Created in COMSOL Multiphysics 5.6

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 as a 2D axially symmetric problem.

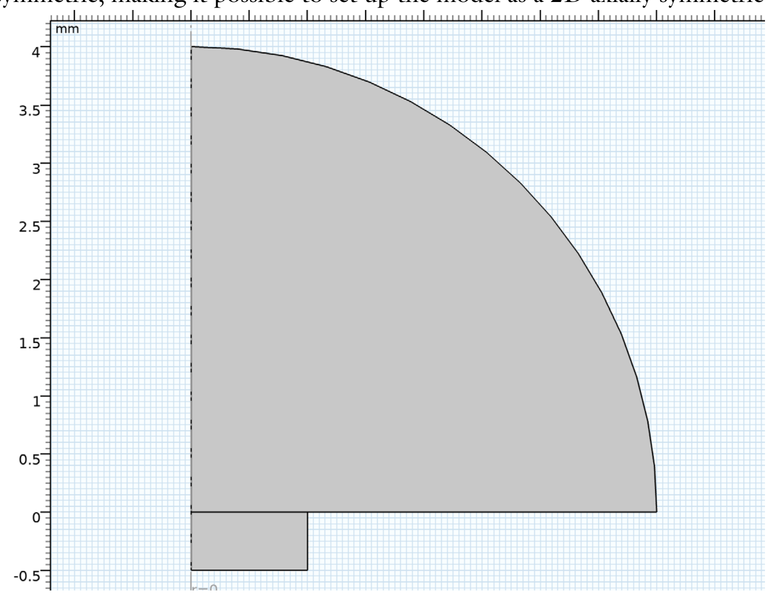


Figure 1: The model geometry.

PHYSICS IMPLEMENTATION IN DOMAINS

The model uses the built-in Acoustic-Piezoelectric Interaction, Frequency Domain multiphysics interface, which contains three fundamental physics interfaces: Pressure

Acoustics, Solid Mechanics, and Electrostatics. The first one solves for the wave equation in the fluid media surrounding the transducer. The latter two are used to model the piezoelectric effect.

In the air domain, in absence of volumetric sound sources, and assuming the pressure varies harmonically in time, the wave equation describing the acoustic pressure distribution is:

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

Equation 1 is solved by the Pressure Acoustics, Frequency Domain interface.

The piezoelectric domain is made of the material PZT-5H, which is a common material in piezoelectric transducers. The piezoelectric material is modeled by solving the Solid Mechanics and Electrostatics interfaces that are coupled via linear constitutive equations that correlate stresses and strains to electric displacement and electric field. These physics interfaces solve for the balance of body forces and volume charge density respectively as shown in Equation 2 and Equation 3.

$$\nabla \cdot \sigma = 0 \tag{2}$$

$$\nabla \cdot D = 0 \tag{3}$$

In COMSOL Multiphysics, this coupling is automatically implemented by the Piezoelectric Effect node located under the Multiphysics branch in the Model Builder.

The structural and electrical analyses are also time harmonic. For historical reasons, in structural-mechanics terminology it is called frequency response analysis, whereas in electrical engineering terminology it is called frequency domain analysis.

In this model, the excitation 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 from 20 Hz to 20 kHz).

BOUNDARY CONDITIONS

An AC electric potential of 100 V is applied to the upper surface of the transducer, and the bottom part is grounded. The bottom surface of the piezo transducer is assigned to a roller boundary condition which prevents it to move vertically, that is, along the *z*-direction.

At the interface between the air and solid domain, the normal component of the structural acceleration of the solid (piezo transducer) boundary is used to drive the air domain. This is described by the following equation:

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

where a_{n} is the normal acceleration.

The acoustic pressure at the interface between the air and solid domain acts as a boundary load on the solid,

$$n \cdot \sigma = p$$

The bidirectional coupling at the solid and air interface boundaries is automatically taken care of by the Acoustic-Structure boundary node located under the Multiphysics branch when you use the built-in Acoustic-Piezoelectric Interaction, Frequency Domain interface. The interface boundaries are automatically detected once you assign appropriate parts of the modeling geometry to the Pressure Acoustics, Frequency Domain and Solid Mechanics interfaces, respectively.

A Spherical Wave Radiation boundary condition is used on the outer surface of the air domain. This helps in implementing the idea that the air domain is infinitely extended in reality, and the spherical wavefronts travel outward from the geometric boundary that truncates the air domain with minimal reflection. Additionally, a Far Field Calculation is also set up on the same boundary, which helps to evaluate the pressure and sound pressure level in the far-field limit. Refer to the *Acoustics Module User's Guide* for more information on these boundary conditions.

Results and Discussion

Figure 2 shows the pressure distribution in the air domain.

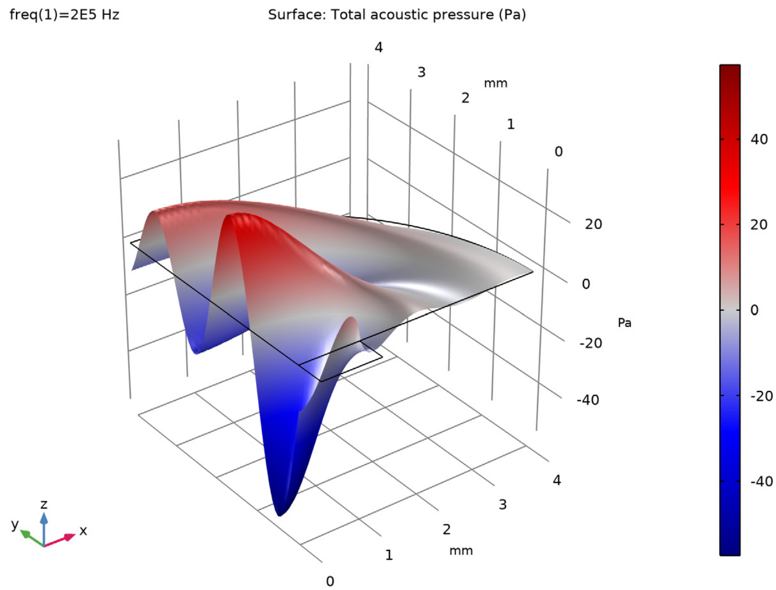


Figure 2: Surface and height plot of the pressure distribution.

Figure 3 shows the pressure distribution along the air-solid interface. Notice that the acoustic pressure is small in comparison to the mechanical stress, which is plotted in Figure 4.

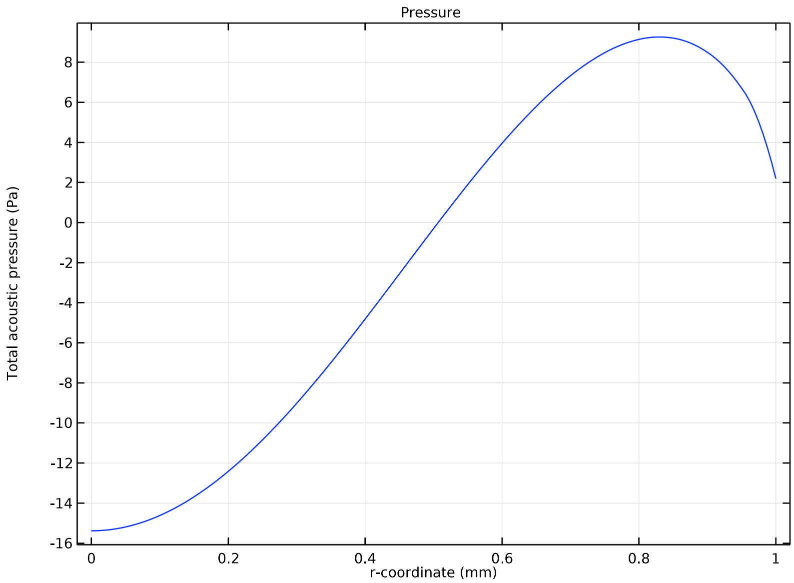


Figure 3: Acoustic pressure at the air-solid interface.

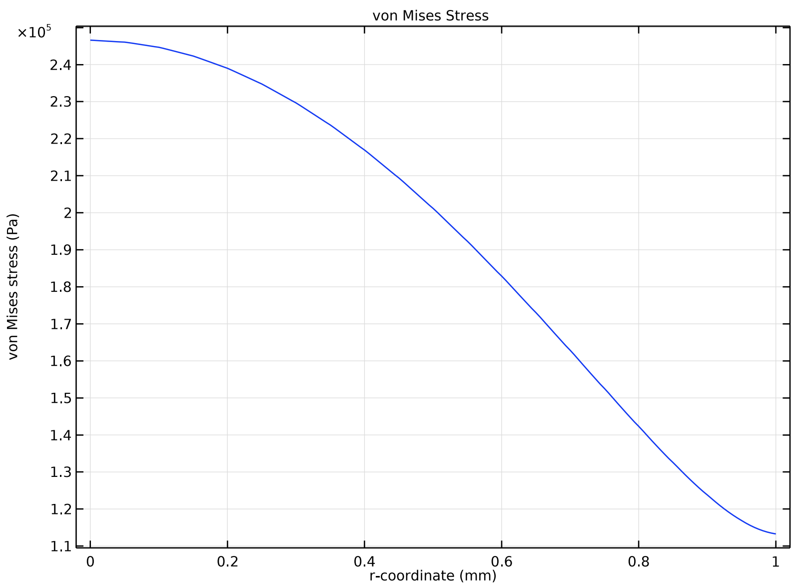


Figure 4: von Mises Stress along the air-solid interface.

The results from the far-field analysis are shown in Figure 5. This figure demonstrates that in the far-field limit, the sound pressure level reaches a maximum right in front of the transducer.

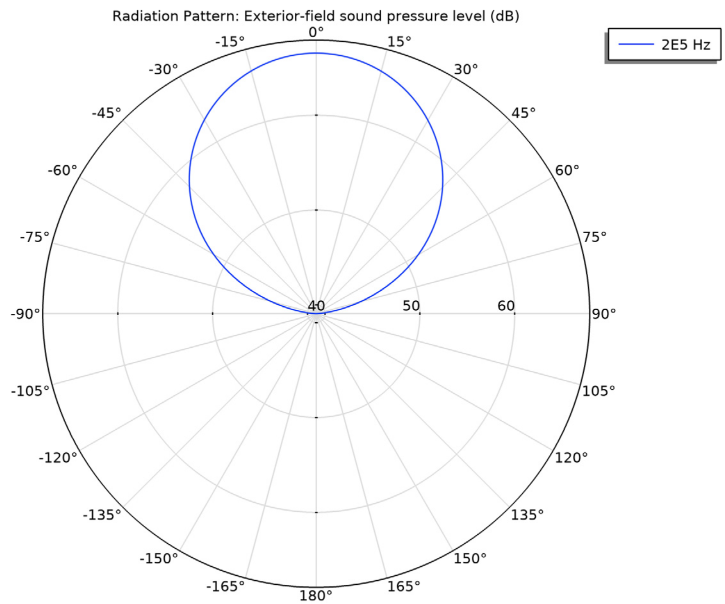


Figure 5: A polar plot of the far-field sound pressure level. The 0 degree axis coincides with the +z-direction of the rz-plane in the 2D axisymmetric model.

Application Library path: `Structural_Mechanics_Module/Acoustic-Structure_Interaction/piezoacoustic_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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Acoustics>Acoustic-Structure Interaction>Acoustic-Piezoelectric Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Frequency Domain**.

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**.

Begin by drawing the acoustic domain.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

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

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

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

5 Click **Build Selected**.

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

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

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

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

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

5 Click **Build All Objects**.

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

Before adding materials, select the domains related to each physics.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 Select Domain 2 only.

SOLID MECHANICS (SOLID)

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

2 Select Domain 1 only.

ELECTROSTATICS (ES)

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

2 Select Domain 1 only.

Define the Acoustic-Structure boundary.

MULTIPHYSICS

Acoustic-Structure Boundary 1 (asb1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Acoustic-Structure Boundary 1 (asb1)**.

2 Select Boundary 4 only.

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 **Built-in>Lead Zirconate Titanate (PZT-5H)**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat1)

Select Domain 1 only.

Note that in the Piezoelectric Material Properties library, you can find more than 20 additional piezoelectric materials. 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 that is poled along the z direction in the rz-plane.

ADD MATERIAL

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

2 In the tree, select **Built-in>Air**.

3 Click **Add to Component** in the window toolbar.

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

MATERIALS

Air (mat2)

Select Domain 2 only.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Add the Spherical Wave Radiation boundary condition on the outer boundary of the air domain.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

Spherical Wave Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Spherical Wave Radiation**.

2 Select Boundary 7 only.

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

Exterior Field Calculation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Exterior Field Calculation**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Exterior Field Calculation**, locate the **Exterior Field Calculation** section.

4 From the **Condition in the z = z^0 plane** list, choose **Symmetric/ Infinite sound hard boundary**.

5 From the **Type of integral** list, choose **Far-field integral approximation for r-> infinity**.

For more information on exterior-field calculation click the **Help** button in the toolbar or press **F1**.

SOLID MECHANICS (SOLID)

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

Roller 1

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

2 Select Boundary 2 only.

ELECTROSTATICS (ES)

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

Ground 1

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

2 Select Boundary 2 only.

Electric Potential 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type `100`.

MESH 1

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

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.2`.

Size 1

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

2 In the **Settings** window for **Size**, 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 text field, type `0.05`.

8 Click **Build All**.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

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

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

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

RESULTS

Acoustic Pressure (acpr)

The first default plot shows a surface plot of the pressure distribution. Change the color table and add a height plot in order to have a plot similar to that shown in Figure 2.

Height Expression 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node.

2 Right-click **Surface 1** and choose **Height Expression**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

Sound Pressure Level (acpr)

The second default plot shows the sound pressure level in the air domain.

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

The third and fourth plots are the 3D revolved plots of the acoustic pressure and the sound pressure level.

Exterior-Field Sound Pressure Level (acpr)

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

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Exterior-Field Sound Pressure Level (acpr)** node, then click **Radiation Pattern 1**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

3 Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.

4 In the ϕ **start** text field, type `-90`.

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

6 In the **Exterior-Field Sound Pressure Level (acpr)** toolbar, click **Plot**.

Exterior-Field Pressure (acpr)

Next, do the same with the sixth default plot, which is the exterior-field pressure.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Exterior-Field Pressure (acpr)** node, then click **Radiation Pattern 1**.
2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.
3 Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.
4 In the ϕ **start** text field, type `-90`.
5 In the ϕ **range** text field, type `180`.
6 In the **Exterior-Field Pressure (acpr)** toolbar, click **Plot**.

Add a plot of the displacement in the piezoelectric transducer.

Displacement

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Displacement` in the **Label** text field.

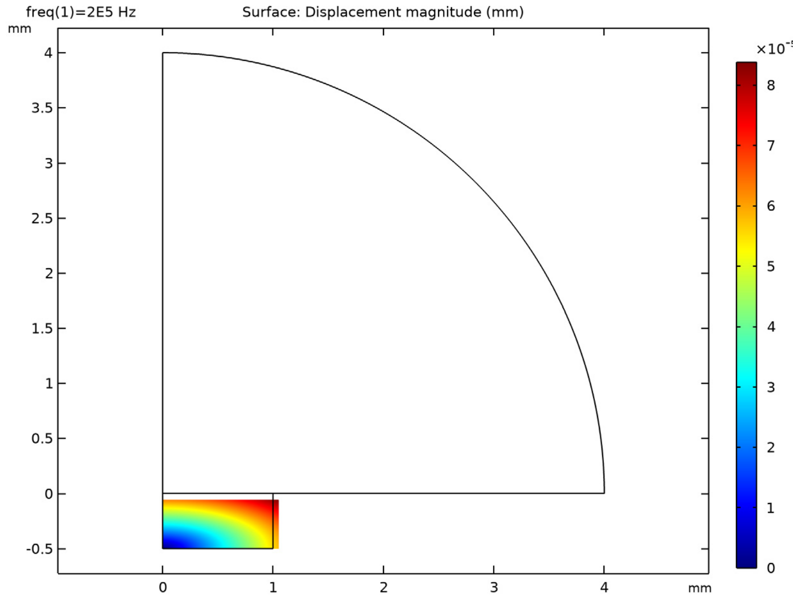
Surface 1

1 Right-click **Displacement** 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)>Solid Mechanics> Displacement>solid.disp - Displacement magnitude - m**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

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



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

von Mises Stress

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

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

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Line Graph 1

1 Right-click **von Mises Stress** and choose **Line Graph**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Stress>solid.mises - von Mises stress - N/m²**.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `r`.

6 In the **von Mises Stress** toolbar, click **Plot**.

Pressure

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

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

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

Line Graph 1

1 Right-click **Pressure** and choose **Line Graph**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

4 From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `r`.

6 In the **Pressure** toolbar, click **Plot**.