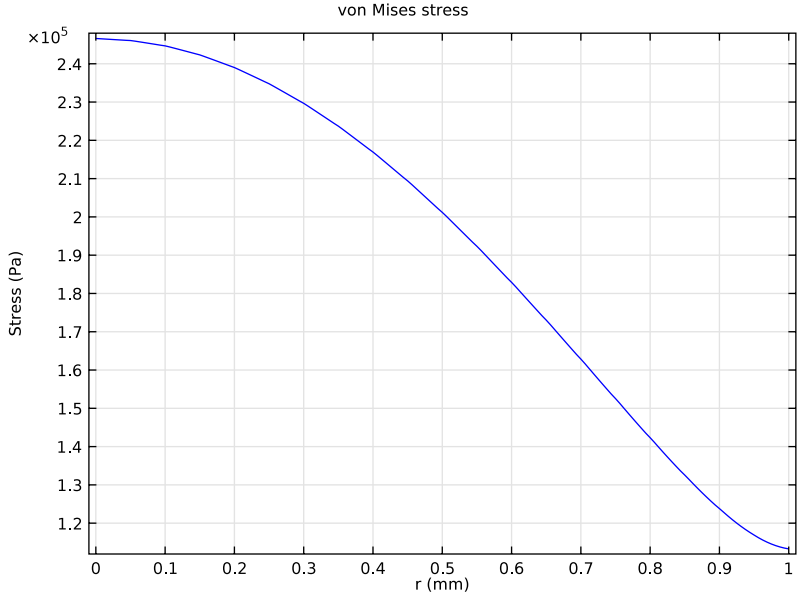


Figure 4: von Mises Stress along the air-solid interface.

 ©2012 COMSOL

The results from a far-field analysis appear in Figure 5. This figure shows that the sound pressure level reaches a maximum right in front of the transducer.

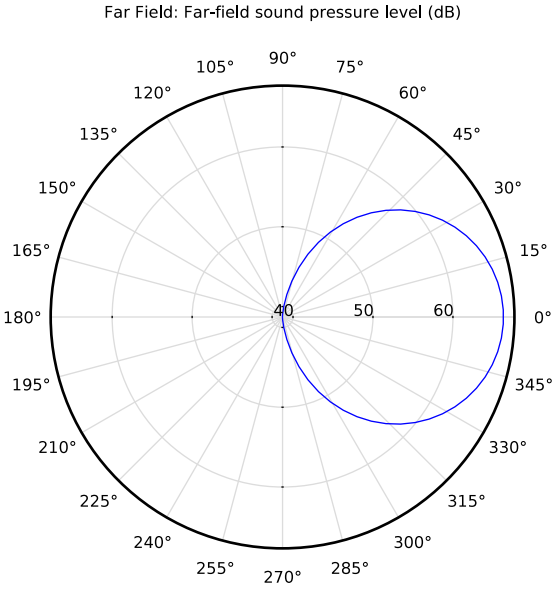


Figure 5: The far-field sound pressure level.

Model Library path: `Acoustics_Module/Tutorial_Models/piezoacoustic_transducer`

Modeling Instructions

MODEL WIZARD

1. Go to the **Model Wizard** window.
2. Click the **2D axisymmetric** button.
3. Click **Next**.
4. In the **Add physics** tree, select **Acoustics>Acoustic-Structure Interaction>Acoustic-Piezoelectric Interaction, Frequency Domain (acpz)**.
5. Click **Add Selected**.

6 Click **Next**.

7 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

8 Click **Finish**.

GEOMETRY 1

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

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

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

Begin by drawing the acoustic domain.

Circle 1

1 Right-click **Model 1>Geometry 1** and choose **Circle**.

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

3 In the **Radius** edit field, type `4`.

4 In the **Sector angle** edit field, type `90`.

Next, add the transducer, which is just a rectangle.

Rectangle 1

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

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Height** edit field, type `0.5`.

4 Locate the **Position** section. In the **z** edit field, type `-0.5`.

5 Click the **Build All** button.

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

MATERIALS

Material Browser

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

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Lead Zirconate Titanate (PZT-5H)**.

4 Right-click and choose **Add Material to Model** from the menu.

 ©2012 COMSOL

Lead Zirconate Titanate (PZT-5H)

1 In the **Model Builder** window, under **Model 1>Materials** click **Lead Zirconate Titanate (PZT-5H)**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

In the Piezoelectric Material Properties library, you can find more than 20 additional piezoelectric materials.

Note: For a piezoelectric material, you can specify the orientation by defining and selecting a new coordinate system. In this model, you will use the default Global coordinate system, which gives you a material oriented in the *rz*-plane.

Material Browser

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

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Air**.

4 Right-click and choose **Add Material to Model** from the menu.

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 Select Domain 2 only.

ACOUSTIC-PIEZOELECTRIC INTERACTION, FREQUENCY DOMAIN

You have only one interface in which to specify the physics. By default all the domain is defined as a Pressure acoustics domain. You must specify where is the Piezoelectric device.

Piezoelectric Material 1

1 In the **Model Builder** window, expand the **Acoustic-Piezoelectric Interaction, Frequency Domain** node, then click **Piezoelectric Material 1**.

2 Select Domain 1 only.

Roller 1

1 In the **Model Builder** window, right-click **Acoustic-Piezoelectric Interaction, Frequency Domain** and choose the boundary condition **Structural>Roller**.

2 Select Boundary 2 only.

Ground 1

1 Right-click **Acoustic-Piezoelectric Interaction, Frequency Domain** and choose the boundary condition **Electrical>Ground**.

2 Select Boundary 2 only.

Electric Potential 1

1 Right-click **Acoustic-Piezoelectric Interaction, Frequency Domain** and choose the boundary condition **Electrical>Electric Potential**.

2 Select Boundary 4 only.

3 In the **Electric Potential** settings window, locate the **Electric Potential** section.

4 In the V_0 edit field, type 100.

Acoustic-Structure Boundary 1

By default, the rest of the geometry is the pressure acoustics domain and the Acoustic-Structure boundary is available at the interface between pressure acoustics and piezoelectric domains. Add the wave radiation condition at the outer boundary.

Spherical Wave Radiation 1

1 Right-click **Acoustic-Piezoelectric Interaction, Frequency Domain** and choose the boundary condition **Acoustical>Spherical Wave Radiation**.

2 Select Boundary 7 only.

Finally, add the far-field calculation feature. This feature adds variables to evaluate the pressure and sound pressure level outside the computational domain.

Far-Field Calculation 1

1 Right-click **Acoustic-Piezoelectric Interaction, Frequency Domain** and choose the boundary condition **Acoustical>Far-Field Calculation**.

2 Select Boundary 7 only.

3 In the **Far-Field Calculation** settings window, locate the **Far-Field Calculation** section.

4 Select the **Symmetry in the z=0 plane** check box.

For more information on far-field calculation click the **Help** button the node's settings window or press F1.

MESH 1

It is important to use a mesh size sufficiently small to resolve the wavelength by at least 5–6 elements per wavelength. At 200 kHz, the wavelength in air is 1.7 mm. In the

 ©2012 COMSOL

piezo material, the presence of both pressure and shear waves makes it somewhat more difficult to define and compute. Because this is a small model, you can afford to use a very fine mesh.

Size

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Size** settings window, locate the **Element Size Parameters** section.

3 In the **Maximum element size** edit field, type `0.2`.

Size 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

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

4 Select Domain 1 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type `0.05`.

8 Click the **Build All** button.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Frequency Domain**.

2 In the **Frequency Domain** settings window, locate the **Study Settings** section.

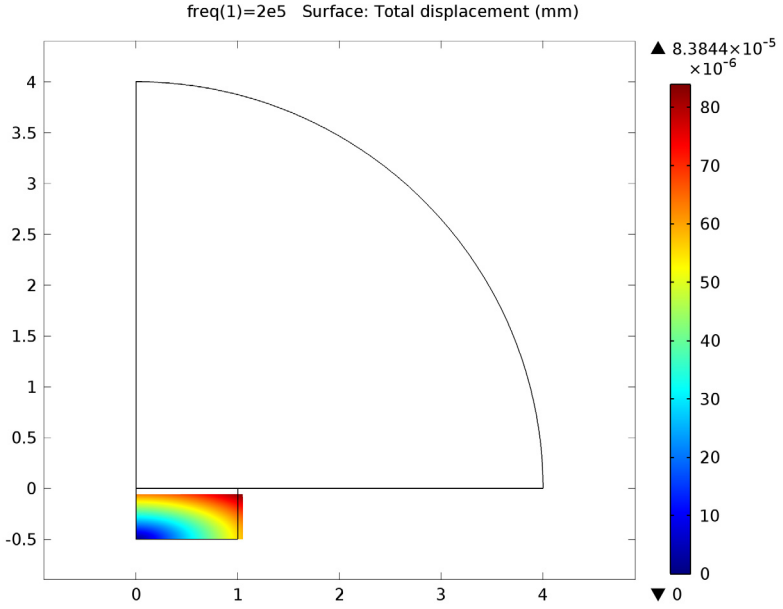
3 In the **Frequencies** edit field, type `200[kHz]`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Displacement (acpz)

The first default plot shows the radial displacement in the transducer.



Potential, 3D (acpz)

The second default plot shows the electric potential in a 3D revolved plot.

Acoustic Pressure (acpz)

The third default plot shows a surface and height plot of the pressure distribution which should be similar to that shown in Figure 2.

Far-Field Sound Pressure Level (acpz)

The fourth default plot shows the far-field sound pressure level. To reproduce Figure 5, you need to adjust the default settings. Note that 0 deg in the polar plot corresponds to the *z*-axis direction.

1. In the **Model Builder** window, expand the **Far-Field Sound Pressure Level (acpz)** node, then click **Far Field 1**.
2. In the **Far Field** settings window, locate the **Evaluation** section.
3. Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.
4. In the ϕ **start** edit field, type `-90`.

 ©2012 COMSOL

5 In the ϕ **range** edit field, type `180`.

6 Click the **Plot** button.

Next, create 1D plot groups to recreate Figure 3 and Figure 4.

1D Plot Group 5

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `von Mises stress`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated edit field, type `r (mm)`.

7 Select the **y-axis label** check box.

8 In the associated edit field, type `Stress (Pa)`.

9 Right-click **Results>1D Plot Group 5** and choose **Line Graph**.

10 In the **Line Graph** settings window, click **Activate Selection** in the upper-right corner of the **Selection** section. Select Boundary 4 only.

11 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Acoustic-Piezoelectric Interaction, Frequency Domain (Piezoelectric Devices)>Stress>von Mises stress (acpz.mises)**.

12 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Geometry and Mesh>Coordinate (Spatial)>r-coordinate (r)**.

13 Click the **Plot** button.

1D Plot Group 6

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Pressure`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated edit field, type `r (mm)`.

7 Select the **y-axis label** check box.

8 In the associated edit field, type `p (Pa)`.

9 Right-click **Results>1D Plot Group 6** and choose **Line Graph**.

10 In the **Line Graph** settings window, click **Activate Selection** in the upper-right corner of the **Selection** section. Select Boundary 4 only.

11 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Acoustic-Piezoelectric Interaction, Frequency Domain (Pressure Acoustics, Frequency Domain)>Total acoustic pressure field (acpz.p_t)**.

12 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **r-coordinate (r)**.

13 Click the **Plot** button.

 ©2012 COMSOL