Created in COMSOL Multiphysics 6.1

Piezoacoustic Transducer

Introduction

A piezoelectric transducer can be used to either 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. In this tutorial a piezoelectric speaker, colloquially known as a "piezo buzzer" will be analyzed.

Model Definition

A piezo buzzer is a type of piezoelectric transducer that uses a piezoelectric crystal attached to a membrane to induce bending on the membrane and thus generate acoustic pressure. This type of transducers are typically driven at the resonance frequency, where the sound is efficiently radiated.

This model simulates a buzzer intended for ultrasonic applications. The membrane and crystal are rotationally symmetric, making it possible to set up the model as a 2D axisymmetric problem.

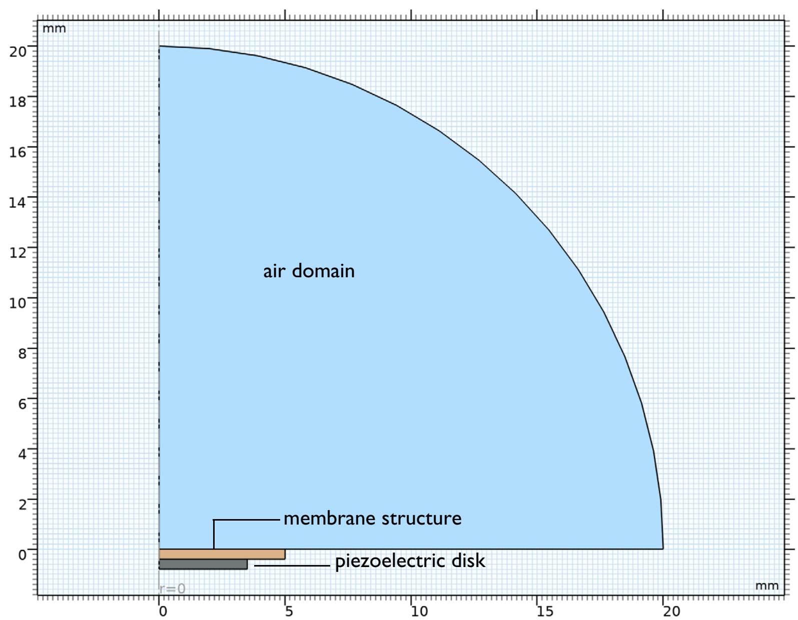


Figure 1: The model geometry.

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 and solids.

In the air domain the Helmholtz equation, that described the distribution of pressure, is solved. 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 1 and Equation 2.

$$\nabla \cdot \sigma = 0 \tag{1}$$

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

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 swept from 10 kHz to 60 kHz, which partially overlaps with 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). This sweep helps identify the resonance frequency at which the transducer is likely to be excited.

BOUNDARY CONDITIONS

An AC electric potential of 5 V is applied to the upper surface of the crystal, and the bottom part is grounded. The crystal is attached to a brass membrane which is fixed at its outer edge.

At the interface between the air and solid domain, the normal component of the structural acceleration of the solid (brass membrane) boundary is used to drive the air domain, while the acoustic pressure at the interface acts as a boundary load on the solid. 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 in the Model Builder. 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.

The *Perfectly Matched Boundary* condition is used on the outer surface of the air domain. It is used to model an open infinitely extended domain. The wavefronts travel outward from the geometric boundary that truncates the air domain with minimal reflection. Additionally, an *Exterior Field Calculation* feature is also set up on the same boundary, it is used to evaluate the pressure and sound pressure level in points exterior to the computational domain. 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 at 60 kHz.

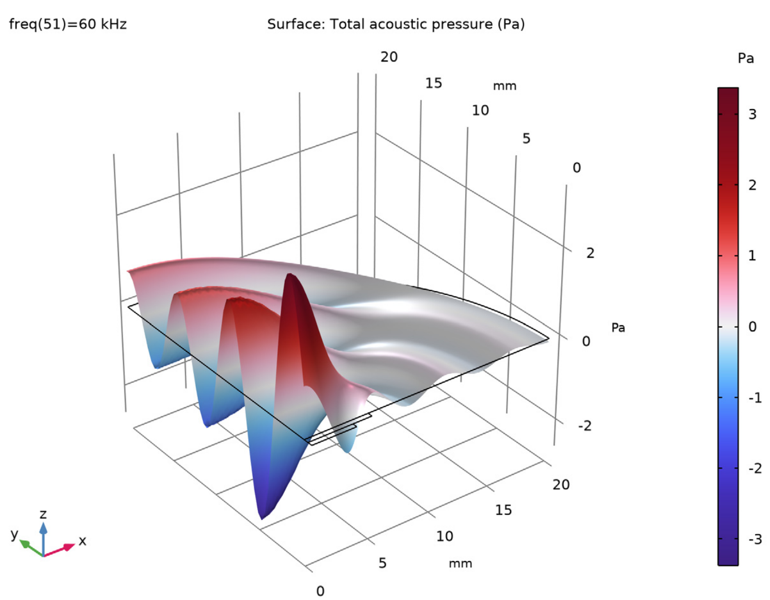


Figure 2: Surface and height plot of the pressure distribution.

Figure 3 shows the on axis sound pressure level at 1 m as a function of the driving frequency. The clear resonance around 33 kHz indicates that this would be the frequency that will be used to drive the transducer.

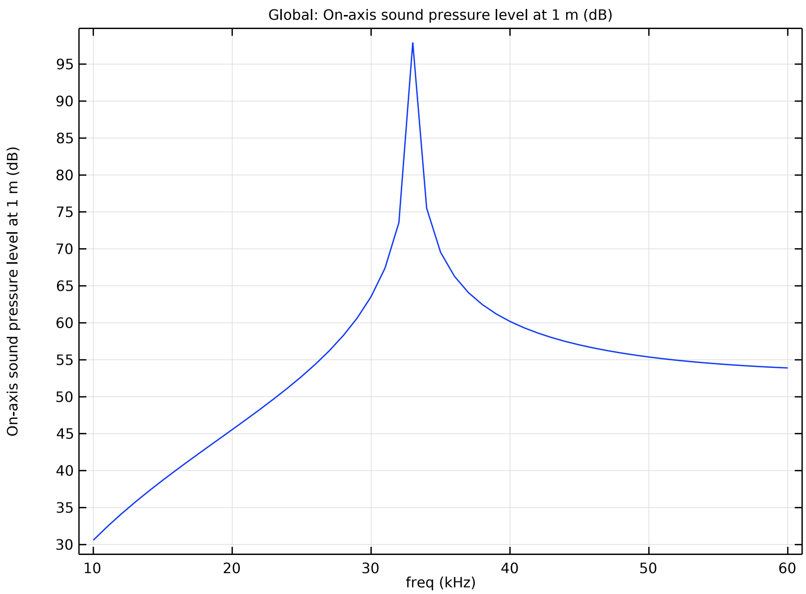


Figure 3: On axis sound pressure level 1 m in front of the transducer.

Figure 4 shows the pressure distribution at the top surface of the brass membrane. Notice that the acoustic pressure is very small in comparison to the mechanical von Mises stress, which is plotted in Figure 5.

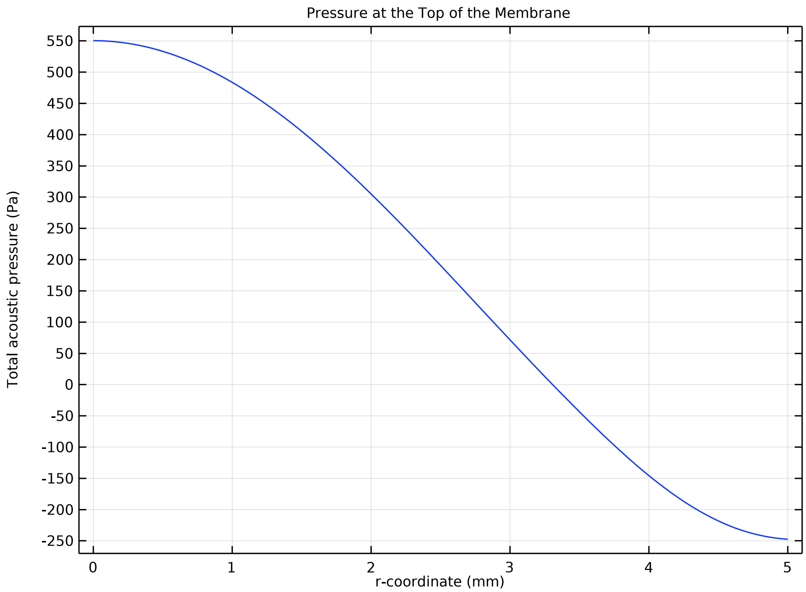


Figure 4: Acoustic pressure at the top of the brass membrane.

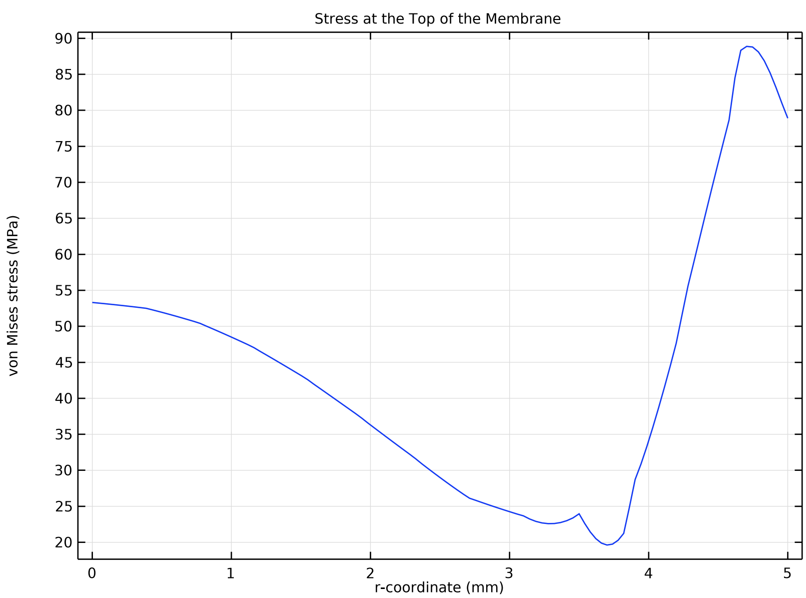


Figure 5: von Mises Stress along the air-solid interface.

The polar plot of the sound pressure level at 1 m from the transducer at different frequencies is shown in Figure 6. Note how the radiated sound pressure level is higher at the resonance frequency and how the complexity and directionality increase as the frequency increases.

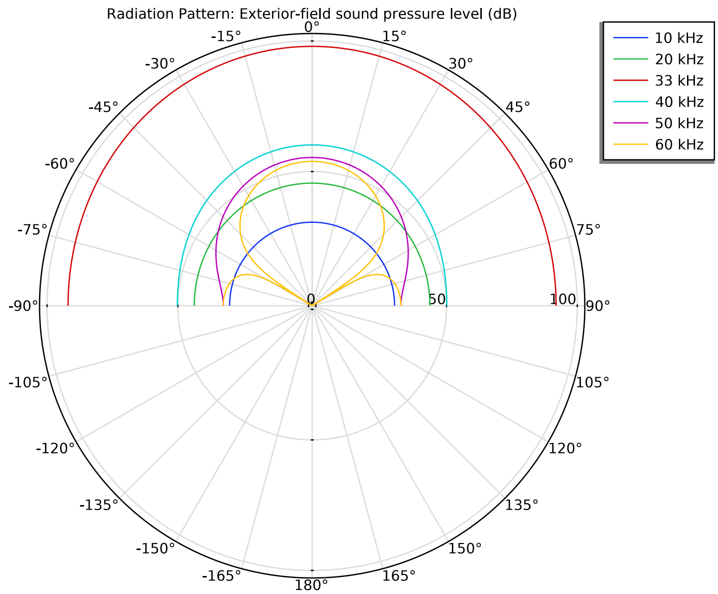


Figure 6: A polar plot of the exterior-field sound pressure level at 1 m. The 0 degree axis coincides with the +z direction of the rz-plane in the 2D axisymmetric model.

The effect of the frequency on the directionality of the transducer can also be seen in Figure 7, which shows the angular width of the source. The beamwidth is defined as the angular arc in front of the transducer where the sound pressure level is above -3 dB from the value in front of the transducer.

The displacement of the membrane at the resonance frequency can be seen in Figure 8.

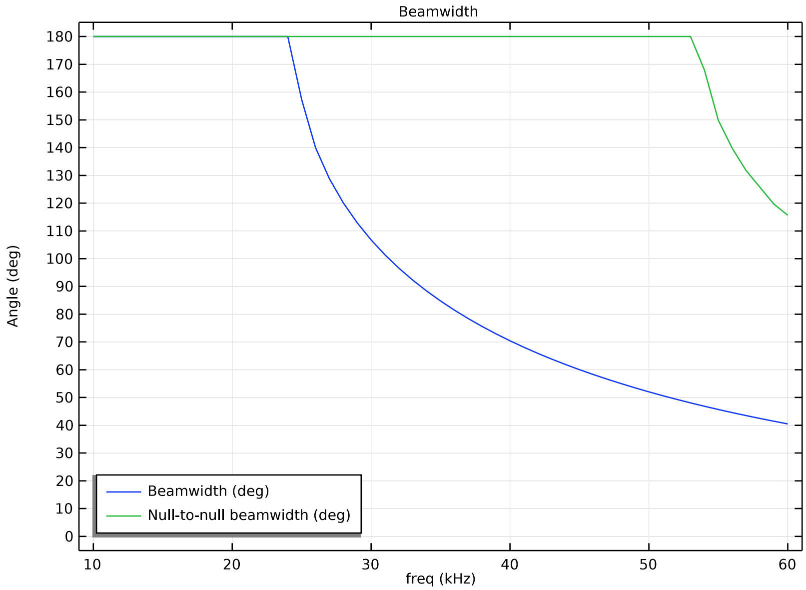


Figure 7: Beamwidth and null-to-null beam width as a function of the frequency.

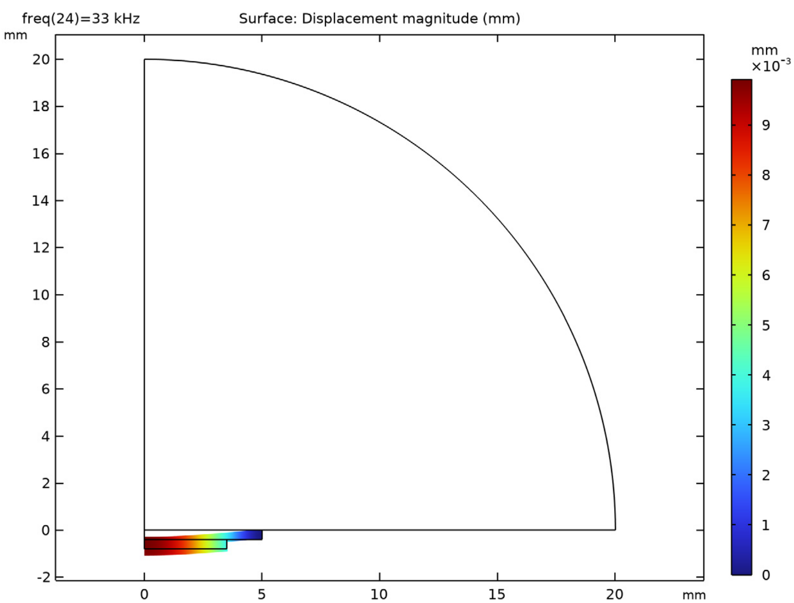


Figure 8: Displacement of the membrane at the resonance frequency.

Application Library path: `Acoustics_Module/Piezoelectric_Devices/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**.

Begin by importing some parameters that will be used during the model definition.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `piezoacoustic_transducer_parameters.txt`.

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**.

Start by drawing the acoustic domain.

Air

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

2 In the **Settings** window for **Circle**, type `Air` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_air`.

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

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

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 177, 221, and 255, respectively.

9 Click **Add to custom colors**.

10 Click **Show color palette only** or **OK** on the cross-platform desktop.

11 Click **Build Selected**.

Next, add the brass membrane, which is just a rectangle.

Brass Membrane

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

2 In the **Settings** window for **Rectangle**, type `Brass Membrane` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `r_memb`.

4 In the **Height** text field, type `th_memb`.

5 Locate the **Position** section. In the **z** text field, type `-th_memb`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 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.

8 Click **Define custom colors**.

9 Set the RGB values to 219, 179, and 135, respectively.

10 Click **Add to custom colors**.

11 Click **Show color palette only** or **OK** on the cross-platform desktop.

12 Find the **Cumulative selection** subsection. Click **New**.

13 In the **New Cumulative Selection** dialog box, type `Structural Domains` in the **Name** text field.

14 Click **OK**.

15 In the **Settings** window for **Rectangle**, click **Build Selected**.

Now draw the piezoelectric material.

Piezoelectric material

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

2 In the **Settings** window for **Rectangle**, type `Piezoelectric material` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `r_pzt`.

4 In the **Height** text field, type `th_pzt`.

5 Locate the **Position** section. In the **z** text field, type `-th_memb-th_pzt`.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 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.

8 Click **Define custom colors**.

9 Set the RGB values to 114, 119, and 122, respectively.

10 Click **Add to custom colors**.

11 Click **Show color palette only** or **OK** on the cross-platform desktop.

12 Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Structural Domains**.

13 Click **Build Selected**.

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

The geometry should look like the one in Figure 1.

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 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

SOLID MECHANICS (SOLID)

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

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

3 From the **Selection** list, choose **Structural Domains**.

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 From the **Selection** list, choose **Piezoelectric material**.

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 From the **Selection** list, choose **Piezoelectric material**.

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)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Piezoelectric material**.

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)

1. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2. From the **Selection** list, choose **Air**.

Brass

1. In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Brass` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Brass Membrane**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`100[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	`8900[kg/m^3]`	kg/m³	Basic

Add Atmosphere attenuation with standard temperature, pressure and humidity values. This effect should be taken into account given the frequency range and the distance for far-field evaluation.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Pressure Acoustics 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)** click **Pressure Acoustics 1**.
2. In the **Settings** window for **Pressure Acoustics**, locate the **Pressure Acoustics Model** section.
3. From the **Fluid model** list, choose **Atmosphere attenuation**.
4. Locate the **Model Input** section. In the ϕ_w text field, type `0.5`.

Add the Perfectly Matched Boundary condition on the outer boundary of the air domain.

Perfectly Matched Boundary 1

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

2 Select Boundary 11 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 11 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**.

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)**.

Fixed Constraint 1

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

2 Select Boundary 9 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 `V0`.

MESH

Proceed and generate the mesh based on the **Physics-controlled mesh** suggestion for Pressure Acoustics, Frequency Domain. This is done by only selecting Pressure Acoustics, Frequency Domain as **Contributor** and then switching to **User-controlled mesh** on the main mesh node. The frequency controlling the maximum element size is per default taken **From study**. Set the desired **Frequencies** in the study step. In general, 5 to 6 second-order elements per wavelength are needed to resolve the waves. For more details, see *Meshing (Resolving the Waves)* in the *Acoustics Module User's Guide*. In this model we use the default **Automatic** option, which gives 5 elements per wavelength. A **Mapped** mesh is then added in the structural domain to resolve the thin geometry.

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 From the **Frequency unit** list, choose **kHz**.

4 Click **Range**.

5 In the **Range** dialog box, type `fmin` in the **Start** text field.

6 In the **Step** text field, type `fstep`.

7 In the **Stop** text field, type `fmax`.

8 Click **Replace**.

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 In the table, clear the **Use** check boxes for **Solid Mechanics (solid)**, **Electrostatics (es)**, **Acoustic-Structure Boundary 1 (asb1)**, and **Piezoelectric Effect 1 (pze1)**.

4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Mapped 1

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

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

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

4 From the **Selection** list, choose **Structural Domains**.

5 Click to expand the **Reduce Element Skewness** section. Select the **Adjust edge mesh** check box.

Size 1

1 Right-click **Mapped 1** and choose **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.
5 Select the **Maximum element size** check box. In the associated text field, type `min(th_memb,th_pzt)`.

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 7 and 9 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type 2.

Proceed and move the **Mapped** mesh above **Free Triangular** in the **Model Builder** tree to resolve the structural domain correctly.

Mapped 1

1 In the **Model Builder** window, click **Mapped 1**.
2 Drag and drop below **Size Expression 1**.
3 In the **Settings** window for **Mapped**, click **Build All**.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, locate the **Study Settings** section.
3 Clear the **Generate default plots** check box.
4 In the **Home** toolbar, click **Compute**.

The first predefined plot shows a surface plot of the pressure distribution. Add a height plot in order to have a plot similar to that shown in Figure 2.

RESULTS

Acoustic Pressure (acpr)

In the **Model Builder** window, right-click **Results** and choose **Add Predefined Plot>Study 1/Solution 1 (sol1)>Pressure Acoustics, Frequency Domain>Acoustic Pressure (acpr)**.

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.

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

Sound Pressure Level (acpr)

1 In the **Model Builder** window, right-click **Results** and choose **Add Predefined Plot>Study 1/ Solution 1 (sol1)>Pressure Acoustics, Frequency Domain>Sound Pressure Level (acpr)**.
2 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.

Acoustic Pressure, 3D (acpr)

Right-click **Results** and choose **Add Predefined Plot>Study 1/Solution 1 (sol1)> Pressure Acoustics, Frequency Domain>Acoustic Pressure, 3D (acpr)**.

Sound Pressure Level, 3D (acpr)

Right-click **Results** and choose **Add Predefined Plot>Study 1/Solution 1 (sol1)> Pressure Acoustics, Frequency Domain>Sound Pressure Level, 3D (acpr)**.

The fifth predefined plot shows the exterior-field sound pressure level. To reproduce Figure 6, you need to adjust the default settings. Note that 0 degree in the polar plot corresponds to the z-axis direction.

Exterior-Field Sound Pressure Level - Selected Frequencies

1 Right-click **Results** and choose **Add Predefined Plot>Study 1/Solution 1 (sol1)> Pressure Acoustics, Frequency Domain>Exterior-Field Sound Pressure Level (acpr)**.
2 In the **Settings** window for **Polar Plot Group**, type `Exterior-Field Sound Pressure Level - Selected Frequencies` in the **Label** text field.
3 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **Manual**.
4 In the **Parameter indices (1-51)** text field, type `1 11 24 31 41 51`.
5 In the **Exterior-Field Sound Pressure Level - Selected Frequencies** toolbar, click **Plot**.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Exterior-Field Sound Pressure Level - Selected Frequencies** 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 Find the **Evaluation distance** subsection. In the **Radius** text field, type `1[m]`.

7 In the **Exterior-Field Sound Pressure Level - Selected Frequencies** toolbar, click **Plot**.

Exterior-Field Sound Pressure Level - All Frequencies

1 In the **Model Builder** window, right-click **Exterior-Field Sound Pressure Level - Selected Frequencies** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Exterior-Field Sound Pressure Level - All Frequencies` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **All**.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Exterior-Field Sound Pressure Level - All Frequencies** 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 **Compute beamwidth** list, choose **On**.

4 In the **Level down** text field, type `3`.

5 In the **Exterior-Field Sound Pressure Level - All Frequencies** toolbar, click **Plot**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 7** click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

Beamwidth

1 In the **Model Builder** window, under **Results** click **1D Plot Group 7**.

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

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

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Angle (deg)`.

6 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

7 In the **Beamwidth** toolbar, click **Plot**.

Annotation 1

1 In the **Model Builder** window, right-click **Beamwidth** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Data** section.

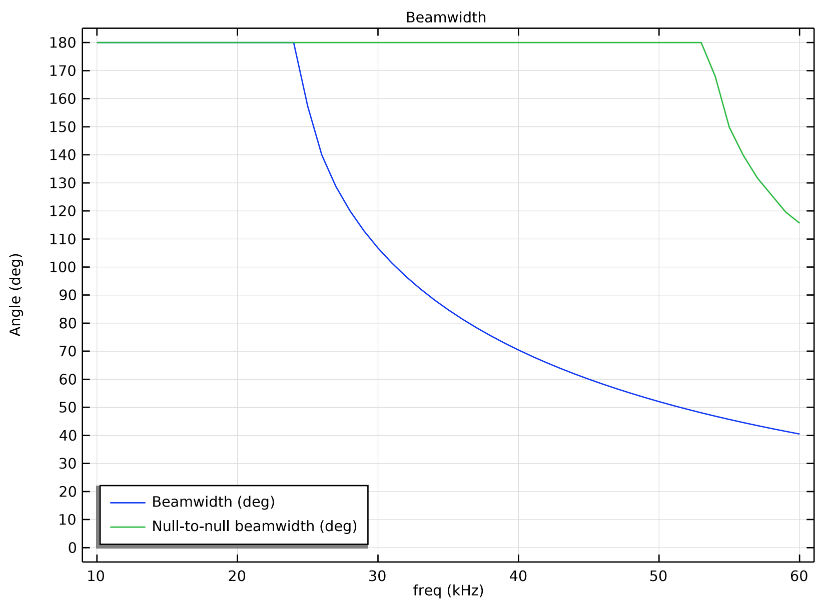
3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **Position** section. In the **r** text field, type 10.

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

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

The image should look like the one in Figure 7.



Add a plot of the displacement in the piezoelectric transducer.

Displacement (solid)

1 In the **Model Builder** window, right-click **Results** and choose **Add Predefined Plot>Study 1/Solution 1 (sol1)>Solid Mechanics>Displacement (solid)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (kHz))** list, choose **33**.

4 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.

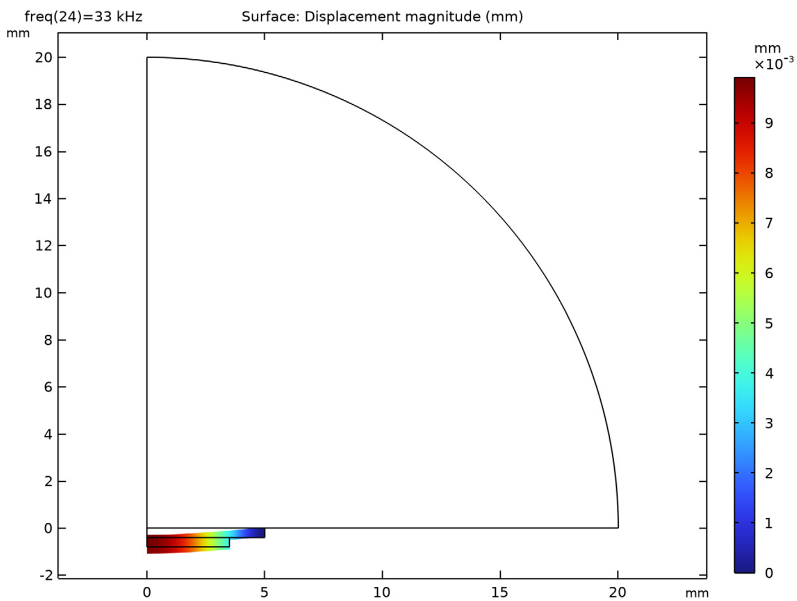
5 Click **OK**.

Surface 1

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

2 In the **Displacement (solid)** toolbar, click **Plot**.

The image should look like the one in Figure 8.



Next, create 1D plot groups to recreate Figure 4 and Figure 5.

Stress at the Top of the Membrane

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

2 In the **Settings** window for **1D Plot Group**, type `Stress at the Top of the Membrane` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **From list**.

4 In the **Parameter values (freq (kHz))** list, select **33**.

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

Line Graph 1

1. Right-click **Stress at the Top of the Membrane** and choose **Line Graph**.
2. Select Boundary 6 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 **y-Axis Data** section. From the **Unit** list, choose **MPa**.
5. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `r`.
7. In the **Stress at the Top of the Membrane** toolbar, click **Plot**.

The image should look like the one in Figure 5.

Pressure at the Top of the Membrane

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 at the Top of the Membrane` in the **Label** text field.
3. Locate the **Data** section. From the **Parameter selection (freq)** list, choose **From list**.
4. In the **Parameter values (freq (kHz))** list, select **33**.
5. Locate the **Title** section. From the **Title type** list, choose **Label**.

Line Graph 1

1. Right-click **Pressure at the Top of the Membrane** and choose **Line Graph**.
2. Select Boundary 6 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 at the Top of the Membrane** toolbar, click **Plot**.

The image should look like the one in Figure 4.

On-axis Sound Pressure Level at 1 m

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `On-axis Sound Pressure Level at 1 m` in the **Label** text field.

Global 1

1. Right-click **On-axis Sound Pressure Level at 1 m** 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
`subst(acpr.efc1.Lp_pext,r,0,z,1[m])`	`dB`	`On-axis sound pressure level at 1 m`

4 Click to expand the **Legends** section. Clear the **Show legends** check box.

5 In the **On-axis Sound Pressure Level at 1 m** toolbar, click **Plot**.

The image should look like the one in Figure 3.